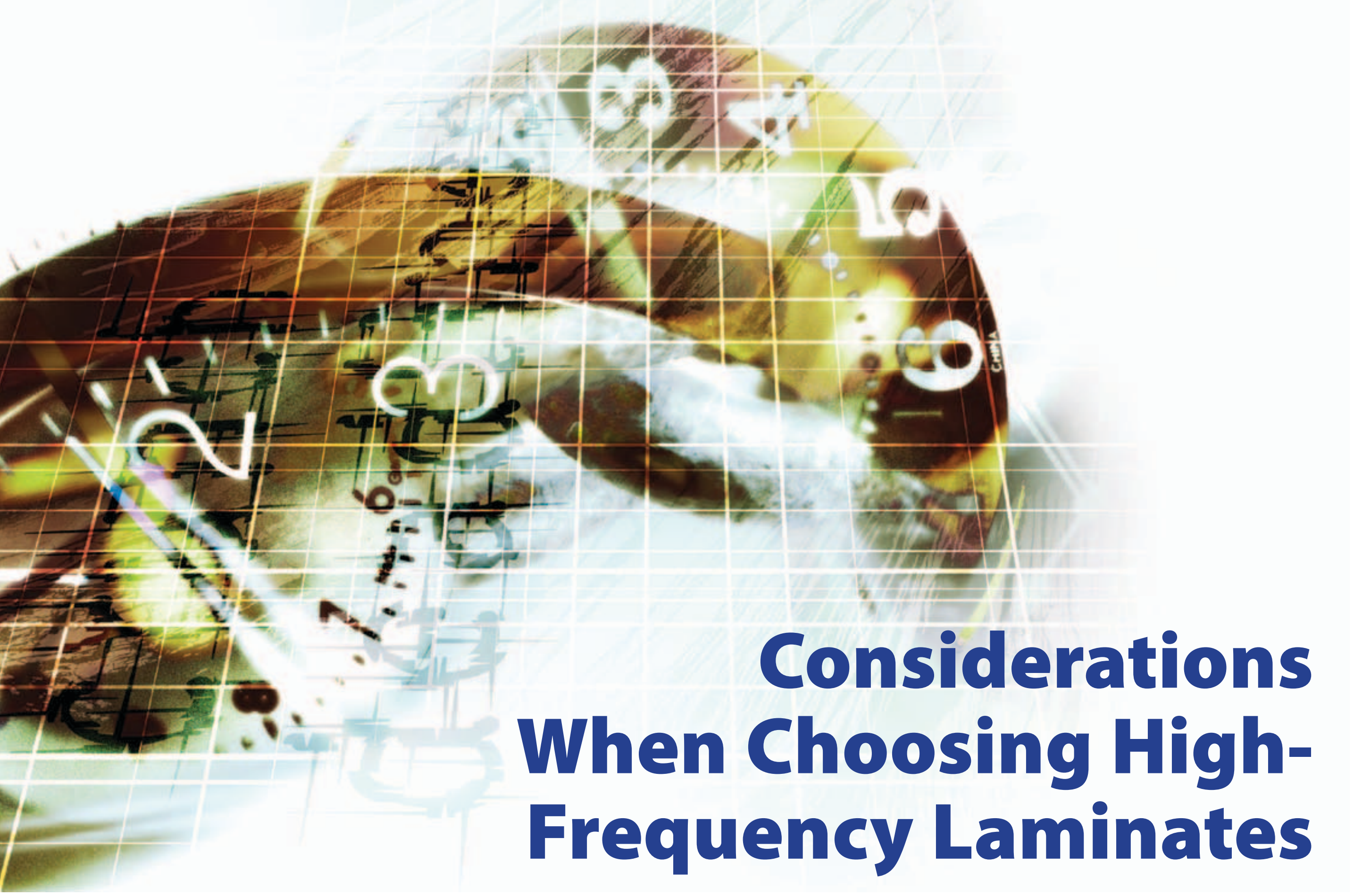

Considerations When Choosing High-Frequency Laminates

John Coonrod

Choosing a laminate for a high-frequency printed circuit board (PCB) can impact the circuit fabrication process, the assembly process, and the end-use performance. For example, a material that is easy to process may not provide the best electrical performance. Selecting the right laminate for an application is a matter of understanding the requirements of a particular design project and the tradeoffs represented by different materials in terms of processing, fabrication, and end-use performance.

Even for fairly simple circuit constructions such as a double-sided PCB, a laminate should be carefully chosen. One of the more popular materials, FR-4, is well characterized in terms of circuit fabrication processing, including for drilling, plating, circuit etching, and finish. The material provides predictable performance and production yields for many applications, but may not be an optimal choice for circuits demanding low loss. Also, circuits requiring tight control of impedance may be better served with a high-frequency laminate. When a particular type of FR-4 is produced at multiple locations, different types and concentrations of glass filler may be used, resulting in inconsistent values of dielectric constant (Dk) from laminate panel to panel. Also, the thickness of FR-4 laminates may not be as tightly controlled compared to high-frequency laminates. Variations in thickness and dielectric constant can result in yield issues for PCB fabricators for boards with narrow impedance tolerances. Although high-frequency laminates have higher costs than FR-4 materials, the improved yields of high-frequency laminates for boards requiring tightly controlled impedances may more than offset the price differences between the two types of laminates.

Of course, all high-frequency laminates are not created equal. Some may offer excellent electrical performance but may be difficult and costly to process for PCB fabrication. For example, nearly pure polytetrafluoroethylene (PTFE) substrates provide outstanding electrical performance in terms of consistent dielectric constant and low loss, but these soft substrates exhibit dimensional instability, are difficult to process in terms of drilling and forming plated through holes (PTHs), and can suffer handling damage. While numerous techniques have been developed to overcome the shortcomings of PTFE, the material also exhibits a relatively high coefficient of thermal expansion (CTE) in the z (thickness) axis, which can result in reliability issues for PTHs in higher-layer-count multilayer PCBs. To improve the reliability of PTFE materials in multilayer applications, materials suppliers have developed special PTFE-based laminates with different filler materials to improve robustness and lower the CTE.

Transmission lines provide an effective means of evaluating the capabilities of different laminate materials in terms of fabrication, assembly, and performance. Microstrip transmission lines, for example, are commonly fabricated on double-sided PCBs. In a simple representation (Figure 1), a microstrip transmission line consists of a conductive signal plane on the top side of the PCB and a ground plane on the bottom side of the PCB.

Loss is an important parameter in a growing number of high-frequency designs and PCB fabricators may need to soon provide boards with loss measurements for well characterized structures, such as microstrip lines. A number of different losses are associated with a microstrip line, including dielectric losses, conductor losses, and radiation losses. Dielectric losses are associated with the dissipation factor (Df) of the laminate. Conductor losses are associated with the circuit design as well as the quality of the copper used for the laminate. Radiation losses are associated with the microstrip design, the laminate thickness, and the operating frequency.

Figure 2 presents a comparison of four different circuit materials, using microstrip-based reference circuitry. A PTFE-based material with special filler to improve PCB fabrication exhibits the lowest loss[1]. A high-performance FR-4 laminate suffers the highest loss of the four materials. In between, two materials not based on PTFE[2] employ the same base dielectric material, with one[3] using a special copper formulation to reduce conductor losses.

Radiation losses are typically a function of frequency and laminate thickness, as demonstrated in Table 1 where laminate thicknesses are shown for frequencies at which radiation losses become significant. The type of application will also determine if the amount of loss for a given frequency can be tolerated. The dielectric constant of the laminate and how well the circuit is impedance matched can also have significant effects on radiation losses. Radiation losses can be minimized by using a laminate with higher dielectric constant and by improving the source and load impedance matches to the PCB.

Radiation losses can also be reduced by transmission line structures other than microstrip. For example, conductor-backed coplanar waveguide (CBCPW) transmission lines typically exhibit less radiation loss than microstrip for a given laminate material. Although CBCPW appears much like microstrip, it uses ground planes on the same layer as the conductors and adjacent to those conductive lines (Figure 3).

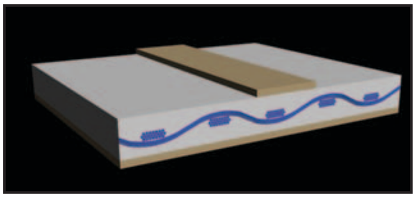

Fig 1 Microstrip Transmission Line

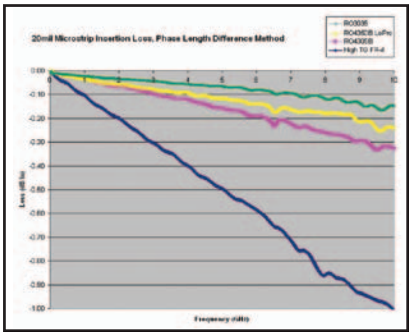

Fig 2 Microstrip Insertion Loss

Although CBCPW exhibits lower radiation losses than microstrip, it typically suffers higher conductor losses than microstrip for a given laminate material. CBCPW transmission lines also offer a wider range of impedances than microstrip transmission lines. For microstrip, conductor width and substrate thickness affect impedance. For CBCPW, those factors, along with the spacing on the signal plane between the signal conductor and the adjacent ground planes, will affect the impedance. CBCPW transmission lines can achieve higher impedances than microstrip lines for a given laminate thickness, allowing the user of thinner laminates with CBCPW.

Laminate Thickness	Radiation Loss Freq. (GHz)
5mil	> 100
10mil	80
20mil	35
30mil	20
60mil	5

Table 1 Laminate thickness and approximate frequency where radiation losses become significant, assuming a substrate with an approximate Dk of 3.0 and low Df.

Laminate Thickness (mils)	Material dk	Conductor Width (mils)
2	4.5	3.3
2	2.2	5.6
3	10	1.5
3	4.5	5.1
3	2.2	8.6
4	4.5	7
4	2.2	11.7
5	4.5	8.9
5	2.2	14.8

Table 2 Example of dielectric constant versus conductor widths for a 50-ohm controlled impedance line.

Thinner laminates are important for higher-frequency circuits to minimize the generation of unwanted traveling waves on the transmission lines. These unwanted traveling waves can interfere with the intended wave propagation of the circuit.

Thinner laminates typically have more issues with scaling or dimensional stability during the circuit fabrication process. They can also suffer handling damages or other difficulties as they approach the limits of automated handling and fabrication equipment. Thinner laminates also require narrower conductors to achieve controlled impedances and these narrower conductors can hinder PCB yields. However, by choosing a high-frequency laminate with a lower dielectric constant, it is possible to increase the conductor width for a thin laminate while still maintaining tight control over impedance. Table 2 provides examples of conductor widths required for laminates with different dielectric constants and thicknesses when maintaining a controlled 50-ohm impedance. The use of lower dielectric-constant materials allows the use of wider conductors, and wider conductors also result in reduced conductor losses.

Multilayer PCBs consist of a variety of different circuit constructions, with three-conductor stripline among the most common.

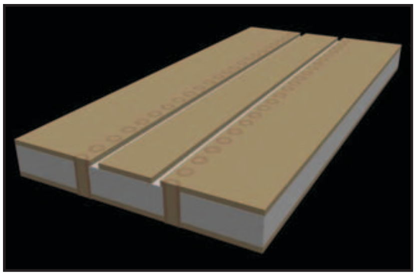

Fig 3 Conductor Back Coplanar Waveguide (CBCPW)

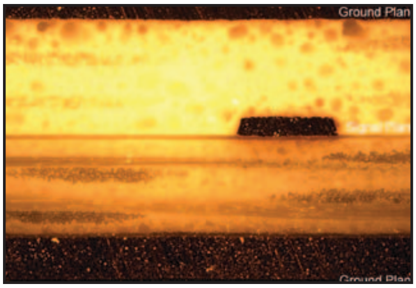

Fig 4 Cross-Sectional View of a Stripline Circuit

Multilayer circuits may include stripline circuit layers with other layers providing power distribution, digital signal processing, and other functions. Often, transmission-line approaches are mixed, with microstrip circuits on the outside two copper layers and stripline circuits on the inner layers.

Stripline (Figure 4) offers numerous electrical performance advantages compared to microstrip and CBCPW. It does not typically suffer from radiation losses, with better overall electrical performance than microstrip and CBCPW. But it is more complex to fabricate than either of these two transmission-line types and with added complexity can come reliability issues.

The buried conductor configuration of stripline can pose challenges for PCB fabricators when forming PTHs. For a simple double-sided PCB, the PTHs need not make good electrical contact with a buried conductor inner layer. For stripline circuitry, the PTHs are required to make a reliable copper plated contact with the buried inner conductor layer by means of the PTHs. The layer-to-layer alignment in a stripline board is also critical compared to simple alignment of a double-sided PCB by means of imaging process. For stripline, the alignment of the layers can be affected by dimensional instability of the laminate materials. Three-copper-layer stripline circuits are usually formed by means of a bonding layer. The stability and other parameters of this bonding layer material must be closely scrutinized when considering laminate materials for stripline circuits.

Stripline can also be difficult to fabricate with PTFE-based laminates, which require nonstandard processing steps. For example, PTFE is difficult to plate with the copper required for forming PTHs for making contact with the inner conductor layer. After drilling a through hole, the exposed PTFE must be prepared to accept copper plating, a condition known as "made wettable." Ceramic-filled PTFE laminates can typically use a special plasma process to make a through hole wettable, but nearly pure PTFE laminates require a special process to make them wettable.

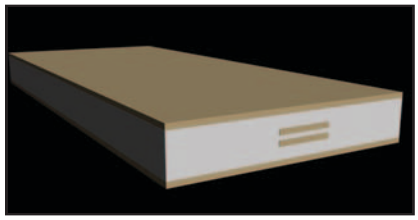

Fig 5 Cross-sectional View of a Broadside Stripline

In addition to minimal radiation losses, a stripline circuit's top and bottom ground planes form a more efficient ground return path with less surface resistance than either microstrip or CBCPW. Transmission-line dispersion is negligible with stripline, if the proper materials are used and the design is done correctly. Transmission-line dispersion occurs when a propagating wave behaves differently at different frequencies. The phenomenon is not unusual with microstrip and is due to a propagating wave being affected by different media. The substrate is the medium for microstrip between the signal conductor and the ground plane, while the air is the medium above the signal conductor. For stripline based on the same laminate, the propagating wave is affected by a single medium so dispersion is not an issue. Dispersion is an important consideration for high-speed digital circuits in which fundamental and higher-order harmonic signals form digital signals. Excessive dispersion can cause irregular propagation of these component signals, resulting in digital signal distortion at higher frequencies and speeds.

To reduce costs, many multilayer PCBs are actually hybrids of several different types of laminates, using higher-performance materials where circuit performance is critical, and lower-cost materials such as FR-4 for less critical functions, such as power distribution. Among the concerns of fabricating a hybrid multilayer PCB is the mismatch of CTE values among the different laminates. Although severe mismatches in CTE values can lead to reliability issues, more often it can cause problems in the flatness of the final PCB. To minimize this problem, special lamination cycles are used to provide a stress relief at the end of the cure cycle that allows the multilayer PCB to relax to a final flat state.

Through-hole drilling can also be an issue with these hybrid multilayer PCBs, for example, when drilling a hole through a soft bonding layer surrounded by two rigid core materials. The drill bit may stretch the soft material into the through-hole barrel, resulting in a poor interconnection to the inner copper conductor layers. To avoid such issues, bonding layers must be carefully selected and drilling through holes conducted with great care. Some bonding materials will need a rougher surface for good bonding and some materials will need special lamination cycles.

A hybrid multilayer PCB can be formed with different dielectric-constant layers to achieve a wide range of electrical performance, for example, mixing low-dielec-

Transmission Line Type	PCB Fabrication	Fabrication Cost	Dispersion	Losses	Impedance Range	Bandwidth
Microstrip	Simple	Low	High	Moderate	Moderate	Low to Moderate
Stripline	Moderate	Moderate	Low to None	Low	Low	High
Conductor Back Coplanar	Simple to Moderate	Low to Moderate	Moderate	Moderate	High	Moderate
Microstrip, Edge Coupled	Moderate	Low	Moderate	Moderate	High	Moderate
Stripline, Edge Coupled	Moderate	Moderate	Low to None	Low to Moderate	Moderate	High
Stripline, Broadside Coupled	Complex	Moderate to High	Low to None	Low to Moderate	Moderate	High
Offset Stripline	Moderate	Moderate	Low to None	Low	Moderate	High

Table 3 Comparison of Transmission Lines for Fabrication and Electrical Considerations

tric-constant layers with higher-dielectric-constant layers. Some layers might be used for a broadside coupled filter based on low-dielectric-constant laminate while other layers of the same PCB used for the same filter configuration with a higher-dielectric-constant material. This gives more range to the filter circuits and can alter the coupling significantly. Figure 5 shows a cross-sectional view of a stripline broadside coupled structure as might be used in a hybrid multilayer PCB.

Hybrid multilayer PCBs have also been formed with combinations of rigid and flexible laminates for special applications. Rigid high-frequency laminates are often combined with a flexible material such as a liquid-crystal polymer (LCP) or thermoplastic circuit materials[4]. The LCP[5] can be used as a cover film to the thermoplastic circuit materials laminate. The flexible LCP and thermoplastic circuit materials are relatively thin, with a non-glass weave and good electrical properties. Connectors are not needed as transitions from the rigid layers to the flexible layers since connections are built into the circuits. Eliminating the need for connector components reduces overall cost while simplifying assembly and improving reliability. By following proper design practices, impedances are matched in the transitions between different material types to ensure clean signal transitions.

> **By following proper design practices, impedances are matched in the transitions between different material types to ensure clean signal transitions.**

In summary, Table 3 compares fabrication and electrical performance issues for different transmission-line circuits. Although subjective, the comparison can serve as a quick reference when selecting laminates for different applications. □

References

1. RO3035™ laminate from Rogers Corporation
2. RO4350B™ and RO4350B LoProTM laminates, both from Rogers Corporation
3. RO4350B LoPro laminate
4. SYRON™ 7000 laminates
5. UTRALAM® 3000 laminates from Rogers Corporation

Note: RO4350B, LoPro, RO3035, ULTRALAM and SYRON are licensed trademarks of Rogers Corporation.

John Coonrod is with Rogers Corporation, Advanced Circuit Materials Division
www.rogerscorp.com