[Selecting PCB materials for high-frequency applications]

John Coonrod, Rogers Corporation - July 11, 2012

Choosing a circuit material for a high-frequency printed-circuit board (PCB) is generally a tradeoff, often between price and performance. But PCB materials are also selected by two key factors: how well they meet the needs of an end-use application and what kind of effort is required to fabricate a desired circuit with a particular material.

These two factors may not mesh: one material may be well suited for a particular application but may pose challenges in terms of circuit fabrication, and vice versa. There is no foolproof, step-b--step procedure for selecting a PCB material. But by relying on some tangible guidelines designed to evaluate a material in terms of its suitability for circuit fabrication and for meeting the requirements of an application, the process of selecting a PCB for a particular application can be simplified. The approach will be demonstrated with some of the more popular high-frequency PCB materials, and where each stands in terms of fabrication qualities and suitability for end-applications.

Commercial high-frequency PCB materials can be categorized as one of seven generic material types, as shown in Table 1. High-performance FR-4 is included in Table 1 because it is often used in combination with other high-frequency materials for certain applications and requirements. However, in terms of electrical performance, FR-4 is not considered a true high-frequency circuit material.

Material	Ease of Circuit Fabrication	Electrical Performance
PTFE with micro glass fiber	Difficult	Excellent
PTFE with woven glass	Difficult	Good
Ceramic-filled PTFE	Moderate	Excellent
Ceramic-filled PTFE with woven glass	Moderate	Good
Ceramic-filled Hydrocarbon	Difficult	Good
Ceramic-filled Hydrocarbon with woven glass	Easy	Good
High Performance FR-4	Easy	Poor

Table 1: Typical circuit material types used in the high-frequency PCB industry.

Choosing materials based on circuit fabrication issues
A number of different mechanical processes are required as part of high-frequency PCB fabrication. In general, the most critical of these would be drilling, plated-through-hole (PTH) preparation, multilayer lamination, and assembly. The drilling process is typically concerned with creating clean holes, which will later be metalized to form viaholes for electrical connections from one conductive

layer to another.

Some concerns with the drilling process include smear, burring, and fracturing of the material. Smearing can be lethal to PCB fabrication using a PTFE based material, since there is no way to remove the smear. Fracturing can be fatal for some of the nonwoven glass hydrocarbon materials; however, most of the woven glass hydrocarbon materials do not have this concern.

The PTH preparation process is relatively well defined and straightforward for most non-PTFE materials, although special processing is required when forming PTHs for PTFE-based materials. Ceramic-filled PTFE-based materials offer PTH preparation options which are more forgiving. However, non-ceramic-filled PTFE materials require a special process which can limit final circuit yields.

Fabricating multilayer PCBs presents many challenges. One is the fact that dissimilar materials are often being bonded together, and these dissimilar materials can have properties which complicate drilling and PTH preparation processes. Also, a mismatch between certain material properties, such as coefficient of thermal expansion (CTE), can lead to reliability problems when the circuit is thermally stressed during assembly. A goal of the material selection process is to find a good combination of circuit materials for a multilayer PCB which enable practical fabrication processing while also meeting end-use requirements.

Designers and fabricators have many choices of materials used to bond together the copper-clad laminates that ultimately form a multilayer PCB. As Table 2 shows, the materials differ in terms of dielectric constant, dissipation factor, and processing temperatures. In general, lower lamination temperatures are to be preferred. But if a PCB must undergo soldering or some other form of thermal exposure, it will be necessary to use a bonding material with high reflow (re-melt) temperature, one which is thermally robust and does not reflow at the elevated processing temperatures.

Bonding Material	Dielectric Constant	Dissipation Factor	Lamination Temperature (°F)	Preparation for PTH	Re-melt Temperature (°F)
FEP	2.10	0.0010	565	Special	520
Ceramic-filled PTFE,Dk=3	3.00	0.0013	700	Special	640
Ceramic-filled PTFE,Dk=6	6.15	0.0020	700	Special	640
LCP	2.90	0.0025	554	Special	520
3001	2.30	0.0030	425	Special	350
Ceramic-filled PTFE,Dk=10	10.80	0.0023	700	Special	640
Thermoset Hydrocarbon	3.90	0.0040	350	Standard	N/A
FR-4	4.50	0.0180	360	Standard	N/A

Table 2: Bonding materials used for fabricating high-frequency multilayer PCBs.

Choosing materials based on circuit fabrication issues (cont.)

The greatest concern during PCB assembly is due to the effects of thermal stress from soldering. Other sources of thermal stress during PCB assembly are from solder rework or exposure to multiple thermal cycles. In terms of circuit materials, effects from thermal stress can typically be projected by comparing the CTE values for different materials, as shown in Table 3.

Material	CTE (ppm/°C)	Electrical Performance
PTFE with micro glass fiber	220	Excellent
PTFE with woven glass	200	Good
Ceramic-filled PTFE	25	Excellent
Ceramic-filled PTFE with woven glass	50	Good
Ceramic-filled Hydrocarbon	20	Good
Ceramic-filled Hydrocarbon with woven glass	35	Good
High Performance FR-4	50	Poor

Table 3: Typical CTE values for materials commonly used in high frequency PCBs.

In general, a circuit material with a lower CTE will be more robust and handle the thermal stress of PCB assembly better than a material with a higher overall CTE. This is one reason why multilayer PCBs typically use more than one type of circuit material. Materials which might provide good electrical performance may have characteristics (such as high CTE) that make them less than robust to handle the thermal stress of PCB assembly.

By using a combination of materials, some with good electrical properties and others with lower overall CTE, a robust multilayer PCB construction can be designed and assembled. Such a construction is known as a hybrid multilayer PCB, which can provide cost as well as performance benefits. More information about hybrid multilayer PCBs can be found on a paper presented at PCB West 2010. [1]

In general, a circuit material with CTE value of 70 ppm/°C or less is considered robust for PCB fabrication and assembly. As Table 3 shows, however, one of the materials with the best electrical performance also has the worst CTE. This is one reason why ceramic-filled PTFE laminates were formulated. They combine excellent electrical performance with very good CTE. Unfortunately, they exhibit poor dimensional stability, since the material is soft and circuit dimensions can be easily distorted. To provide good electrical performance and CTE with improved dimensional stability, ceramic-filled PTFE laminates with woven glass reinforcement were developed.

When making a choice in high-frequency circuit materials based on fabrication issues, the clear-cut favorite would be ceramic-filled hydrocarbon material with woven glass. These materials feature a low dissipation factor typically on the order of 0.003 and are robust in terms of most circuit fabrication processes. If better electrical performance is required, the choice would be ceramic-filled PTFE with woven glass. These materials typically have a dissipation factor in the range of 0.002 and are generally fabrication-process friendly.

The major concerns in fabricating PCBs with these materials relate to drilling and PTH preparation. For the best electrical performance, the choice is micro fiber glass PTFE, although this material can be difficult in terms of fabricating more complex circuit constructions. The material, which is nearly pure PTFE, is often used for simple high-frequency circuitry such as microstrip filters and couplers. Additionally this material is often used in a hybrid multilayer circuit, in which it supports critical functions, while other materials more friendly to circuit fabrication processes are used for the remainder of the multilayer PCB.

Choosing materials based on end-use applications
There are several different concerns for choosing materials for high frequency applications. A good example in chart form is given from the Rogers Corporation Product Selector guide on the website and a portion of this is shown in Table 4.

Product	Dielectric Constant, Dk ε_r @ 10 GHz (Typical)		Dissipation[7] Factor TAN δ @ 10 GHz (Typical)	Thermal[8] Coefficient of ε_r -50°C to 150°C ppm/°C (Typical)	Volume Resistivity Mohm • cm (Typical)	Surface Resistivity Mohm (Typical)	Moisture[4] Absorption D48/50 % (Typical)	Thermal[6] Conductivity W/m/°K (Typical) 80°C ASTM C518
	Process[7]	Design[12]						
RO3003™ PTFE Ceramic	[7] 3.00 ± 0.04	3.00	0.0013	11	10^{12}	10^{11}	0.05	0.50
RO3006™ PTFE Ceramic	6.15 ± 0.15	6.50	0.0020	-160	10^{3}	10^{3}	0.02	0.79
RO3010™ PTFE Ceramic	10.20 ± 0.30	11.20	0.0022	-280	10^{12}	10^{11}	0.05	0.95
RO3035™ PTFE Ceramic	3.50± 0.05	3.60	0.0018	-50° to 10°C: -34 10°C to 150°C: -11	10^{7}	10^{7}	0.08	0.50
RO3203™ PTFE Ceramic Woven Glass Reinforced	[7] 3.02 ± 0.04	3.02	0.0016	-75	10^{7}	10^{7}	0.06	0.48
RO3206™ PTFE Ceramic Woven Glass Reinforced	6.15 ± 0.15	6.60	0.0027	-212	10^{7}	10^{7}	0.05	0.67
RO3210™ PTFE Ceramic Woven Glass Reinforced	10.20 ± 0.50	10.80	0.0027	-459	10^{4}	10^{4}	0.13	0.81
RO4003C™ Hydrocarbon Ceramic	[8] 3.38 ± 0.05	3.55	0.0029	+40	1.7×10^{10}	4.2×10^{9}	0.04	0.71
RO4350B™ Hydrocarbon Ceramic	3.48 ± 0.05	3.66	0.0037	+50	1.2×10^{8}	5.7×10^{8}	0.05	0.69

Table 4: A tabular format for comparing high-frequency materials.

Table 4 provides a quick comparison of different circuit materials based on key electrical performance parameters, including dielectric constant (Dk), dissipation factor (Df), thermal conductivity, and CTE. Two values of Dk are listed for each material: process and design.

The process Dk refers to the value determined by industry-standard IPC test method, IPC-TM-650 2.5.5.5c at 10 GHz. This value is used as a process control for making the substrate. The test method is reliable and well proven, but the Dk value is specific to that test methodology and that test frequency. The test method uses a clamped stripline resonator and is a fixture mechanism allowing large volumes of materials to be tested, which is necessary for a laminate manufacturer. However, the fixture is not representative of an actual stripline circuit or a microstrip circuit, and the use of process Dk values in computer-aided-design software simulation tools has been known to yield erroneous results.

In some cases, process Dk values may not be ideal for design purposes. For that reason, a second set of Dk values, the design Dk numbers shown for each material in Table 4, were determined using actual microstrip transmission line circuits, across a wide frequency range. These values are more appropriate for circuit design and modeling.

Table 4 also lists tolerance values for Dk for each material. Some high-frequency applications have very tight specifications for impedance control and the Dk tolerance is a good indicator of how well this material may be suited for those applications. In addition, Table 4 shows values for Df for each material, which is related to dielectric losses. For an application that requires low-loss performance, a material with lower Df value would be a logical choice, although this choice should also be weighed against the ease or difficulty of PCB fabrication with that material.

Conductor losses

In addition to dielectric losses, conductor losses are important when comparing circuit materials. Especially for thin circuits, conductor losses can be more significant than dielectric losses.

Conductor losses can be impacted by circuit design, circuit configuration, and the thickness of conductive metals, as well as the surface roughness of the copper conductor layers. An excellent paper discussing this issue [2] has shown that conductor losses are higher for materials with higher amounts of copper surface roughness, compared to materials with smoother copper conductor surfaces. When comparing measurements by this parameter, the surface roughness measurement of concern is the root-mean-square (RMS) roughness of the copper surface.

A smooth copper conductor layer such as rolled annealed copper will typically have surface roughness RMS values around 0.3 microns. A low-profile electrodeposited (ED) copper conductor layer will typically have a surface roughness of around 0.8 microns, with standard ED copper at about 1.8 microns and high-profile copper at about 3 microns.

Figure 1 shows how increased copper surface roughness can result in increased loss. The same substrate — RO4350BTM laminate from Rogers Corp., — was used in both cases. This circuit material is a common ceramic-filled hydrocarbon woven glass. The higher-loss performance with frequency is plotted for this material with standard high-profile ED copper, while the lower loss results from using the same material with low-profile copper having a much smoother surface.

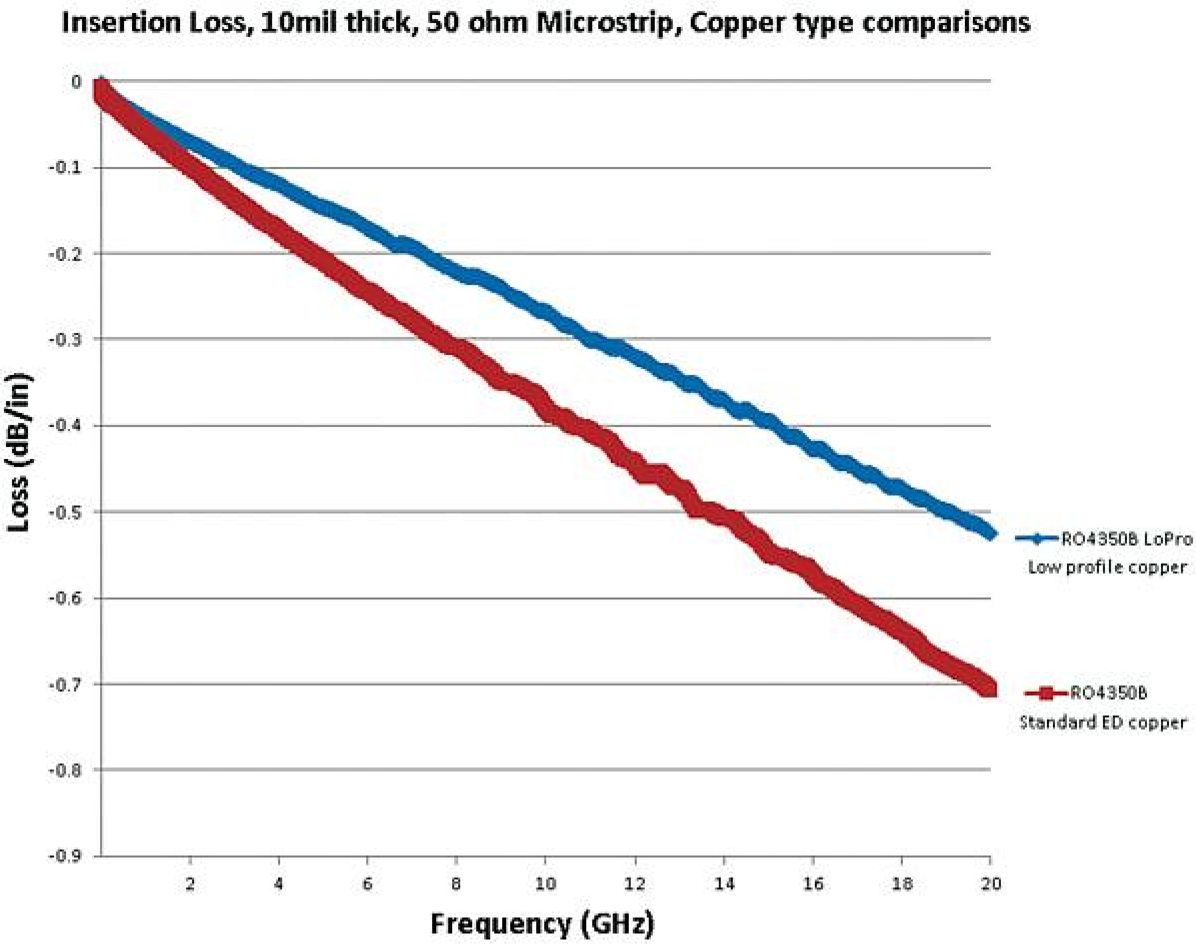

Figure 1: These plots of Insertion loss with frequency compare the same ceramic-filled hydrocarbon glass woven high-frequency laminate with standard ED copper and with smoother, low-profile copper.

Referring back to Table 4, another material property of interest is thermal coefficient of dielectric constant, or TcDk. Often overlooked in material comparisons, this is a measure of how much the dielectric constant (Dk) will change with changes in temperature. Given as changes in relative dielectric constant, ?r, in parts per million (ppm) for changes in temperature (in °C), large values of TcDk can be an indicator that circuits which perform well under ideal laboratory conditions may not fare as well under less controlled conditions, notably with large swings in temperature.

Another important material parameter In Table 4 is thermal conductivity, or the capability to move heat through a circuit material. This parameter is important to consider for high-power applications in which a large amount of heat must be dissipated. A substrate with high thermal conductivity can assist the thermal management issues with these applications.

Many standard PCB materials have thermal conductivities in the range of 0.25 W/m/K. Additionally, some materials with typically good electrical performance, such as micro fiber PTFE materials, also have thermal conductivities around 0.20 to 0.25 W/m/K. By adding a ceramic filler, the thermal conductivity of a circuit material can be improved. Table 4 shows that the ceramic-filled high-frequency materials have significantly better thermal conductivity than most standard PCB materials, generally with two to three times better thermal conductivity. This improvement can help solve many thermal management issues in high-power PCB designs.

Another material parameter listed in Table 4, moisture absorption, can also be important to consider for high-frequency applications. In an environment with high humidity, a circuit material that absorbs a high amount of moisture will exhibit increases in Dk and loss, both impacting PCB performance.

Circuit materials with high moisture absorption may not suffer degraded performance in controlled environments, but performance can be quite variable in more hostile operating environments. Many standard PCB materials have moisture absorption in the range of 1%. As Table 4 shows, however, most materials formulated for high-frequency applications are characterized by moisture absorption that is considerably less than 1%. For most high-frequency applications, laminates with moisture absorption values of less than 0.25% are considered acceptable.

There are many issues to consider when choosing a circuit material for high-frequency PCB applications. Some are related to fabrication issues for producing the most robust PCB possible, and some to achieving the best electrical performance possible for a given application. Because of various tradeoffs, the material for fabricating the most robust PCB may not be the same one for the highest electrical performance for an application.

Multilayer hybrid PCBs represent one way to choose a blend of materials to combine robustness and good electrical performance. By using charts of material properties such as Table 4, it is easier to compare the critical properties of different high-frequency materials and to simplify that choice when striving for the best tradeoff between ease of fabrication and best electrical performance.

References
[1] John Coonrod, "High Frequency PCBs Using Hybrid and Homogenous Constructions," PCB West 2010, September 2010.
[2] J.W. Reynolds, P.A. LaFrance, J.C. Rautio, and A.F. Horn, III, "Effect of conductor profile on the insertion loss, propagation constant, and dispersion in thin high frequency transmission lines," DesignCon 2010.

This article originally appeared on EE Times Europe.