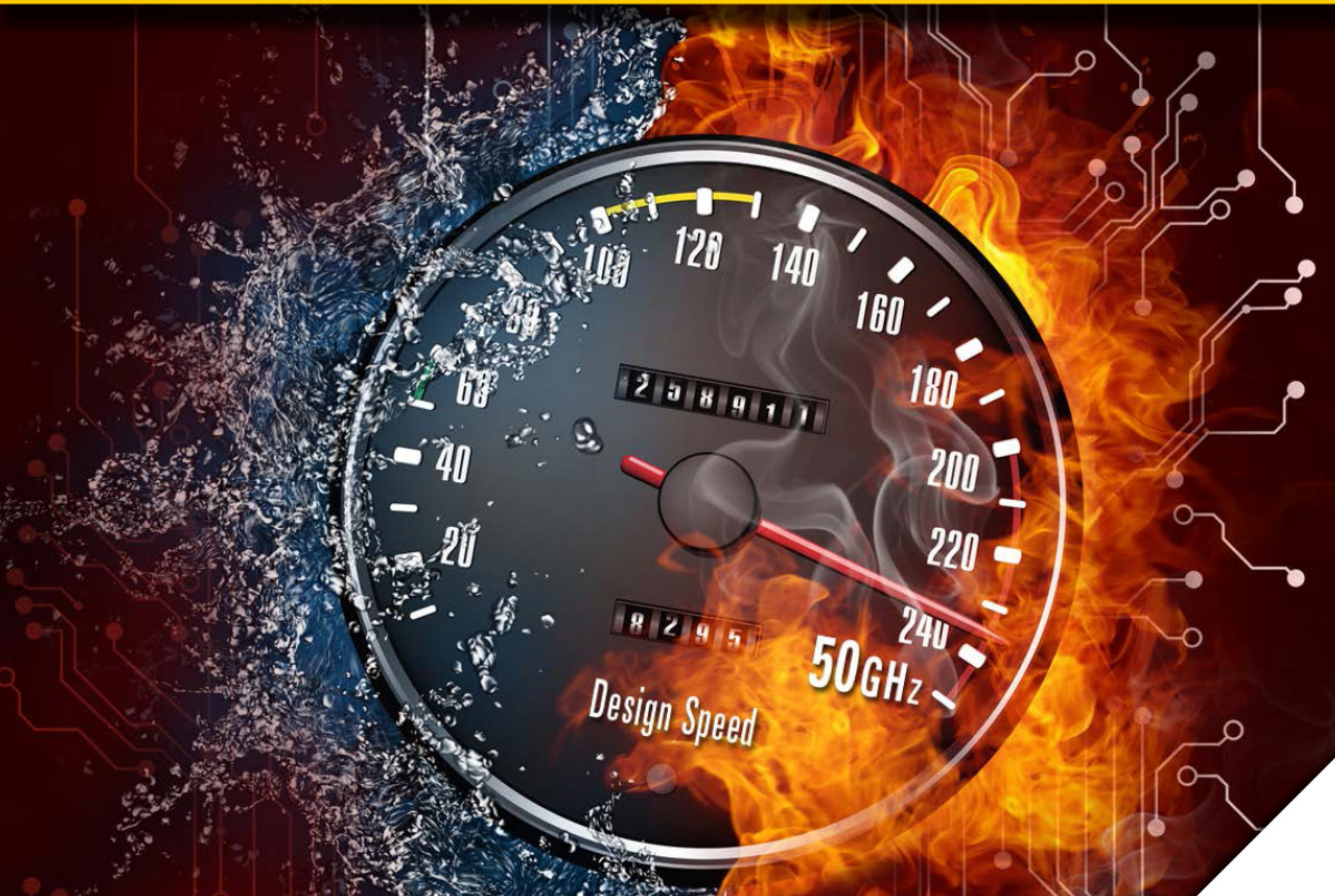
THE pcb design MAGAZINE
AN I-CONNECT007 PUBLICATION
February 2014

Effective Routing of Multiple Loads
by Barry Olney
p.30

What's Driving High-Speed PCB Design?
by Amit Bahl
p.48

HIGH-SPEED DESIGN



PCB and Packaging Design up to 50GHz:

Identifying Dielectric and Conductor Roughness Models

by Yuriy Shlepnev, Page 12

FAQ: Microwave PCB Materials

by John Coonrod
ROGERS CORPORATION

The landscape of specialty materials changes so quickly that it can be hard for product developers to keep up. As a result, PCB designers are inundated with data about microwave PCB materials. But very often, it's difficult to find useful information regarding these specialty substrates. So, this month, we present some of the most frequently asked questions about microwave materials encountered at Rogers.

Q: At what frequency is it necessary to transition from FR-4 types of materials to high-frequency circuit materials?

A: This is a challenging question to answer because different technologies can tolerate more or less performance from a laminate. I'll give a few examples and basic guidelines.

Semiconductor technology has developed enhanced signal processing to the point where a FR-4 material could be used at higher speeds and frequencies than was once thought possible. In most cases where high-speed digital applications reach 10 Gbps or more, you will need to use a high-frequency laminate. There are exceptions, and in some cases a lower data rate PCB will also demand a high-frequency laminate.

High-frequency RF circuits, which are less concerned with insertion loss, could use FR-4 in some of these applications. However, high-frequency laminates offer more than just low loss; they provide very well-controlled dielectric constant. In many RF applications, the control of dielectric constant for the material can be as critical as substrate thickness control. As a general statement, FR-4 materials are typically not used above 3 GHz in RF applications due to insertion loss concerns. However, when dielectric constant control is a critical concern, high-frequency materials should be used instead of FR-4 materials.

Q: Can high-frequency circuit materials be mixed with FR-4 materials for hybrid multilayer PCBs?

A: Yes, this is often done, and there are fewer compatibility issues than one may assume. A multilayer hybrid PCB using many layers of FR-4 and a few layers of high-frequency circuit materials is commonly utilized for several different reasons. In some cases, the reason is cost-related; the more expensive high-frequency laminates are used only in the layers where better electrical performance is needed.

"With many hybrid builds, the materials are often compatible, however special circuit fabrication processing is necessary. The material suppliers and circuit fabricators should always be involved when hybrid builds are required."

In other cases, a mix of materials may be used for improved reliability. Some PTFE-based laminates have high coefficient of thermal expansion (CTE), which can be a concern when a circuit is built with many layers of this material. Hybrid multilayers have been built with very thermally stable FR-4 (with a low CTE) used on non-electrically critical layers to help offset the higher CTE of some PTFE laminates. As a side note, not all PTFE laminates have a high CTE and many with ceramic fillers have extremely good and low CTE values.

Another case is when it is necessary for a multilayer PCB to use materials of significantly different dielectric constant. In those cases, different high-frequency materials may be mixed together to achieve the goal. With many hybrid builds, the materials are often compatible, however special circuit fabrication processing is necessary. The material suppliers and circuit fabricators should always be involved when hybrid builds are required.

Q: Are high-frequency circuit materials more difficult to fabricate than standard FR-4 materials?

A: Like so many questions and answers in the engineering world, the answer to this question is "it depends." There are several different types of materials used for high-frequency applications. The two most common circuit materials used are PTFE-based (such as DuPont's Teflon) and filled hydrocarbon. Both of these categories have subcategories and the following gives some distinction as it relates to circuit fabrication.

PTFE materials that are unfilled or nearly pure PTFE are typically more problematic for the PCB fabricator, because it's a soft material with a high CTE, and there are dimensional stability issues and special plated through-hole (PTH) processing challenges. PTFE laminates that are reinforced with woven glass negate many of the problems with softness, handling, and dimensional stability. When a laminate is a filled PTFE substrate, the CTE can be significantly improved, and the PTH preparation process is more forgiving. For ease of fabrication, the recommended material is a woven glass-reinforced ceramic-filled PTFE laminate. This laminate offers extremely good electrical performance and can be relatively friendly for PCB fabrication.

The different versions of filled hydrocarbon laminates can vary quite a bit, but in general they can be processed in the same processes as a FR-4 laminate. However, different parameters will be necessary for these processes, but usually the same equipment can be used. A few differences worth mentioning include the need for different speeds/feeds at drill, potentially less drill life, different parameters for PTH preparation using plasma or permanganate, and lamination parameters which are often very different from FR-4 prepreg.

Q: Why does the high-frequency industry need laminates with so many different dielectric constant options?

A: The need for different dielectric constant relates more to RF applications than digital applications. In the microwave range of frequencies (about 300 MHz to 30 GHz), the PCB circuit patterns are often the microwave circuit component. As an example, a bandpass filter might not be a component that is soldered onto the PCB, but is actually the conductor pattern of the PCB. An edge-coupled series of conductors and space can act as a bandpass filter at microwave frequencies. A simple microstrip edge-coupled filter is shown in Figure 1 and its function is based on wavelength which is related to the dielectric constant of the material.

Wavelength is a property of an electromagnetic wave that propagates on the circuit. As the name implies, the wavelength is the physical length of the wave and this length is dependent on the dielectric constant of the material and the frequency. The size of many microwave components is based on a fraction of the wavelength; often ½ or ¼ wavelength is used to define circuit features.

In Figure 1, there are line segments with small spaces between them. It may be hard to believe but electrical energy will pass through this circuit very effectively, at the right frequency. The length of each pair of conductors (with a space between them) is ¼ the wavelength at the intended frequency. With conductor lengths at ¼ of the wavelength, there will be a lot of electric energy radiated on one of these elements and that energy couples to its neighboring element (line segment). At the right frequency, the energy of the propagating wave will jump from one element to another, going down this circuit. However, when energy is introduced into this circuit pattern at a different frequency, the wavelength will be different and the energy will not couple from one element to another. Therefore, no electrical energy will propagate.

The physical size of this bandpass filter is related to wavelength at the frequency of interest. If the circuit material dielectric constant is much higher, the size of the circuit would shrink in order to maintain the same wavelength properties. Having a high dielectric constant material can reduce the size of a microwave circuit. This is only one example of many, but in general, microwave circuitry is very dependent on the dielectric constant of the material because the technology uses wavelength properties to produce many different types of circuit functions. Wavelength is very much related to dielectric constant, and a higher dielectric constant yields shorter wavelength.

Q: Why is insertion loss getting so much attention on new applications?

A: Actually, the microwave industry has been concerned with insertion loss for decades. However, now more digital applications are focusing on this concept. Insertion loss is a complicated subject, so I can only give a simple overview here. But in general, insertion loss is the total electrical loss of a high-frequency/high-speed circuit.

Insertion loss is made up of a combination of many losses and one of them is dielectric loss. Usually dielectric loss is related to the dissipation factor (tan-delta, loss tangent) of the circuit material. As the name implies, dielectric loss is the property of a circuit where losses occur due to the substrate. Conductor losses are

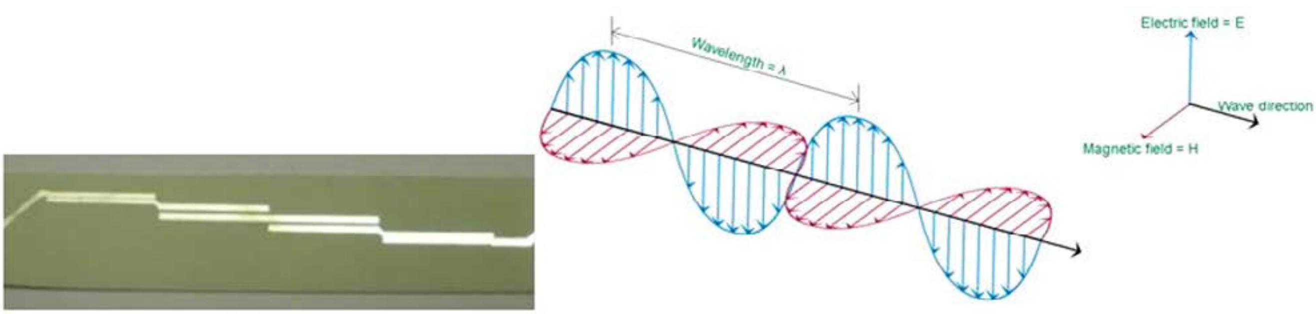


Figure 1. Microstrip edge-coupled bandpass filter as part of the PCB pattern, and a wavelength illustration.

also part of insertion loss. This is loss that is associated with the copper conductors and finish. There is no perfect conductor, but copper is very good for conductivity. If another metal is added to the copper, the overall conductivity will typically decrease which causes more loss. The different finishes used in the PCB industry are typically lower in conductivity than copper and more loss occurs when some plated metal finish is added to the copper conductor. Also, the surface roughness of the copper conductor can cause more loss with a rougher (high-profile) copper surface. Conductor losses are frequency-dependent due to skin depth, and at some frequencies, the conductor losses are minimal while other frequencies can be significant.

Thermal management issues can be a concern when insertion loss causes the circuit to heat up when RF power is applied. A circuit with lower insertion loss will heat less with applied power. In the case of high-speed digital applications, insertion loss will cause the digital pulse to decrease in amplitude and cause possible distortion. Having a circuit with low insertion loss will allow the digital signal to maintain the good integrity needed.

There are many potential interactions between PCB fabrication, circuit design and material properties which need to be understood to ensure the finished circuit will meet the needs of the end user. It is always recommended to consult the circuit material's manufacturer for new designs and applications to ensure the optimum material is used for the application. **PCBDESIGN**

John Coonrod is a market development engineer for Rogers Corporation's Advance Circuit Materials Division. To contact Coonrod, click here.