PCB007 PRESENTS

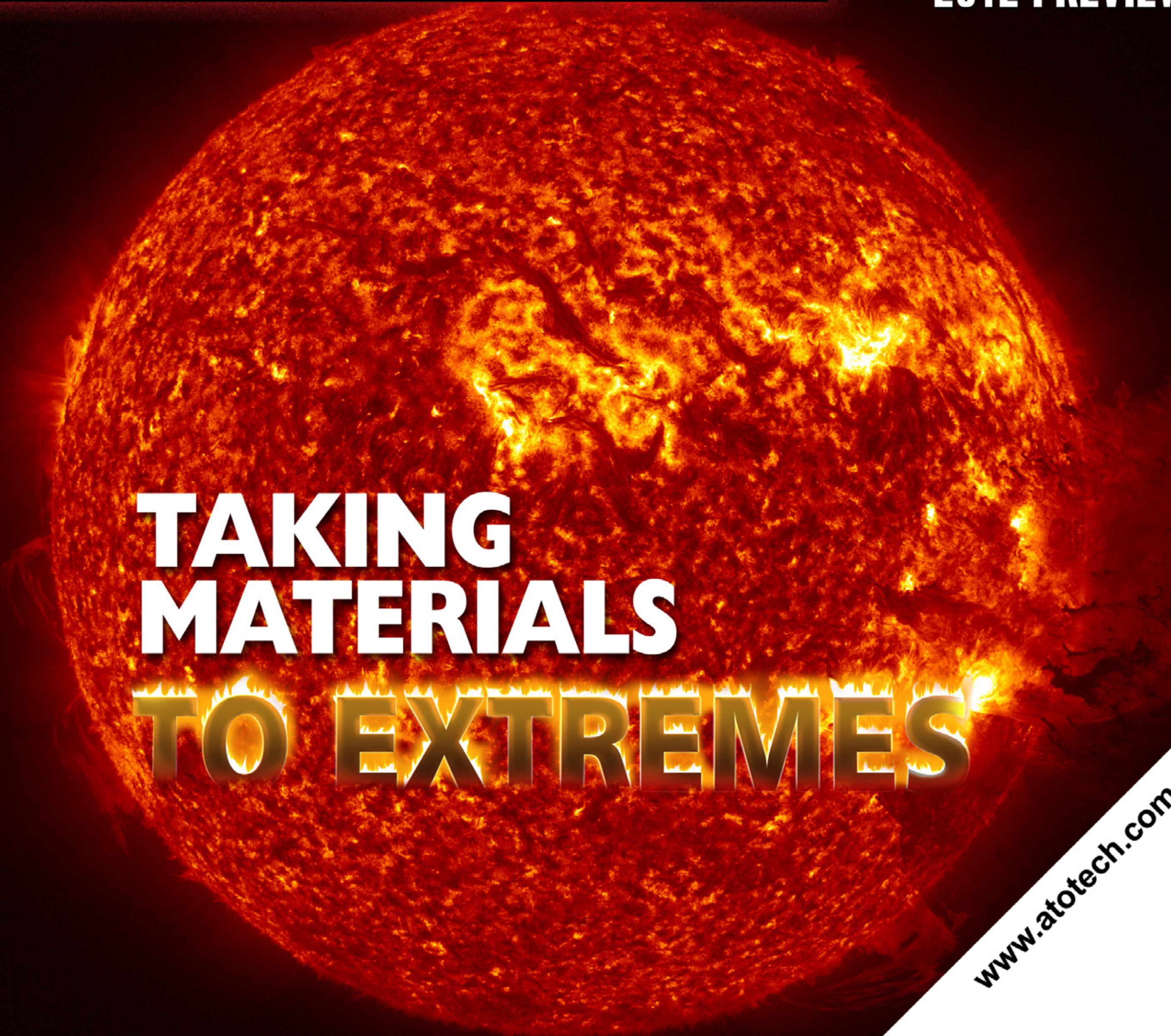
February 2012

the pcb magazine

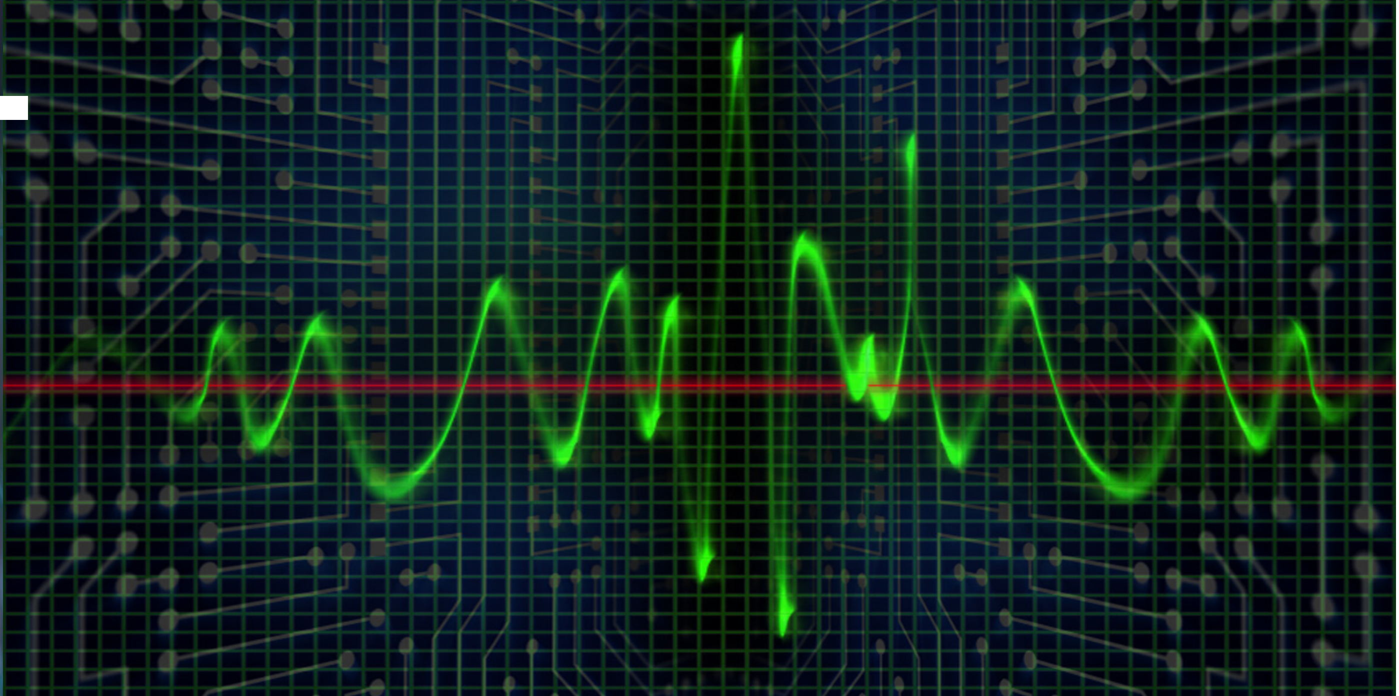
AN I-CONNECT007 PUBLICATION

APEX EXPO IPC

2012 PREVIEW



TAKING MATERIALS TO EXTREMES



Circuit Materials and High-Frequency Losses of PCBs

by John Coonrod
ROGERS CORPORATION
ADVANCED CIRCUIT MATERIALS DIVISION

SUMMARY — As PCB technology advances, concerns for high-frequency electrical losses are becoming prevalent. There are many aspects to consider regarding losses with circuit design, fabrication and configuration. There are also many issues to consider, with circuit materials playing a critical role.

The PCB industry is diverse, from dynamic flexible circuitry that is as thin as a human hair, to multilayer backplanes which could be used as a diving board. Although that comparison is a little extreme, great diversity exists in the PCB industry regarding experience with electrical losses as well. The sector of the PCB industry that caters to microwave applications is accustomed to loss issues. Many other sectors of the industry probably have a mix of experience on this topic.

Recently, there have been more requirements for PCBs to have loss testing performed as an outgoing quality check. This is similar to what has been done in the past with controlled impedance boards requiring testing to ensure the circuit is within an impedance tolerance prior to shipping the product. However, there are vast differences between testing for impedance and loss testing. Additionally, materials and circuit fabrication issues regarding losses are significantly different than impedance concerns.

In general, impedance issues are related to the dielectric constant (Dk) of the material, the precision of the conductor-etched features and the thickness of the PCB in electrically critical layers. There are more issues to consider with losses. These include dissipation factor (Df) of the material, copper surface roughness, precision of the conductor etched features, conductor length and impedance mismatches. The impedance mismatch concern refers back to the issues for impedance control; therefore loss issues have their own list combined with impedance issues. Additionally, and depending on the circuit construction, the plated finish and/or solder mask applied to the conductors can also have an impact on losses.

Electrical losses for a PCB design become important when the circuit is no longer acting like a simple interconnect, but is actually a transmission line. This is defined as the circuit conductor length being longer than one-fourth the wavelength and for a 3GHz signal that is about

“Electrical losses for a PCB design become important when the circuit is no longer acting like a simple interconnect, but is actually a transmission line.”

a 2-inch length, subject to the Dk of the materials and circuit construction. Actually, a better rule of thumb would be to assume the circuit is acting like a transmission line when the length is 1/10 of the wavelength. In terms of high-speed digital, the same definition applies, but it is typically related to the conductor length in terms of propagation delay and with regard to rise time.

Insertion loss is the term generally used to refer to total loss of a RF system. When looking at the PCB only, the insertion loss is comprised of four components of loss which include dielectric, conductor, leakage and radiation. Radiation loss is not going to be discussed; however, it can be an issue for some microstrip configurations. Leakage loss is generally not an issue with most PCB applications, where the substrate has very high resistivity values. Dielectric loss is mostly related to the Df of the material; however, some anomalies in the circuit fabrication process can have an impact on this subject. Conductor loss has several components of interest regarding materials and PCB fabrication, which will be expanded on in the remainder of this article.

Assuming radiation and leakage losses are not concerns, the two components making up insertion loss are dielectric loss and conductor loss. The ratio of dielectric loss to conductor loss can be important to understand for a particular construction, as it can vary based on construction and frequency. To illustrate this concept, several models were run on a common circuit construction as well as circuits tested for loss. The circuit construction was a simple microstrip transmission line and the software model used was MWI-2010 [1]. The microstrip circuit is a double-sided, non-plated through-hole using RO4350B™ copper clad laminate with ½ ounce ED copper. Three different substrate thicknesses were chosen to show the difference in total loss and their components. Additionally, circuits were tested for insertion loss and that data is shown in Figure 1.

The information shown in Figure 1 tells quite a bit about insertion loss. To begin with, it can be seen that the measured data (purple) compares relatively well to the total loss (green) of the model. The model has the ability to break

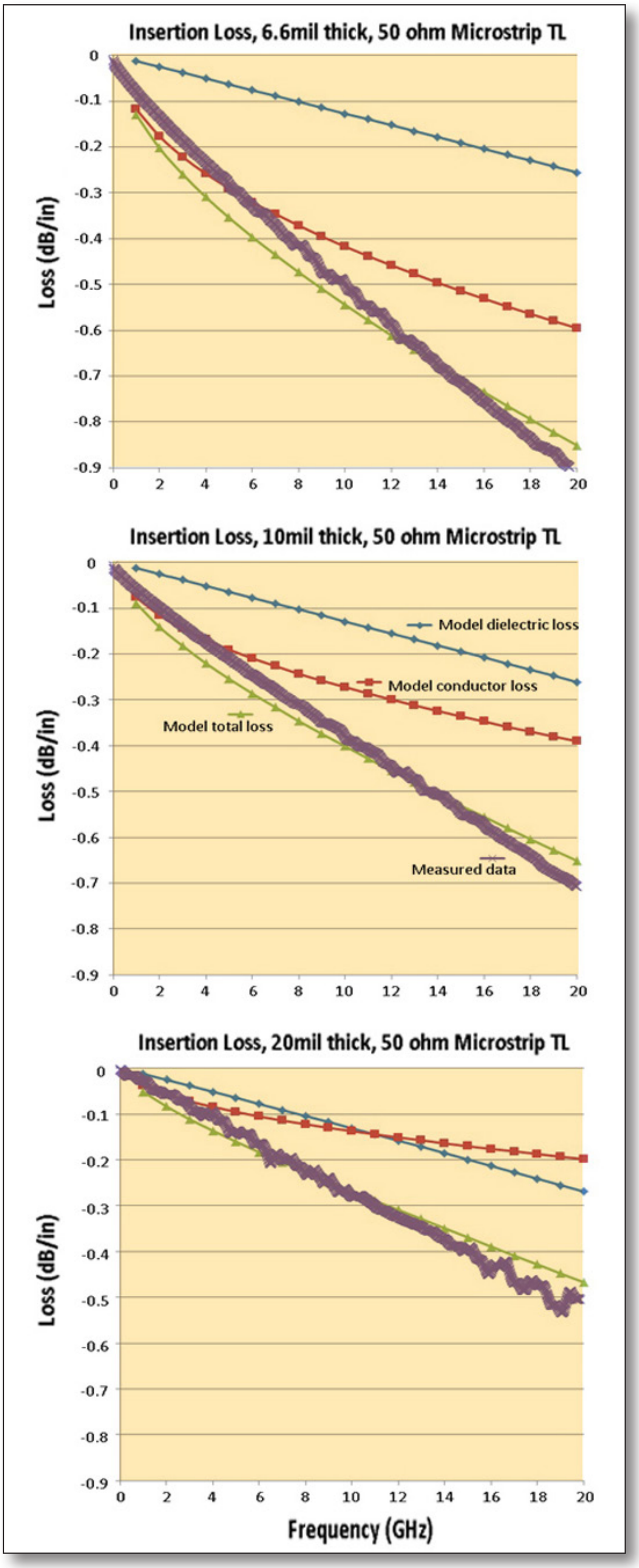
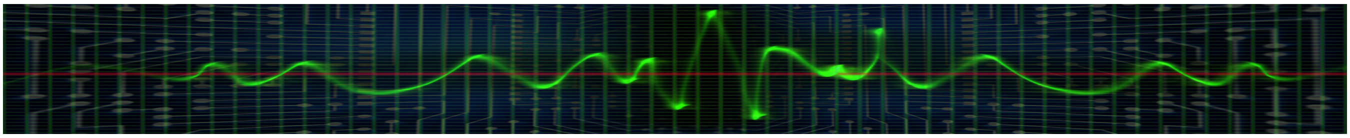


Figure 1: Insertion loss comparisons of microstrip transmission line circuits using different substrate thickness.



out dielectric (blue) and conductor loss (red) separately. Looking at the ratio of conductor and dielectric loss at 10 GHz on the 20mil circuit, there is about a 1:1 relationship. The dielectric and conductor loss are both about -0.14 dB/in at 10GHz, and added together, this gives us the total loss. The 10mil circuit shows that the conductor loss is about double the dielectric loss and at 10GHz they are -0.27 dB/in and -0.13 dB/in, respectively. The 6.6mil circuit shows that the conductor loss is dominating the overall insertion loss and at 10 GHz, it is about -0.42 dB/in and the dielectric loss is -0.13 dB/in.

When comparing circuits of different thicknesses, it is obvious that the trend of thinner circuits brings more insertion loss. That is correct for several reasons, one of which is controlled impedance. All of these circuits were 50 ohm transmission lines; when a thinner substrate is used, a narrower conductor must be used to maintain the 50 ohm structure. The narrower conductor will have higher conductor losses and overall insertion loss. The RO4350B circuit material that was used here is considered a low-loss material with a Df of 0.0037. It can be seen in Figure 1 that the dielectric loss is relatively unchanged with the thickness variation of the circuits. In reality, there are some small changes in dielectric losses with these circuits, but it does show that the effects of conductor loss are important when using a thinner substrate. The copper clad laminate in this study used a high profile copper and it is known [2] that a rough copper surface will increase conductor losses.

To illustrate the effects of the copper surface roughness, circuits were made with the same substrate, but varying the copper type, which has a different surface roughness. The 10mil circuit was used where the conductor losses are about double the dielectric losses (Figure 1). The curves in Figure 2 show the results of insertion loss on circuits using the same substrate, but with the copper having a significantly different surface roughness.

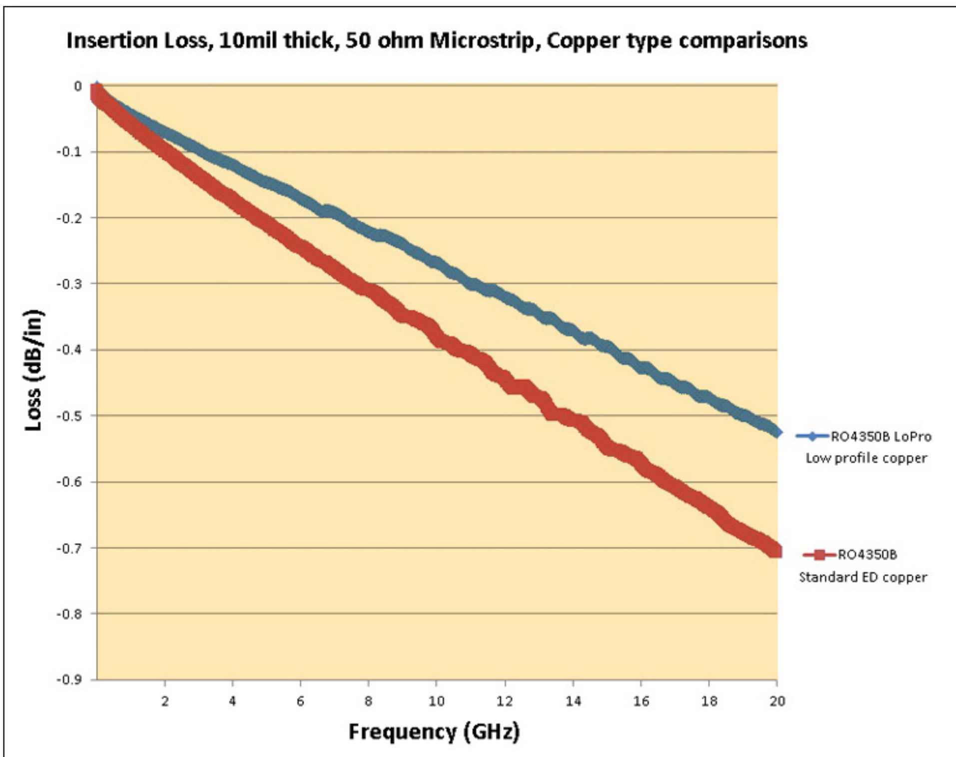


Figure 2: Insertion loss comparison of 10mil thick microstrip transmission line circuits using the same substrate, but with different copper.

As shown in Figure 2, the circuits using the material with smoother copper, RO4350B LoPro™ laminate have much lower insertion loss. This is mostly due to the smoother copper surface of the LoPro technology. The LoPro technology also yields a copper clad panel with slightly lower Dk than the standard RO4350B laminate, allow-

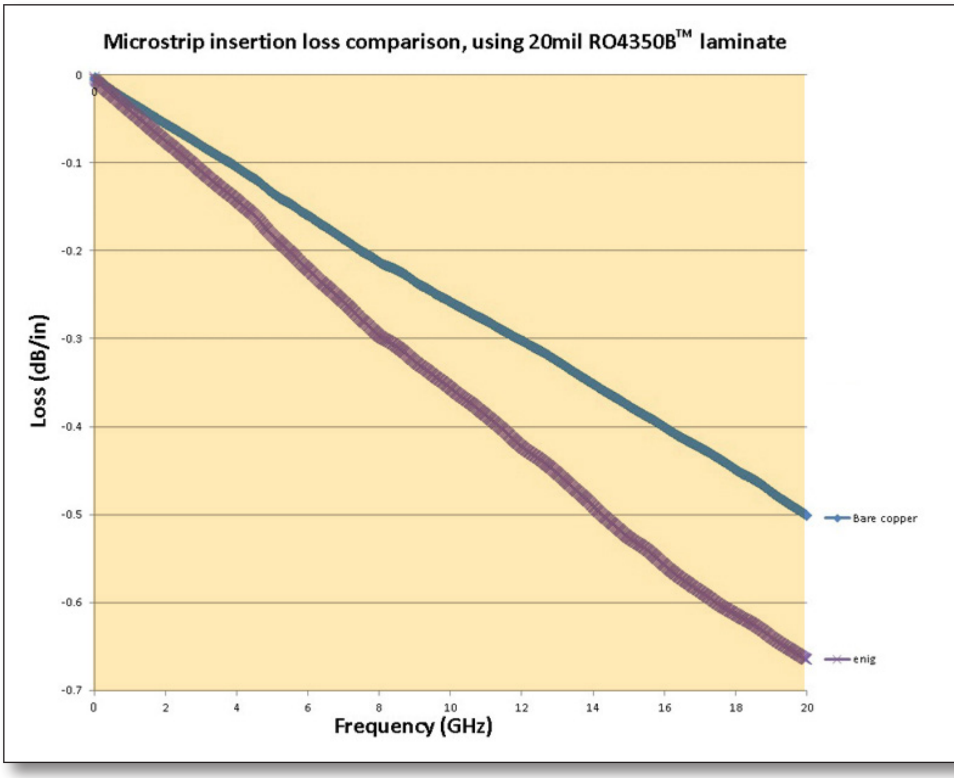


Figure 3: Insertion loss comparison of 20mil microstrip transmission line circuits using standard RO4350B laminate and comparing performance of bare copper to ENIG finish.

ing a somewhat wider conductor to maintain 50 ohms. The copper surface roughness used for these two laminates are 2.8 microns for the standard RO4350B laminate and 0.6microns for RO4350B LoPro circuit materials; the roughness measurements are in terms of RMS (root mean square) .

Besides copper surface roughness, there are other issues which can impact conductor losses. For microstrip or coplanar transmission line circuits, the addition of some plated finish can cause more losses. ENIG (electroless nickel immersion gold) is a common finish with many proven good attributes; however, it can cause more conductor losses. The losses due to ENIG can be difficult to quantify for many reasons. There are several different ENIG processes and each has their unique properties. Some of the ENIG processes appear to have less impact on conductor losses than others. Additionally, the losses due to ENIG are frequency-dependent, in the sense that nickel is much less conductive than copper and at the right frequencies much of the nickel could be used by the RF current. At higher frequencies, less of the conductor is used and if the skin depth current is mostly using the nickel layer, the conductor losses will increase. To compound this issue, the nickel layer has some natural thickness variation. As an example, two exact PCB constructions can have significantly different loss performance due to the variation of the nickel. Figure 3 shows a comparison using the 20mil circuit from Figure 1, with and without ENIG. Insertion losses are shown.

As previously mentioned, dielectric losses are mostly due to the Df of the substrate. It has already been shown that when the laminate is thinner, the dielectric losses are less significant, however that data has been shown with a low-loss substrate. A higher-loss substrate such as FR-4 will certainly have higher losses and as frequency increases the losses will increase significantly more than the low-loss material shown here. The RO4000® circuit materials have been formulated to be very stable over a wide range of microwave frequencies and that is why the dielectric losses do not change much on the different thickness circuits shown in Figure 1. In contrast, if a high-loss material such as FR-4 is used, that will show much higher losses and the losses will likely increase with frequency in a non-linear manner.

As a summary illustration, an insertion loss comparison is shown in Figure 4, displaying results of several 20mil microstrip transmission line circuits using very different substrates. The RO4000 circuit materials discussed thus far will be shown along with a high Tg FR-4 and a ceramic filled PTFE substrate (RO3035™ circuit materials).

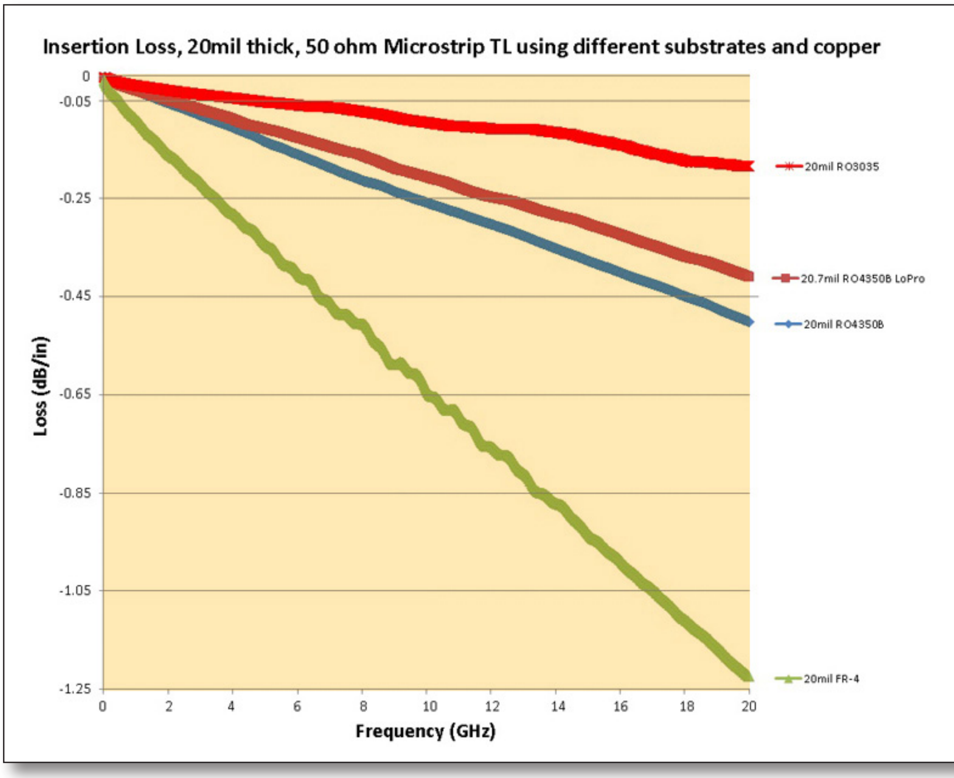


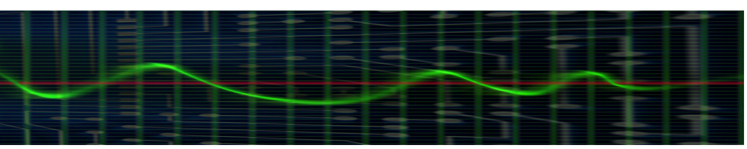
Figure 4: Insertion loss comparison using several different substrates for 20mil thick microstrip transmission line circuits.

It can be seen from the data that insertion loss can be influenced by several different aspects. A thinner circuit will have more conductor losses and the choice of copper is more important. The plated finish can also have an impact on losses. For an application where minimizing losses is critical, then typically a very low-loss substrate would be used in combination with a very smooth copper. This was shown in Figure 4 with the RO3035 laminate using a rolled annealed copper. **PCB**

[1] MWI-2010 software is a free download from Rogers Corporation, Technology Hub. This software performs simple impedance and loss predictions using the Hammerstad and Jenson models: E. Hammerstad and O. Jenson, "Accurate models of microstrip computer aided design", 1980 MTT-S Int. Microwave Symp. Dig., May 1980, pp. 407-409. The copper roughness model in this software is limited to frequency and is from Morgan: S. P. Morgan, "Effect of surface roughness on eddy current losses at microwave frequencies," J. Applied Physics, p. 352, v. 20, 1949.

[2] J.W. Reynolds, P.A. LaFrance, J.C. Rautio & A.F. Horn III, "Effect of conductor profile on the insertion loss, propagation constant, and dispersion in thin high-frequency transmission lines", DesignCon 2010.

RO4350B, LoPro, RO4000 and RO3035 are licensed trademarks of Rogers Corporation.



John Coonrod is a market development engineer for Rogers Corporation, Advanced Circuit Materials Division. About half of his 24 years of professional experience has been spent in the flexible PCB industry doing circuit design, applications, processing and materials engineering. Coonrod's regular column, Lightning Speed Laminates appears bi-monthly in The PCB Magazine.