Understanding PCBs for HIGH-FREQUENCY Applications

Electrical performance at microwave frequencies can be aided by the surface finish and laminate choice. **by JOHN COONROD**

Printed circuit boards for microwave circuits must meet a special set of requirements. They must support transmission of signals with the diminutive wavelengths of radio-frequency (RF) and microwave frequencies, and they must do so with minimal loss and stable, consistent performance. To best understand high-frequency PCBs, it helps to review the types of transmission lines and structures typically used in RF/microwave circuits and how PCB characteristics relate to electrical performance at higher frequencies.

The three most common transmission-line technologies used in microwave circuits are microstrip, coplanar and stripline circuits. Of the three, microstrip transmission lines are most often used in high-frequency PCBs, since they are relatively simple to fabricate and with fewer electrical variables to consider than the other two approaches. **FIGURE 1** shows a simple drawing of a microstrip PCB. A microstrip consists of a conductive strip and a wider ground plane, separated by a dielectric layer.

Electromagnetic propagation in a microstrip circuit occurs by means of transverse-electromagnetic (TEM) plane waves. In an ideal microstrip circuit, signal energy propagates perpendicular to the electric (E) and magnetic (H) fields. In an actual microstrip circuit, because propagation also takes place in the dielectric material between the conductors, as well as in the air above the conductors, propagation occurs in a quasi-TEM mode.

There are four types of signal losses in a microstrip transmission line: conductor, dielectric, radiation, and leakage losses. Leakage losses usually are not a concern, due to the high volume resistivity (resistance) of PCB materials used for microwave circuits. At microwave frequencies, radiation losses tend to be more of an issue for microstrip circuits than for coplanar or stripline circuits. Dielectric losses are a function of the PCB substrate material; in terms of loss performance, different materials can be compared by a parameter known as dissipation factor. Lower values of dissipation factor signify laminates with lower dielectric losses. Conductor losses are not quite as simple to size up because they are linked to a number of different variables in a microstrip circuit.

Conductor losses are related to the way that current flows along a conductor. Known as "skin effect," current will tend to flow closer to the surface of a conductor at higher frequencies. As frequency increases, the skin depth is less and a signal's current flows along the conductor by using less of the conductor at higher frequencies. Most of the current density in a microstrip signal conductor is at the interface between the two copper planes and at the bottom corners of the conductor (**FIGURE 2**).

Skin depth, δ, can be calculated from the equality in Eq. 1:

$$\delta = [1/(2\pi f\mu\sigma)]0.5 \quad \text{(Eq. 1)}$$

where f is frequency, μ is the permeability of the conductor,

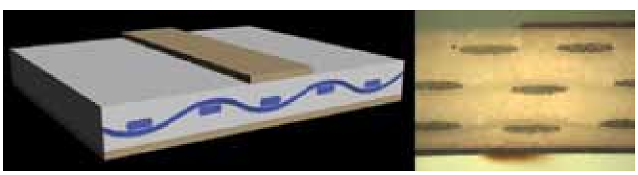

FIGURE 1. Drawings of a microstrip as a 3D representation (left) and a cross-sectional view (right).

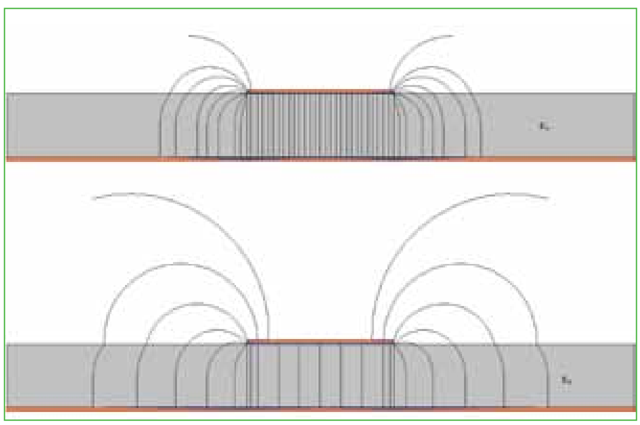

FIGURE 2. A comparison of electric field lines and current density (blue) for a microstrip transmission line at higher (top) and lower frequencies (bottom).

and σ is the conductivity of the conductor, typically copper in an RF/microwave microstrip PCB. (The conductivity of copper is generally accepted to be about 5.8×10^7 S/m.) The parameter for permeability in Eq. 1 is actually a complex quantity given by $\mu = \mu_0\mu_r$, where μ_0 is the permeability of free space and μ_r is a multiplier related to the type of metal used as the conductor. For copper, the value of μ_r is assumed to be about unity (1), although there are some exceptions, as well as issues related to different conductivity values.

For example, the surface roughness of the copper conductor, at a circuit's copper-substrate interface, can impact the loss of a microstrip circuit. A rougher conductor surface suffers higher losses. Several methods have been developed to account for the copper roughness, and a simple model is the Morgan rule,[1] which is a multiplier of the conductor losses (α_c). Generalized conductor loss and the Morgan rule are given by Eqs. 2 and 3, respectively:

$$\alpha_c = 1/\delta \quad \text{(Eq. 2)}$$

$$\alpha_{c + \text{roughness}} = \alpha_c\{1 + (2/\pi)[\tan^{-1}(\Delta/\delta)^2]\} \quad \text{(Eq. 3)}$$

where $\alpha_{c + \text{roughness}}$ is the total conductor loss, including loss due to copper conductor roughness. Parameter Δ in Eq. 3 represents the root-mean-square (RMS) surface roughness of the copper conductor. As with many models, the Morgan rule is limited at certain frequencies, and is typically more accurate at frequencies of less than 10GHz.

How do these models and relationships translate into actual PCB applications? In looking at Eq. 1, it is apparent that skin depth decreases with increasing frequency. In Eq. 2, as the skin depth, δ, decreases, the conductor loss, α_c, will increase. Higher frequencies translate into higher conductor losses. As frequencies increase, the effects of copper conductor surface roughness also increase, to a point where Eq. 3 will reach a saturation point at its highest value.

To demonstrate the effects of copper conductor surface roughness, **FIGURE 3** offers a comparison of the same dielectric substrate, but with two different levels of copper surface roughness. In both cases, microstrip transmission lines were fabricated on the PCBs. The PCB substrate with the rougher copper conductor surface is standard Rogers RO4350B laminate material, while the substrate with the smoother copper conductor surface is RO4350B LoPro. The curves for the two microstrip transmission lines in Figure 3 are also compared to a microstrip model[2] using the Morgan rule, with good correlation at lower frequencies.

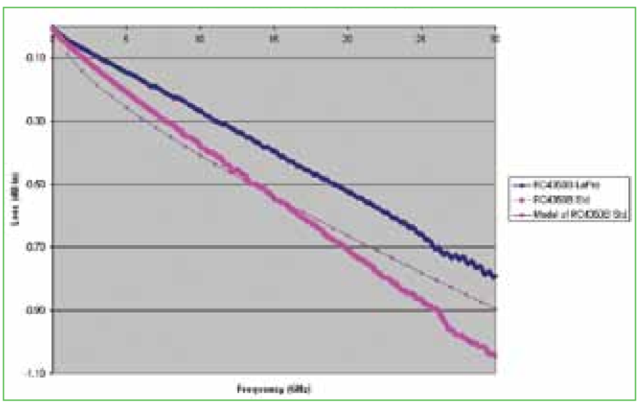

FIGURE 3. Insertion-loss responses for identical 0.010" microstrip circuits fabricated on a standard RO4350B laminate with normal copper conductor surface roughness and RO4350B LoPro laminate. The third response was generated by Rogers' MWI-2010 Microwave Impedance Calculator, which uses the Morgan rule to account for conductor surface roughness.

Different types of PCB conductor finish can also provide different results in terms of conductor losses at higher frequencies. For example, an electroless-nickel/immersion-gold (ENIG) plated finish is often used on copper conductors. As Figure 2 shows, most electric fields in a high-frequency microstrip transmission line lie between the copper layers, although a significant current density exists at the corners of the signal conductor. ENIG plating affects EM fields at a conductor's edges and corners. At lower frequencies, current flows within the skin of the conductor and uses the copper. But at higher frequencies, where the skin depth is less, most of the current density is concentrated in the NiAu finish of an ENIG-plated conductor. This gold plating is typically very thin, 10 microinches or less, while the nickel layer is considerably thicker, typically 150 to 300 microinches.

Gold is slightly less conductive than copper and has no ferromagnetic properties ($\mu_r = 1$), so it has relatively little impact on the conductor's loss characteristics at higher frequencies. But nickel is much less conductive than copper (about one-third that of copper), and nickel also has strong ferromagnetic properties, with a high permeability value and with μ_r value of about 500. Lower conductivity will increase conductor loss. High μ_r value will decrease the skin depth (per Eq. 1) and keep the current density in the conductor within a narrow region of low-conductivity nickel. To minimize this effect, PCB suppliers typically use one of a number of different ENIG processes, often with a form of nickel alloy, to minimize unwanted ferromagnetic properties in conductors.

The manner in which copper is treated in the process of making a PCB's copper foil can impact conductor losses. For example, when two PCBs with different copper types but with nearly identical conductor surface roughness profiles were evaluated, they were found to have very different loss responses. The copper with the inferior loss performance was found to have undergone a nickel allow treatment. In general, a conductor composed of ore treated with a ferromagnetic material will exhibit degraded conductor losses in microwave transmission lines.

Numerous PCB variables influence the impedance of a microstrip transmission line, such as laminate dielectric constant (known as Dk, ε_r, relative permittivity), thickness, copper weight, and control of circuit etching. For high-frequency applications, it is important that a PCB laminate have well-controlled Dk, as well as tightly controlled thickness, since variations in either will result in variations in transmission-line impedance.

A number of other factors can influence the impedance of a high-frequency PCB's transmission lines. Dispersion, for example, is often overlooked. Dispersion is a microstrip transmission-line property in which the propagation characteristics are different at lower frequencies than at higher frequencies. Dispersion can also be a concern in PCB materials where the Dk value is considerably different at lower and higher frequencies. Dispersion typically plagues PCB laminates not nominally engineered for high-frequency applications, but is minimized in higher-quality PCB materials meant for high-frequency circuits. To demonstrate differences in dispersion characteristics for different materials, **FIGURE 4** compares high-performance FR-4 substrate with RO4350B laminate, both with microstrip transmission lines fabricated on 0.020"-thick substrates.

Environmental conditions can also play a role in how well a PCB material maintains impedance, especially at higher frequencies. Many traditional PCB materials may not have been formulated for stable Dk performance in changing or hostile environments. All PCB materials are characterized by a parameter known as thermal coefficient of dielectric constant, or TCDk, in units of ppm/°C. This parameter describes how much the dielectric constant will change with changes in temperature. These changes in Dk will also change the impedance of the microstrip transmission lines, so lower values of TCDk (resulting in minimal effects on impedance) are preferred. For example, it is not unusual for standard FR-4 to exhibit a TCDk value of 200 ppm/°C or more. In contrast, many high-frequency PCB laminates are engineered to exhibit a TCDk value of 50 ppm/°C or less.

Humidity can also affect PCB performance. If a PCB material is prone to absorb moisture, the water content can impact loss performance and impedance stability. Many standard PCB laminates have moisture absorption values of 2% or more, which means in a humid environment, the laminate can absorb moisture readily, and the electrical properties change. Compared to PCB materials, the Dk of water is very high (about 70). In an environment with high humidity, excessive moisture absorption can raise a PCB material's Dk and increase its dielectric loss. PCB materials formulated for high-frequency use typically exhibit low moisture absorption, with values of 0.2% or less.

Microstrip is probably the most popular high-frequency transmission-line technology, but coplanar transmission lines are also widely used in RF/microwave circuit designs. There are many different variants of coplanar transmission lines. The coplanar structure most often used in high-frequency circuits is known as coplanar waveguide (CPW) or specifically conductor-backed coplanar waveguide (CBCPW). **FIGURE 5** presents a simple drawing of a CBCPW transmission line.

CBCPW transmission lines offer a number of benefits compared to microstrip, including much lower radiation losses and very low dispersion. CBCPW transmission lines can support extremely wide bandwidths, as well as a wide range of impedance values, for ease of matching to low-impedance devices such as microwave power transistors. The primary limitations of CBCPW transmission lines have to do with their inherently higher conductor losses compared to microstrip, and the need for forming plated through-hole (PTH) via holes for signal and ground connections between circuit layers.

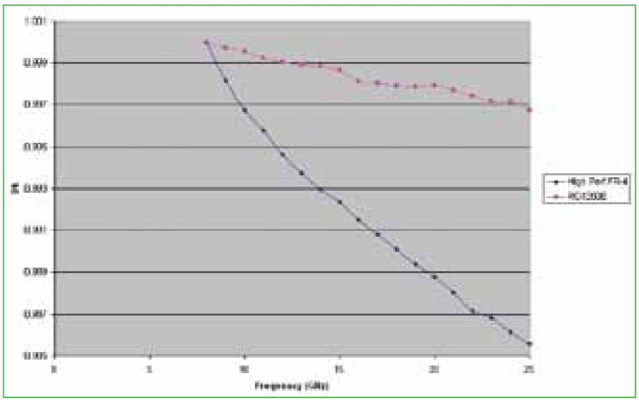

FIGURE 4. Comparison of dispersion for 0.020"-thick microstrip transmission lines using phase length differential method.

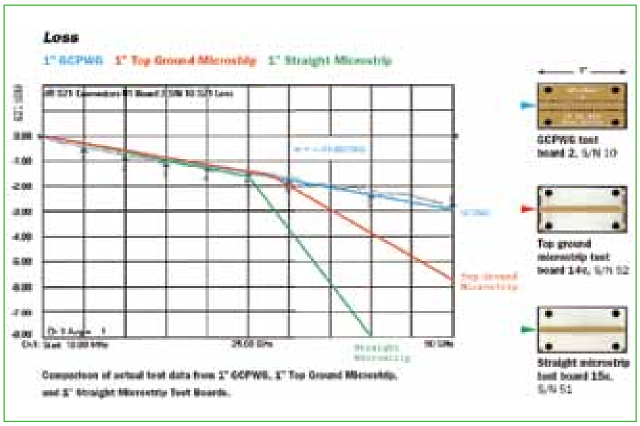

FIGURE 6. A study[3] revealed differences among conventional microstrip, microstrip with a top ground launch or coplanar launch, and CBCPW (referred to as GCPWG in the study) transmission lines.

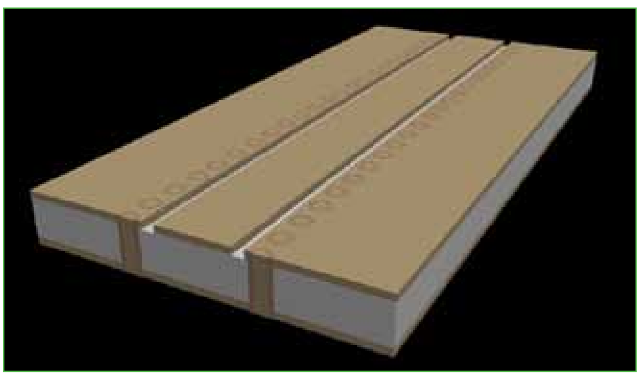

FIGURE 5. This simple drawing shows a conductor-backed coplanar waveguide (CBCPW).

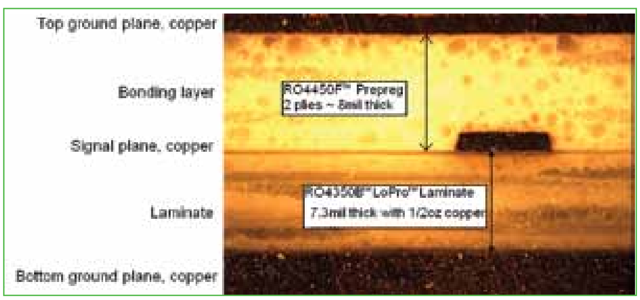

FIGURE 7. This magnified image shows a cross-section of a stripline transmission line.

Microstrip radiation losses can be significant above certain frequencies and/or with certain circuit geometries. At very high frequencies, radiation losses can dominate the performance of a microstrip circuit and negate the benefits of using conductors with smooth copper or laminate material with low dissipation factor. One way to avoid the high radiation losses of microstrip at high frequencies is through the use of CBCPW transmission lines. When properly designed, CBCPW transmission lines can support quasi-TEM wave propagation at very high frequencies, beyond the frequency limit of microstrip. This can be seen in the results of a study performed by Southwest Microwave, Inc. (southwestmicrowave.com) comparing different transmission-line structures at test frequencies through 50GHz (**FIGURE 6**). The "knee" in the loss curve for the microstrip structures shown in Figure 6 is where radiation losses become dominant. When properly designed, a coplanar transmission-line structure does not exhibit this frequency dependency.

Stripline is probably the most stable of the three main high-frequency transmission lines. Sometimes called flat coaxial transmission line, it features a signal layer sandwiched between top and bottom ground planes. In contrast to microstrip, stripline has numerous benefits, including no radiation losses and no dispersion. It can support true TEM wave propagation and is capable of extremely wideband frequency performance. With its double ground plane and buried signal structure, external electrical influences have little or no effect on stripline circuits.

There are also drawbacks to stripline transmission-line technology. Fabrication costs for stripline are higher than those for microstrip or CBCPW transmission-line structures. Stripline is also more limited to the range of possible impedance values, and signal losses in stripline are higher than for either microstrip or CPCPW circuits. **FIGURE 7** shows a magnified cross-sectional view of a stripline structure.

Stripline transmission lines suffer higher loss compared to microstrip transmission lines because microstrip benefits from partial wave propagation through the air above the circuit; the dielectric losses of air are lower than those of the laminate materials surrounding the conductor layer in a stripline circuit. A stripline circuit structure will also use a narrower signal conductor for a given impedance, such as the 50Ω typically used in microwave circuits, than a microstrip circuit structure, and the narrower conductor will result in higher conductor losses compared to the wider conductor used in microstrip. Compared to microstrip, a stripline circuit will be affected more by the copper conductor surface because of the two ground return paths. A smooth copper conductor surface can provide performance benefits, whereas a rough copper conductor surface can contribute to higher conductor losses.

This brief comparison of three high-frequency transmission-line types has offered some insight into the PCB material characteristics that can affect high-frequency performance, such as conductor losses. By better understanding the benefits of circuit materials formulated for high-frequency applications, circuit designers can more readily achieve their final goals in terms of electrical performance at RF/microwave frequencies. **PCD&F**

REFERENCES

1. S. P. Morgan, "Effect of Surface Roughness on Eddy Current Losses at Microwave Frequencies," *Journal of Applied Physics*, v. 20, 1949, p. 352.
2. MWI-2010 Impedance model (download at rogerscorp.com/acm). The microstrip model uses the Hammerstad and Jenson model: E. Hammerstad and O. Jenson, "Accurate models of microstrip computer aided design," 1980 Microwave Theory & Techniques Symposium (MTT-S) International Symposium Digest, May 1980, pp. 407-409.
3. Bill Rosas, "Optimizing Test Boards for 50 GHz End Launch Connectors: Grounded Coplanar Launches and Through Lines on 30mil Rogers RO4350B with Comparison to Microstrip," Southwest Microwave Inc., 2007.

JOHN COONROD is a market development engineer at Rogers Corp., Advanced Circuit Materials Division (rogers.com); john.coonrod@rogerscorp.com.