Choosing Circuit Materials for Millimeter Wave Applications

By John Coonrod

> **It is possible to choose wisely when selecting PCB materials for use at these higher frequencies.**

Millimeter-wave frequencies are attractive for communications and other applications for the broad bandwidths available at these frequencies, from 30 to 300 GHz. But finding printed-circuit-board (PCB) materials that provide high performance levels for reasonable prices at these frequencies can be challenging. However, by understanding the key parameters and characteristics of concern for PCB materials at millimeter-wave frequencies, such as how different circuits behave for different types of PCB materials at millimeter-wave frequencies, it is possible to choose wisely when selecting PCB materials for use at these higher frequencies.

Many of the concerns when designing microwave circuits apply to higher-frequency millimeter-wave circuits, where these concerns can easily become more severe or carry greater impact. These issues include limiting spurious wave mode propagation problems, minimizing conductor and radiation losses, achieving effective signal launch, minimizing unwanted resonances, and controlling dispersion.

Guidelines

Numerous guidelines help minimize wave propagation issues, such as using a circuit laminate that is relatively thin. A general rule is to use a laminate that is thinner than one-quarter wavelength at an application's highest operating frequency. In practice, better results can be achieved by using a laminate that is thinner than one-eighth wavelength at the highest operating frequency, to eliminate unwanted resonances between different circuit planes in a circuit assembly. Such resonances can interfere with the desired propagation for a circuit and also generate surface waves that can disrupt the desired wave propagation. The width of the signal conductors is also related to the thickness of a circuit laminate, since a thinner laminate will use a narrower conductor width. To help eliminate mode issues, the conductor width should be one-eighth wavelength or less at the highest operating frequency.

These rules for laminate thickness and conductor width apply directly to high-frequency microstrip circuits; other types of circuit configurations may be more forgiving.

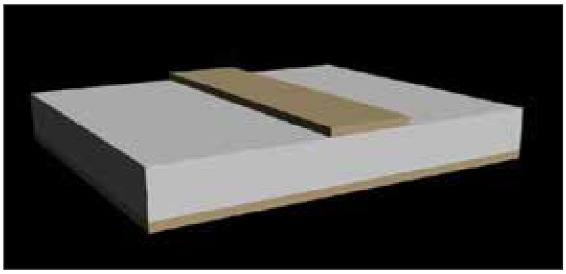


Figure 1a. • Microstrip transmission line circuit.

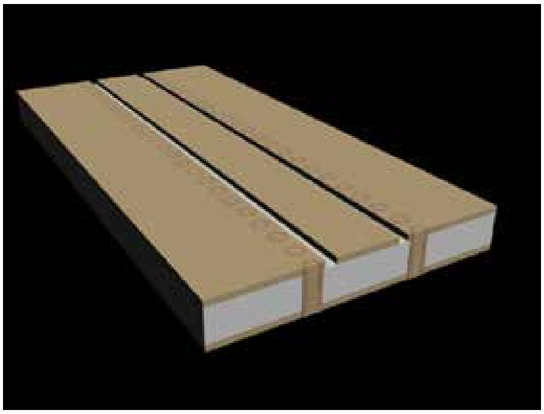


Figure 1b. • Grounded coplanar waveguide transmission line.

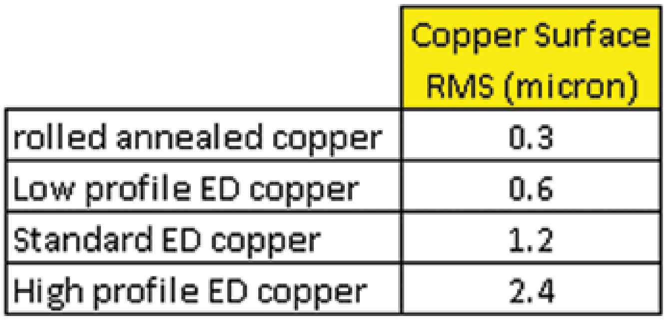
For grounded coplanar-waveguide (GCPW) circuits, which are also known as conductor-backed coplanar waveguide (CBCPW), thicker laminates have been shown to exhibit minimal mode issues at millimeter-wave frequencies.

In the microstrip diagram of Figure 1a, the microstrip transmission line circuit has a set distance between the signal layer and the ground plane. If that distance (substrate thickness) is one-quarter wavelength, resonance can occur between the copper planes and this resonance may interfere with the desired wave propagation. Additionally, if the substrate is one-quarter wavelength thick and the conductor width is narrower than one-quarter wavelength, a resonance may not occur or be marginalized. If both the substrate is thicker than one-quarter wavelength, and the conductor width is one-quarter wavelength or more, unwanted resonances and wave propagation issues will likely occur. Figure 1b illustrates a GCPW circuit. Even when the substrate thickness and conductor width are equal to one-quarter wavelength, a resonance may be avoided due to the close coupling of the coplanar ground planes. The coplanar ground planes are adjacent to the center signal conductor and are grounded by means of plated through holes (PTHs). Of course, tradeoffs are part of any choice of high-frequency circuit configuration, and GCPW will suffer higher conductor loss than microstrip. However, depending upon the operating frequencies, this may not mean more overall insertion loss due to a GCPW circuit possibly exhibiting less radiation loss than a microstrip circuit.

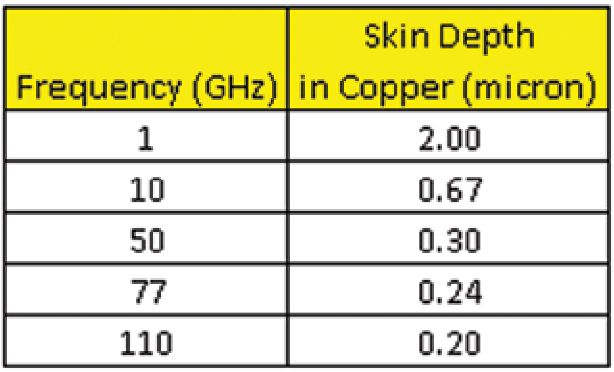
For high-frequency transmission lines and transmission-line circuits, insertion loss is actually a total of various component losses: dielectric loss, conductor loss, radiation loss, and leakage loss. PCB materials for high-frequency applications typically have high volume resistivity, with resulting minimal RF leakage losses. Dielectric losses are related to the tangent delta or dissipation factor of the circuit substrate material. These losses are affected by any additional substrate materials, such a soldermask or prepreg/bonding layers. Soldermask are typically not used at RF/microwave frequencies and especially at millimeter-wave frequencies, since these are very high loss materials where a dissipation factor of 0.02 is not uncommon. Soldermasks are not typically characterized by well-controlled dielectric constant (Dk), and using soldermasks can lead to impedance mismatches, and their impact on increasing return loss and ultimately insertion loss.

Thickness Variations

Soldermasks are often guilty of thickness variations when applied to one circuit to the next or even within the same circuit, which can result in unwanted impedance variations. Soldermasks also typically have high moisture absorption characteristics, which can seriously degrade the performance of a PCB. Moisture, basically water, has a Dk of about 70 and very high dissipation factor, both values much higher than the circuit material, so that as water or moisture is absorbed, a circuit material's Dk will rise and its loss will increase. As a result, soldermask should be used sparingly or not at all at millimeter-wave frequencies.



	Copper Surface RMS (micron)
rolled annealed copper	0.3
Low profile ED copper	0.6
Standard ED copper	1.2
High profile ED copper	2.4



Frequency (GHz)	Skin Depth in Copper (micron)
1	2.00
10	0.67
50	0.30
77	0.24
110	0.20

Table 1a. Copper surface roughness values are listed for copper types typically used in RF/microwave PCBs. Table 1b. Copper skins depths are shown versus frequency.

As thinner substrates are used, as for millimeter-wave circuits, conductor loss becomes more of a concern; conductor loss also grows more significant with increasing frequency. Copper with a roughened surface is often used for improved adhesion to the dielectric material in a PCB. But this surface roughness can also result in higher loss. As a rule of thumb, when the skin depth for a frequency of interest is equal or less than the copper surface roughness, the surface effects of the conductor will be significant. At millimeter-wave frequencies, the skin depth is commonly less than the copper surface roughness.

Copper surface is measured by different methods and in different units of measure. For RF/microwave applications, the appropriate copper surface roughness measurement is Rq or root mean square (RMS). Table 1a lists copper roughness for several copper types used for high-frequency PCBs. As Table 1b shows, the skin

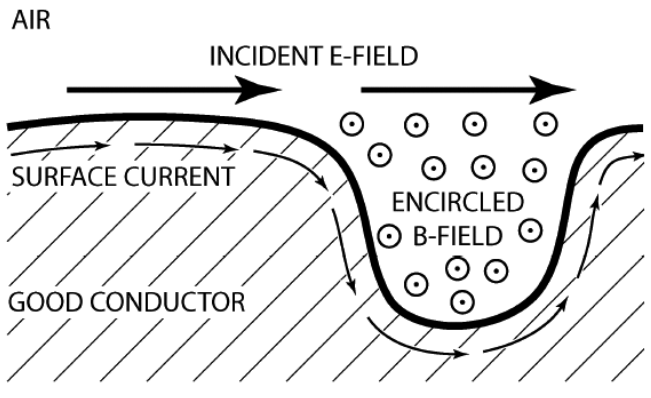


Fig. 7. Magnetic field encircled by the surface current flowing on a rough conductor and excited by the incident electric field results in substantial surface inductance, above and beyond that generated by the smooth surface skin effect.

Figure 2 • Illustration from a study (1) regarding the effects of copper surface roughness on the propagation constant and insertion loss.

depth in copper is equivalent to the copper surface roughness for even the smoothest copper at millimeter-wave frequencies.

Looking at these values in Table 1, a designer working at 50 GHz may decide that the choice of copper may not matter, since all of the copper types have a surface rougher than the skin depth, but that is a wrong assumption. The rougher surface will create more parasitic inductance and cause a change in the surface impedance as well as increase in insertion loss [1]. Figure 2 offers the results of a study showing the effects of copper surface roughness on propagation constant and insertion loss.

To further highlight the conductor roughness difference, Figure 3 shows insertion loss curves for one substrate with different copper types. Standard RO4350B™ laminate from Rogers Corp. was used with copper having an average roughness of 2.5 µm RMS, while RO4350B LoPro™ laminate was used with copper having an average roughness of 0.6 µm RMS. Although some noise is present in both curves at 50 GHz, the trend is clear, with the smoother copper yielding lower loss. There is a slight difference (0.7 mils) in thickness between the substrates, but the laminates are thin enough where the conductor losses dominate.

The plated finish applied to copper on final circuit production can also impact conductor loss, especially at higher frequencies. Unfortunately, many of the metals used as a finish for PCBs are less conductive than copper, and the addition of these finishes results in an increase in conductor loss. For example, electroless-nickel-immersion-gold (ENIG) finish is commonly used for PCBs, even though nickel is less conductive than copper and using an ENIG finish inevitably results in an increase in conductor loss. A typical conductor stack-up with an ENIG finish would start with the base copper for the circuit, then a barrier layer against copper oxide migration, which is the nickel, and on top of the nickel is the gold. In terms of thickness, immersion gold is a self-limiting process that typically produces a gold thickness of around 0.2 µm while the nickel has a thickness of 5.0 µm. For the skin depths at millimeter-wave frequencies, the nickel will be used and some of the gold.

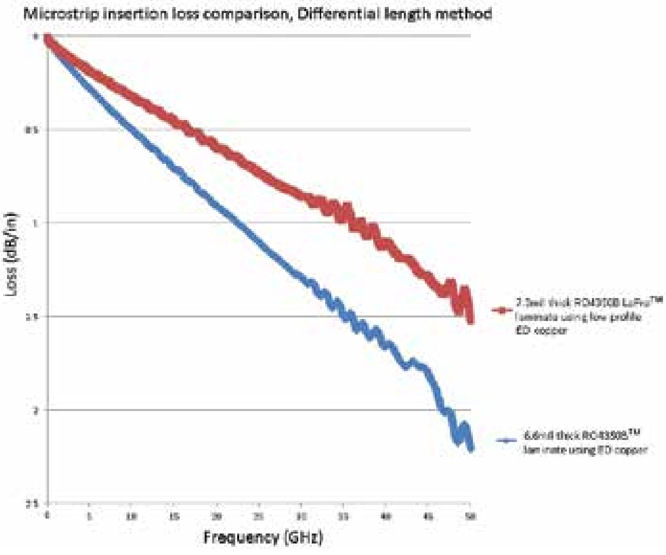


Figure 3 • Essentially the same substrate material shows two different loss curves due to differences in copper surface roughness for the substrates.

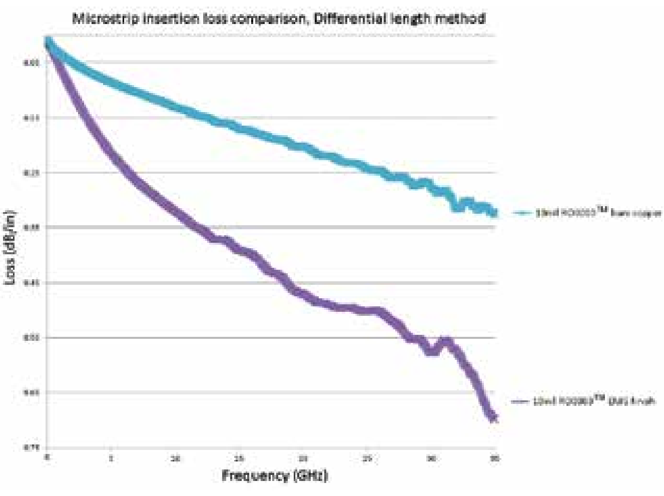


Figure 4 • These insertion loss curves were measured for microstrip transmission-line circuits using the same substrate material but with bare copper and with ENIG finish.

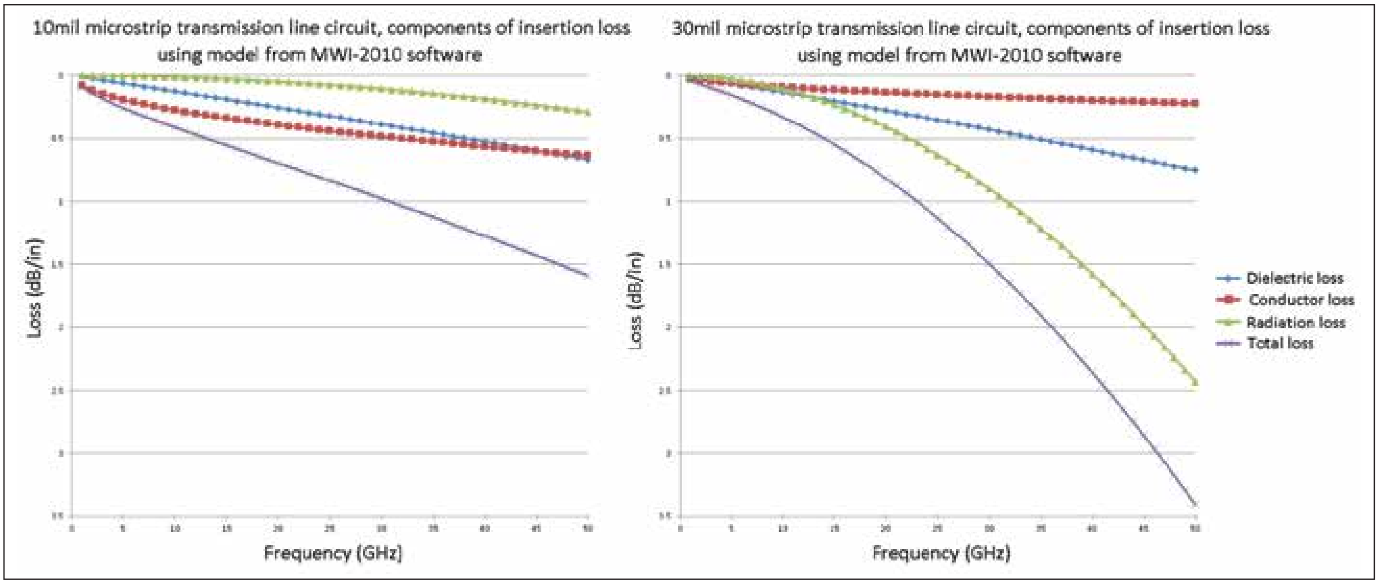


Figure 5 • Comparison of microstrip transmission line circuit models, using the same material at different thicknesses and illustrating the different components of insertion loss (total loss).

But at higher millimeter-wave frequencies, more of the gold finish will be used. But gold is still less conductive than copper, so using this finish will result in a penalty in conductor loss.

Figure 4 shows insertion loss curves for circuits made on the same material, but comparing the insertion loss for circuits with bare copper to circuits with the ENIG finish. Figure 4 helps to illustrate several issues. The material with ENIG finish shows a clear trend towards higher loss than the material with bare copper. But the loss characteristics are somewhat different at lower frequencies. This is largely because the nickel is so thick and the current density due to skin depth is using more nickel than copper or gold at the lower frequencies. At about 20 GHz, the skin depth is such that the gold is being used more. As the frequency increases, more of the gold is used and the insertion loss curve for the ENIG material starts to parallel curve for the material with bare copper.

Silver

Pure silver is more conductive than pure copper, but the immersion silver process used to add silver as a PCB finish actually employs silver alloy and not pure silver. It is close to pure silver, so the conductivity of the silver alloy is very good and close to that of copper. The immersion silver process is self-limiting, so that silver is added in a thin coating, typically in the range of 0.2 µm. Unfortunately, silver will oxidize over time, in contrast to gold which does not oxidize. Still, even though oxidation of silver will change its appearance, it apparently does not significantly impact the insertion-loss performance of a finished PCB. Studies by the author of circuits with silver oxides of more than 2.5 years in age did not show significant difference in insertion loss.

It should be pointed out that signal launch was an issue with the data shown in Figure 4. Those curves were produced with the aid of a commercial vector network analyzer (VNA) capable of measurements to 50 GHz. But the curves were cutoff at 35 GHz due to poor signal launch, as evidenced by the noise above 35 GHz. With more effective signal launch, the ENIG curve in Figure 4 could be expected to parallel the loss curve for the material with bare copper from about 25 GHz to 50 GHz and possibly higher.

As noted, insertion loss has many loss components, and understanding those components can be helpful to millimeter-wave circuit designers. To help gain this understanding, a personal computer (PC) software program, MWI-2010, available for free download from the Rogers Corp. website (www.rogerscorp.com), can show the different components of insertion loss. The program is based on work by Hammerstad and Jenson [2] outlining computer routines for modeling microstrip transmission lines for impedance and loss characteristics. The MWI-2010 software's capability to predict microstrip radiation loss, which has been found to be relatively accurate, is based on the work of Wadell [3].

Figure 5 shows different loss components of circuit insertion loss for two different circuit-material thicknesses, using the modeling power of the MWI-2010 software. The circuit model assumed the proper conductor width for a 50-Ω transmission line, using a circuit material with Dk of 3.66 and 1-oz. copper. If radiation loss is initially ignored, the relationship between dielectric loss and conductor loss is apparent. At lower microwave frequencies (below 15 GHz), the thinner 10-mil-thick circuit reveals conductor loss to be the dominant component of total insertion loss. The thicker, 30-mil-thick circuit has higher dielectric loss than conductor loss. Within this range of frequencies, a circuit designer's choice of materials based on copper (conductor loss) and dissipation factor (dielectric loss) will be based on the thickness of the circuit. Within the range of frequencies shown in Figure 5, radiation losses are not dominant, although they are significant for the 30-mil-thick circuit at 15 GHz.

Radiation Losses

Radiation losses shown in Figure 5 are dependent on both frequency and circuit thickness. At less than 15

GHz the radiation loss for the 10-mil-thick circuit is nearly insignificant while the radiation loss for the 30-mil-thick circuit is significant. This is a general rule that thinner circuits are better for minimizing radiation loss. When millimeter-wave frequencies (above 30 GHz) are considered, radiation loss can contribute a significant amount to total loss for a thicker circuit compared to a thinner circuit.

In addition to PCB material thickness, radiation loss is also dependent on a PCB material's Dk value. Circuit materials with higher Dk values tend to exhibit less radiation losses than those with lower Dk values, although typically at the cost of higher conductor loss. Also, to achieve the same impedance, narrower signal conductors are needed on a material with higher Dk value than on one with lower Dk value, and narrower conductors will suffer higher conductor loss than wider conductors.

Circuit design will impact radiation loss, as any impedance mismatch will typically have radiation loss associated with it. Impedance mismatches are not uncommon in RF/microwave circuits, and can depend on circuit configuration. Stripline circuits, for example, typically exhibit no radiation losses, while microstrip circuits, such as the circuit of Figure 5, can be prone to radiation loss, dependent on circuit thickness and other issues. When radiation losses are a concern, a GCPW circuit design can limit radiation losses at millimeter-wave frequencies. This has been detailed in a study on optimizing the signal launch at 50 GHz for GCPW as well as other circuit approaches [4].

The signal launch, of course, is a key element to achieving good performance at higher frequencies, such as in millimeter-wave circuits. Signal launching and radiation losses are related, since an effective signal launch, in which energy makes a transition from one wave propagation mode to another, will yield minimal radiation loss. For example, a typical RF connector operates in a transverse-electric (TE) mode while a planar PCB operates in a transverse-electromagnetic (TEM) propagation mode. A GCPW or microstrip circuit can operate in a quasi-TEM mode while stripline can work in a true TEM mode. At any change in mode propagation, such as where the connector meets the circuit board, a transition is made and any stray reactances or impedance mismatches can lead to radiation losses.

Designers of millimeter-wave circuits for high-frequency applications should not hesitate to contact their suppliers of high-frequency circuit materials to better understand the tradeoffs presented by different materials at these higher frequencies and the options that are available in different PCB materials for millimeter-wave applications. Numerous dielectric substrates are available with different copper types and different measures of surface roughness. There are often many choices of circuit materials within a single product family, in terms of Dk value and dissipation factor. Manufacturers of high-frequency circuit materials are generally willing to work closely to help circuit designers achieve optimum performance goals on new and existing microwave and millimeter-wave applications.

About the Author:

John Coonrod is Market Development Engineer for the Advanced Circuit Materials Division of Rogers Corporation, Chandler, AZ. He can be reached at john.coonrod@rogerscorp.com.

References:

[1] Allen F. Horn, John W. Reynolds, and James C. Rautio, "Conductor Profile Effects on the Propagation Constant of Microstrip Transmission Lines," IEEE MTT-S International Microwave Symposium, 2010.

[2] E. Hammerstad and O. Jenson, "Accurate models of microstrip computer aided design," 1980 MTT-S International Microwave Symposium Digest, May 1980, pp. 407-409.

[3] Brian C. Wadell, "Transmission Line Design Handbook," Artech House, Norwood, MA, 1991, p. 99.

[4] Bill Rosas, "Optimizing Test Boards for 50 GHz End Launch Connectors: Grounded Coplanar Launches and Through Lines on 30-mil RO4350B with Comparison to Microstrip," Southwest Microwave, Inc., Tempe, AZ, 2007, www.southwestmicrowave.com.