From March 2003 *High Frequency Electronics*

Digital Signal Integrity, Circuit Board Design and the Interconnect Interface

By Dale Reed
Trompeter Electronics

At microwave frequencies, all signals are affected by the physical construction of circuits and interconnections. This article describes how these characteristics affect the performance of high-speed digital signals

The objective of this article is to share valuable truths learned over the course of a career watching and working with design engineers as they deal with signal transport issues in real world situations. Without using calculus or detailed technical charts, our objective is to understand and offer clarity on the reasons why high frequency board level fabrication and packaging is not "business as usual" for high-speed digital signals, and why getting the signal on and off the board has our interest.

This issue's cover represents the practical circuit construction issues that affect the performance of microwave frequency digital signals.

Components and Their Interconnections

To examine the world of electronics hardware, one useful perspective is to categorize the various pieces of the mix as (1) components and (2) connecting elements that hook components together. From this perspective, components include integrated circuits, resistors, capacitors, and other devices that "do something to" or "respond to" the electronic signal. The connecting elements include the set of transmission lines that enable the functionality of these components. (While wireless transmission lines do exist, for the purposes of this article our focus is with wired—or, *wireline*—transmission lines). Interconnects are the junctions between the components and the transmission lines. Some are separable (involving connectors) and some are not (solder joints). All interconnections are potential sources of degradation to the performance of the transmission line.

There are two fundamental types of wired transmission lines used in digital signal transport, whether they are from component to component or from device to device. Differentiated by the physics involved, they carry either (1) light pulsed on or off (optical fiber) or (2) an electrical sine wave, shaped by the harmonics of the waveform to look like a square wave (copper).

Fiber has proven to be a wonderful high capacity solution for long haul transport of digital data signals because it can carry signals over dramatically higher distances without requiring costly amplification or signal regeneration. The application of wave division multiplexing utilizes frequency stacking technology, a trick from the copper world, to give fiber bandwidth an additional boost. The downside of fiber is that before binary code can be used for logic devices like programmed integrated circuits, it requires conversion to light—the on/off pulses of the source laser—then, at the other end of the fiber, another conversion back into its electrical counterpart. This is referred to as OEO (electrical/optical/electrical) conversion. We now understand that the hype and promise of the pure photonics end-to-end optical network is likely a decade off, largely because logic and switching/routing functions are still best solved in copper technology—for both technical and economic reasons. Copper technology, while mature, provides virtually all of the component-to-component and the lion's share of the device-to-device transmission line assignment.

To be sure, as line rates have risen in frequency and in data rate (interrelated), new problems have surfaced. The signal limits of copper are now better understood than ever before, and this article explores some of those limitations, how they are overcome, and suggests that progress will continue in this field.

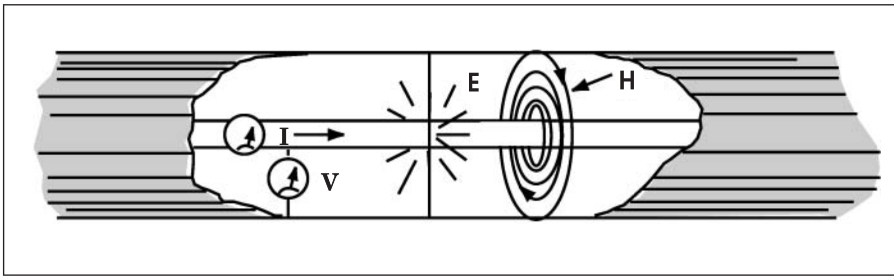

Figure 1 · A signal in a coaxial transmission line is characterized by current in the center conductor, which creates the propagating magnetic field (H); and the voltage potential between the center and outer conductors, or the electric field (E). The outer conductor is the "ground" reference for voltage and carries the return current. Due to skin effect, at high frequencies all current flows on the inner surface (plus skin depth) of the outer conductor.

Design Issues

When engineers talk about higher frequency signal lines, they often characterize their needs in terms of "S" parameters. Ever wonder what the "S" in S parameters stands for? It means "scattering." The signal line performance at a transition can be visualized by thinking of the transition as being in a box. Then, you can picture the effects of a less-than-perfect transition as scattered signals—bounced around by the effects of attenuation, return loss, ingress (EMI from outside) and crosstalk from an adjacent signal line.

Attenuation is another name for signal loss, usually over distance, but also due to components or materials that absorb some of the signal energy. Obviously, without adequate signal strength, the ability to distinguish the signal from noise may be compromised. Losses may be due to radiation of a portion of the signal or dissipation as heat. As the frequency increases, the electromagnetic field that surrounds the signal movement becomes more important, as does the dielectric material that this field moves through. Thus, selecting materials with a low dielectric constant—and resulting high propagation velocity—becomes desirable.

Return loss is a measure of impedance mismatch, typically characterized at the interconnection points. A perfect match transfers all the signal energy into its intended destination (the "load" or "termination"). Return loss represents the magnitude of the portion of a signal that is reflected from a mismatched termination. If large enough, the portion reflected back on itself may be a source of data errors as ones and zeros in the digital data stream are confused.

Ingress is the intrusion of outside interference. If designed without adequate shielding, any transmission line runs the risk of becoming an antenna and picking up spurious signals and noise.

Crosstalk is similar to ingress, except that the intruding signal is not external, but is electromagnetically coupled from an adjacent signal line (an inducted signal).

When managing a high frequency signal with copper wireline technology, the best signal line is symmetrical with regard to spacing from the active trace to the ground, with a known low loss material in between. These are the characteristics of coax (Figure 1). In a perfect world, the material in coaxial transmission lines would be air, but it is difficult to maintain accurate spacing without a solid material. The use of structural materials such as polyethylene (PE), polytetrafluoroethylene (PTFE), and other dielectric materials provides a supporting structure with manageable signal performance degradation. The material selected depends on the frequency involved, but for most applications the best solid material is PTFE.

Most point-to-point interconnection of electronic elements is accomplished on printed circuit boards (PCBs). The PCB supports the components and provides the transmission lines between them, using etched traces of copper on the PCB in lieu of discrete wires or cables.

With the increasing use of high frequency board-level packaging for wireless telecommunications applications, it is important to remember that "wireless" phenomena (electro-

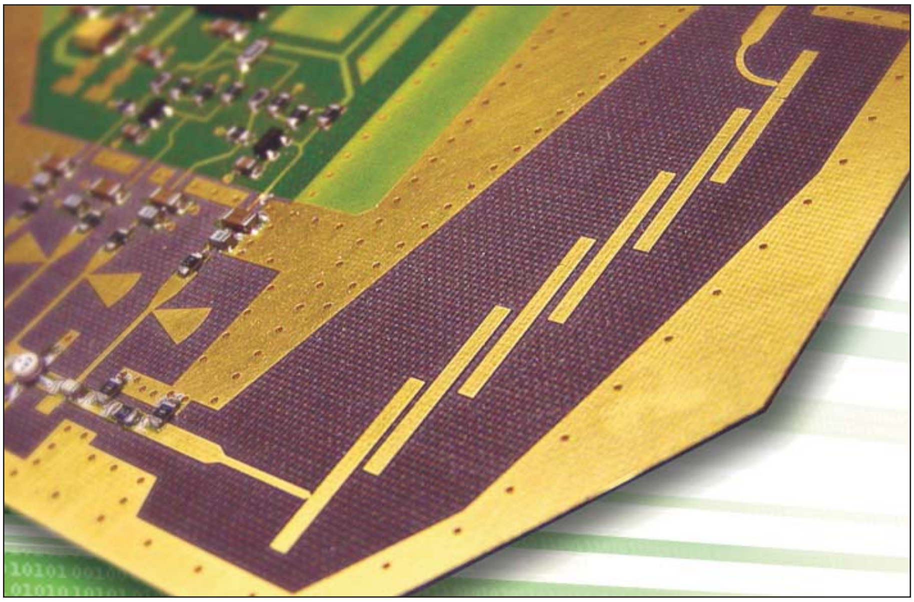

Figure 2 · Coupled resonant transmission line segments can be used as a bandpass filter, an example of the intentional use of electromagnetic behavior on printed circuit boards.

magnetic effects) occur on the PCB itself. For example, two closely spaced adjacent traces might be designed to produce electromagnetic fields to function as a coupler. In another case, a series of cascading patches or transmission line segments will function as a bandpass filter component at microwave frequencies, as shown in Figure 2. These are desired behaviors; keeping unwanted coupling and radiation under control are key elements in PCB design.

Digital designers are familiar with “twisted pair” as a very cost effective solution to low end high frequency RF transmission line problems. The twist maintains balanced currents by alternating each wire’s exposure to the surrounding environment. Without the twist, crosstalk problems between adjacent pairs would be huge, since they are so closely spaced. Unfortunately, twisted pair lines are pretty hard to create on a PCB.

Wavelength-Size Dimensions

High frequency electronics is one situation where size matters. At 100 MHz (the clock rate on a typical Pentium® chip in 1996) the wavelength is several meters long. Even the harmonics, those odd multiples of the fundamental wave that allow it to become “square” and exhibit a far faster rise time for that crisp on/off condition, are relatively low frequency. Low enough, in fact, that the full sinusoidal wavelength is usually larger than the PCB carrying the wave. However, at 1 GHz the wavelength is only 10 inches, and for the 5th harmonic of the waveform it is only 2 inches.

At the board level, digital signal integrity is currently being maintained with the use of exotic laminate materials, a change from the low cost workhorse FR4 that has been the standard choice for the last four decades. High performance laminate suppliers have introduced improved products that address many of the inherent difficulties—coefficient of thermal expansion issues in the z-direction, dimensional stability, and cost—that dogged previous product offerings.

It is useful to remember that, until recently, the microwave market was dominated by the military industry’s emphasis on electrical/electromagnetic performance over low cost assembly technology or low cost materials. Currently, the need for multilayer high frequency materials and high reliability plated thru hole (pth) applications is being met with new, highly engineered polytetrafluoroethylene-based-but-heavily-filled laminate products. These materials solve the problem of z-axis coefficient of thermal expansion (CTE) mismatch between the laminate and the copper plated into the pth (also called a via hole).

A recent development in this area is better affordability of the new laminates. The older class of high performance PTFE-based exotic microwave laminates were in the range of $100 per square foot. New laminate products, including some that are not PTFE-based, have tackled the cost agenda and several good candidates are emerging in the $8/sq. ft. range. By comparison, FR4 is approximately $2/sq. ft.

PCB Transmission Lines

At higher frequencies, new complex board-level engineering issues arise. For example, the tidy 90 degree turns that digital designers use to conserve routing space on the board are destructive to high frequency performance, causing VSWR problems due to reflection from the sharp bend at the corners. One high frequency effect of this conventional PCB layout practice is return loss reflection that cancels the incoming signal, due to reflections that are 180 degrees out of phase. This can be solved by using mitered corners or by using a radius bend to change of direction of the PCB trace.

In addition, the length of traces relative to each other becomes important as designers try to match phase on the board. Etch precision becomes critical for lateral perfection, and in-trace-pinholes are not allowed. Even

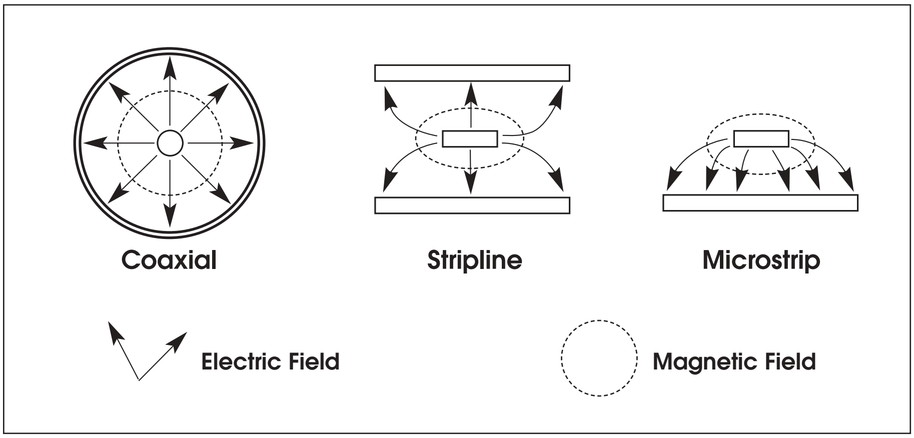

Figure 3 · Types of transmission lines. Stripline is the best planar approximation of coax, but must be implemented between layers of a multilayer PCB. Microstrip is the most common PCB transmission line.

the cross-sectional geometry (trapezoid or hourglass) of the trace becomes important as conductor skin effect becomes significant with rising frequency.

At high frequencies, feature size and the precision of that feature matter. Since the three key variables for a given transmission line are conductor-to-ground spacing, dielectric constant, and conductor characteristics, the simultaneous precision of these three things is paramount to achieving the demanding “design budget” for their variations. On PCB etched flat conductors, for example, it is not uncommon to have fairly “fat” trace widths of 20 mils but with ±0.0006 tolerance overall as measured at the trace flare. Tolerances for dielectric constant of the laminate itself is usually held to 2 decimal places!

As noted earlier, coax is the ideal transmission structure. On a PCB, coax is best replicated by stripline design. Stripline, however, has major drawbacks in the level of difficulty with fabrication of the PCB itself and the corresponding cost. Over the years, designers have learned to make use of a related approach known as copper-backed co-planar microstrip PCB design, also referred to as microstrip. This type of transmission line is used extensively and has been thoroughly characterized by the microwave industry. Figure 3 compares the behavior of these transmission line types.

Microstrip rides on the surface of the board and can be thought of as approximately 70 percent of a stripline. While the open top exposes microstrip electromagnetic fields to ingress (noise and interference) and egress (radiation), the signal impact is largely predictable and can be managed.

One factor to be managed is consistent electrical characteristics. For example, the use of soldermask for coverage of surface traces is a workhorse PCB fabrication technique in surface mount technology (SMT) assembly. In an RF design, the broad use of soldermask degrades the performance of microstrip transmission lines, since the soldermask itself has an uncertain conductivity and uncertain thickness. How is this solved? One answer is solder dams that minimize the use of the mask yet still provide the assembly function of preventing solder from wicking down during reflow. These dams need only have a 20 mil footprint to do their job, and the microstrip function is nearly untouched by their presence in typical PCB sizes and configurations.

The Interconnect Transition

At microwave frequencies, transitions are very important to minimize return loss. One of the most challenging transitions is getting the high frequency signal on and off of the PCB itself. Connectors are a necessary evil and almost always a source of signal degradation as the frequency increases. Separable connectors are greater culprits than solder joints or similar permanent connections.

At this point, our discussion of signal limits gets specific. It may be that there are no known signal limits in frequency other than cost. But cost is a big one. As frequency rises, the wavelength shrinks and the physical geometries needed to support the wave shrink proportionately. Remember that on our PCB, etched traces become narrow, vias become even smaller, dielectrics become tight-toleranced and lumped element features become so tight that fabrication cost is impacted.

Connectors are challenged in much the same way. In BNC technology, for example, any connector will get a signal on and off the board, but the effect on signal integrity may vary greatly, depending on the design of the connector.

One manufacturer might sell a million pieces of a low cost ($1.00/ea.) PCB-mounted right angle 75 ohm jack connector annually. Another company offers what appears to be the same part at eight times the price, and selling several hundred thousand annually. The difference in the connectors is largely in performance excellence at high frequencies. The maker of the high performance connector is heavily involved in managing the transition from microstrip to coax with a design that has a significantly higher fabrication cost.

Fortunately for the industry, connector manufacturers are actively working to broaden design solution options by continuously introducing new products, such as a more elegant side launch version of our example

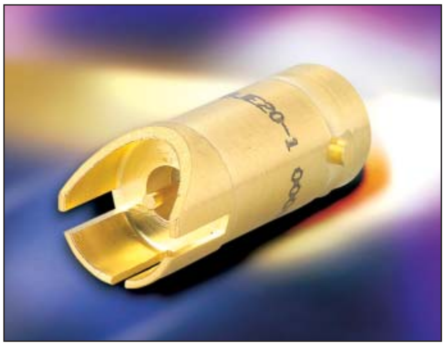

Figure 4 · Photo of a low cost edge-mounted BNC connector designed for reliable high frequency performance.

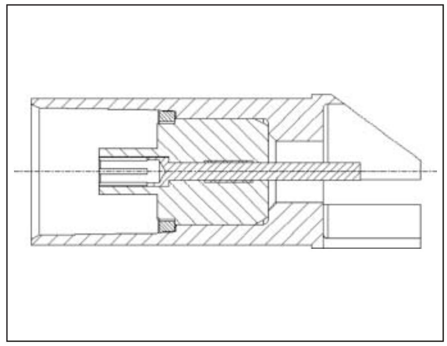

Figure 5 · Cutaway drawing of the connector of Figure 4, showing the various elements of its internal construction.

BNC jack that reaches a price point even lower than one dollar. Figures 4 and 5 describe such a connector from Trompeter Electronics.

In the Future

Systems that operate at even higher frequencies are a certainty for the future of electronics. One high-volume application that is under development is automotive collision avoidance technology. The designers of these systems have opted for a frequency of 77 GHz, partly due to the known interactions of moisture in air with microwave transmissions like radar. This is a much higher frequency than any present low cost system, but performance drives this application and cost reduction will follow.

While 77 GHz technology is rather expensive today, we can look ahead to when collision avoidance technology will become standard equipment on automobiles. Ten years ago, did anyone think we would have GPS auto-based mapping technology as a standard option on new cars? Or did anyone think that we would have 2+ GHz microprocessors on PCBs with 300+ MHz bus speeds?

This article has presented the design challenges that must be addressed when using wired transmission lines for digital signals, with extra attention where they transition from one platform to another, i.e., PCB to coaxial cable. While the issues regarding high frequency signal integrity management are difficult, the talent of our collective engineering personnel and the things they think of continues to amaze.

Author Information

Dale Reed is Corporate Vice President for Sales and Marketing for Sterling Holding Corporation, and is responsible for sales and marketing at Trompeter Electronics and Semflex. He holds a BA degree from Trinity College, an MBA from Georgia State University and a refresher mini-MBA from Stanford University. He has been active in high frequency issues, mainly in printed circuit board design, for more than twelve years and has published several technical papers and magazine articles on design, fabrication and quality topics. He can be reached at Trompeter Electronics, 31186 La Baya Drive, Westlake Village, CA 91362, or by e-mail at dale.reed@trompeter.com