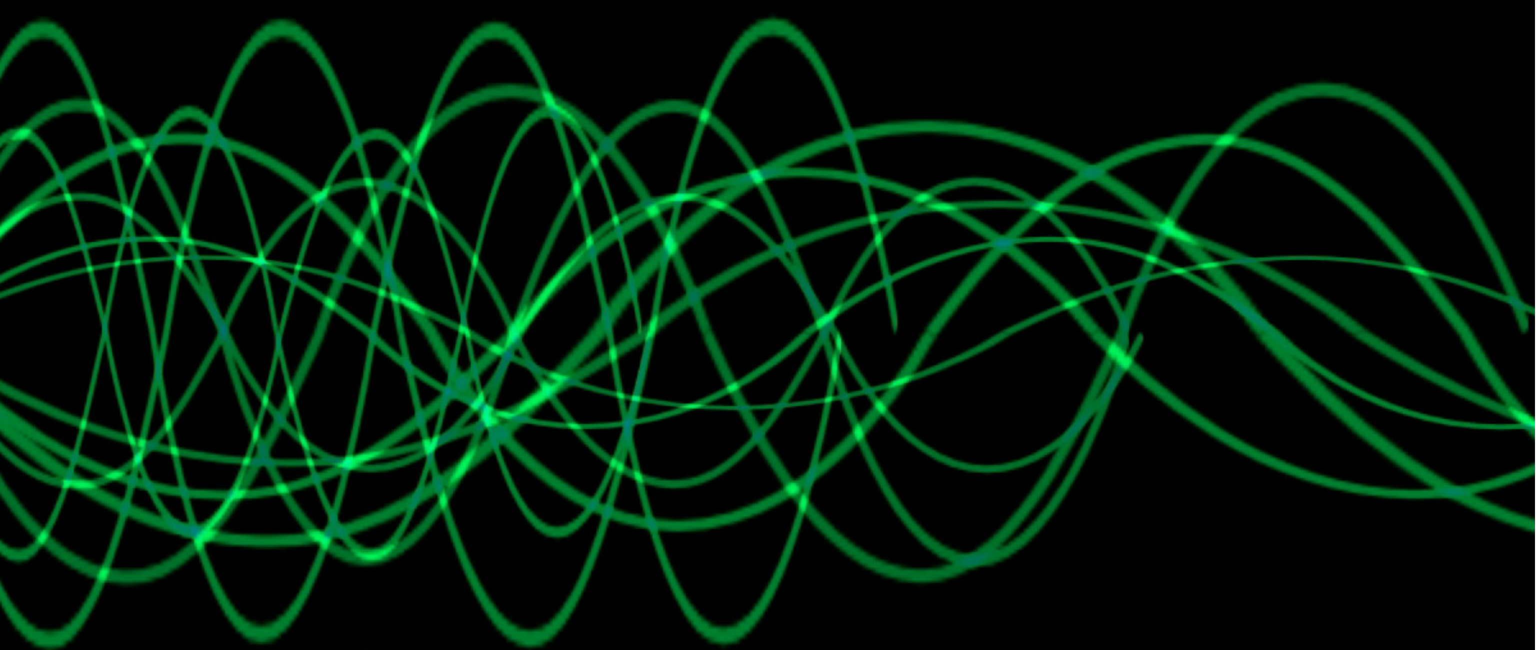
THE pcb design MAGAZINE

AN I-CONNECT007 PUBLICATION

August 2013

SIGNAL INTEGRITY



SEE PAGE 8

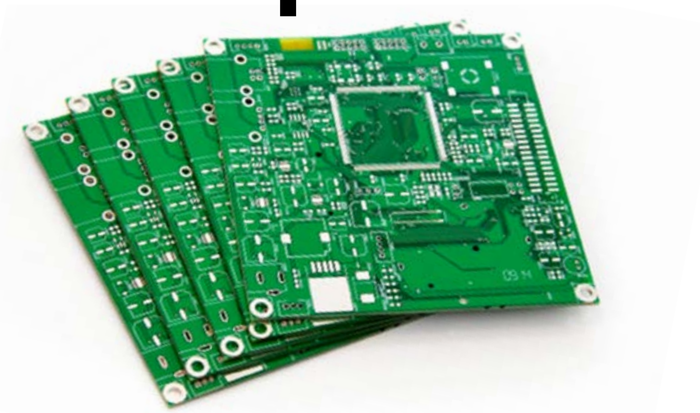
Feature Article of the Month

Differential Signal Design (Part 1)

by Lee Ritchey

PCB Materials that Empower Signal Integrity

by John Coonrod
Rogers Corporation,
Advanced Circuit Materials Division



Signal integrity is a broad study of issues related to electrical performance. It can be associated with RF applications; however, it is more commonly related to high-speed digital concerns. Several attributes associated with circuit materials can contribute to good signal integrity.

Impedance control for PCBs is typically a concern when considering signal integrity. The factors impacting impedance control are circuit thickness, conductor width and spacing, copper thickness and the dielectric constant (Dk) of the material. There are interactions between these variables, but one point to consider is that the thickness of the circuit has much to do with how sensitive the other factors are to affecting impedance changes. A simple example is to consider a double-sided microstrip 50 ohm transmission line at two different thicknesses. Using a circuit with 10 mil thick material and a Dk of about 3.9, a conductor width change of 1 mil will cause about a 2.8% change in impedance. For a thinner circuit using the same material 4 mils thick, a conductor width change of 1 mil results in an impedance change of 6.8%.

Also considering the same example, the Dk control of the material is less sensitive to impedance change regardless of the circuit thickness. A difference of 0.1 for the Dk in the previous example will yield a difference in impedance of 1.1% for the 10 mil circuit and also 1.1% for the 4 mil circuit.

For circuits requiring a tight impedance tolerance, all aspects that can affect impedance should be considered. Even though Dk appears to be a relatively low concern, if a circuit has a tight impedance tolerance such as ± 5%, then obviously Dk control is worth considering. The laminates used in the high-frequency segment are formulated for tightly controlled Dk tolerance and should be used for applications where the impedance tolerance has a narrow specification.

Circuit thickness is a driving factor for many variables associated with impedance change and given this consideration, the best circuit fabrication process should be used to have good thickness control. Most digital PCBs are multilayers, and with regard to the previous microstrip example, that circuit would be on the outer two copper layers of the PCB. These layers could be created by a foil lamination using prepreg or a core lamination using a laminate. Most high-frequency laminates have a tightly controlled thickness tolerance. It generally makes more sense to use the core lamination process with one of these laminates as opposed to the foil lamination in order to get a circuit with better thickness control.

Over the past few years, more attention has been placed on insertion loss for digital PCBs. One of several reasons for this is signal integrity. A circuit with higher insertion loss will cause a digital signal to lose amplitude, which will affect the integrity of the pulse shape. Additionally, all circuit materials have an increase in dissipation factor with an increase in frequency, which generally means that the insertion loss will be worse for higher frequencies.

Digital pulses are formulated from multiple RF waveforms at different frequencies. High speed digital signals are formulated by using a fundamental frequency, combined with multiples (harmonics) of that frequency. Generally speaking, a digital signal at a rate of 1 Gbit/s will use multiple analog waves. The fundamental analog frequency will be 0.5 GHz and multiples of this frequency are 1.5 GHz, 2.5 GHz, etc. If the material dissipation factor changes significantly from 0.5 GHz to 2.5 GHz, that will impact the formation of the digital signal and can be an issue for signal integrity. Many PCB materials do not have a significant difference in dissipation factor across this range of frequencies. However, this issue can be a very different concern for rates that are much faster.

There are now many high-speed digital applications at 10 Gbit/s and higher. These applications use a fundamental frequency of 5 GHz and harmonics of 15 GHz, 25 GHz, etc. In this range of frequencies, most common PCB materials will have a very significant difference in dissipation factor and cause serious signal integrity issues. This is one reason high-speed digital PCBs are using special circuit materials that are formulated for high-frequency applications. These materials are formulated for having a low dissipation factor with minimal change across a wide range of frequencies. The materials have been used historically in high frequency RF applications and even now used in applications of 77 GHz and higher. Besides the dissipation factor improvement, the materials are formulated for tight thickness control and Dk control, both being beneficial to signal integrity.

Signal integrity issues continue to become more complex as speeds increase. Understanding the attributes of high-frequency circuit materials can be very advantageous in minimizing signal integrity issues. When dealing with signal integrity issues, your material supplier can help you choose the appropriate circuit material to achieve the best performance from your particular technology. **PCBDESIGN**

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

video interview

Micro-Waves, Macro-Challenges

by Real Time with... IPC

John Coonrod talks to Guest Editor Kelly Dack about Rogers Corporation's high-frequency laminates and his paper about the topic of insertion loss and how Rogers materials, combined with prudent design and manufacturing methods, are solving the industries most challenging microwave challenges.

realtimewith.com