RF and Microwave Basics Impact PCB Design

It is a given that printed circuit board designs are utilizing higher frequencies to meet performance demands. As data rates increase, the resulting bandwidth requirements are driving the upper limit of signal frequency to 1 GHz and beyond. And while this is a far shot from millimeter wave technology (30 GHz), it is indeed RF and low-end microwave.

RF requires a design engineering approach that addresses the associated stronger electromagnetic field effects which naturally occur at these higher frequencies. These fields can induce signals in adjacent signal lines, or PCB traces, creating undesirable crosstalk (interference and overall noise), undermining system performance. Return loss (signal reflected back into the incident oncoming signal) is primarily caused by impedance mismatch and has much the same impact of added noise and interference to the primary signal.

There are two effects of high return loss, both of which are bad news. First, signal reflection back towards the source adds noise to the system, making it more difficult for the receiver to distinguish noise from the signal. Second, any reflected signal is fundamentally a degradation of the signal itself since the “meaning” or shape of the inbound signal can be altered. While a digital system can be far more forgiving since it is only attempting to recognize a one or a zero (on or off), the use of harmonics for faster pulse rise times involves weaker signals at higher frequency. And, while we can implement forward error correction technology to fix some of these effects, the result is system degradation as capacity gets consumed in redundant transmissions. A much better answer is to understand and engineer the RF effects to help, not hurt, your signal management assignment. Overall recommended target values for return loss are minus 25 dB at the highest frequency of interest (usually the worse-case data point), which converts to about 1.1 VSWR.

Traditional PCB design has been driven by "smaller, faster and cheaper". At RF frequencies on a PCB, "faster" does not always allow for "smaller" due to some realities of RF signal management design:

1. The primary way to manage unwanted crosstalk is by ground plane management, trace-to-trace spacing, and/or reduction of stub inductance.
2. The primary way to reduce return loss is to match impedance. This involves effectively managing the dielectric materials and spacing between the active trace and the ground, particularly in transitions.

Since interconnect points are the weakest link in the electronic chain, each should be challenged and solved as their electromagnetic properties become the dominant engineering issue with the use of RF frequencies. The three major categories of interconnect in a board system are chip-to-board, within the PCB, and getting the signal on and off the PCB from an external device.

Chip-to-PCB

Within the chip itself performance is secure and processing speeds are already well into the 1 GHz range. Pentium IV, Itranium, and even faster chips with huge input/output interconnection counts are already being introduced or designed. At the recent Wireless Workshop in Sedona, AZ (now called GHz Interconnect Workshop – go to www.az.ww.com) one of the most stimulating topics being discussed was various known and proposed ways of dealing with rising I/O count and frequency. The basic problem is that interconnect density has become so high that the fundamental particle size of the materials is becoming the limit. An innovative answer put forward was use of a very local wireless transmitter built into the chip for the purpose of moving data to adjacent board devices.

Regardless of where this takes us, it was clear to that audience that IC design is far ahead of PCB platform design with respect to the use of high frequencies.

Within the PCB

Techniques and guidelines for high frequency PCB design do exist:

- To reduce return loss, miter corners on transmission line traces (see Figure One).
- Utilize high performance dielectric board laminates with tightly controlled dielectric constant values. This allows engineered management of the electromagnetic field that is moving through the dielectric adjacent to the trace itself.
- Complete PCB design specifications regarding high precision etching (usually helped by specifying one-half ounce copper, tolerancing the trace width to +/-0.0007 overall, managing the undercut and cross sectional view of the trace geometry, and specifying the plating condition of the side-walls of the trace itself). These steps result in overall management of the geometry and plated surface of the trace (conductor), important due to skin effect, a phenomenon associated with microwave frequency. See Figure Two.
- Avoid using leaded components due to stub inductance of the protruding lead. At these frequencies, surface mount components is strongly preferred.
- On signal vias, avoid pth technology in sensitive board areas due to the unwanted stub inductance of the hole. (Imagine a pth on a 20-layer board to connect signal layers 1 and 3 , the "stub" is the pth itself radiating onto layers 4-19).
- Provide generously for ground planes. Stitch them together with mode suppression holes to inhibit the 3D electromagnetic fields covering the board.
- Select electroless nickel/immersion gold instead of HASL for plating. This surface offers better surface properties for high frequency currents (see “skin effect” explained in Figure Two). In addition, this highly solderable plating involves less lead and is better for our kids and the planet that they live on.
- Soldermask prevents the unwanted flow of solderpaste. However, applying soldermask all over the surface of the board effectively alters the flow of electromagnetic energy in a microstrip design due to coverage of uncertain thickness and unknown dielectric. Instead, use only solder "dams" as soldermask.

If these issues are unfamiliar to you, tap into the rich knowledge base of a microwave board design engineer experienced in the military segment. You can discuss your price point boundary conditions with them suggesting, for instance, that use of copper-backed coplanar microstrip design is more cost effective than stripline, and that this matters to you. These talented engineers may be unaccustomed to cost limits, but their skill set is complex. Attempting to develop young "green" engineers that are inexperience with RF effects and how to effectively deal with them may prove to be a long-term project. Other solutions are appearing such as improved computer models that offer RF effects built in to the software.

PCB to Outside World

Imagine that we solve all the signal management problems on the board and in the interconnects to the discrete components soldered to them. What about getting the signal on and off the board into a wire (copper or fiber) for connection to a device some distance away? As an innovator in coax technology, our company has been working on this with some important results (see examples in Figure Three).

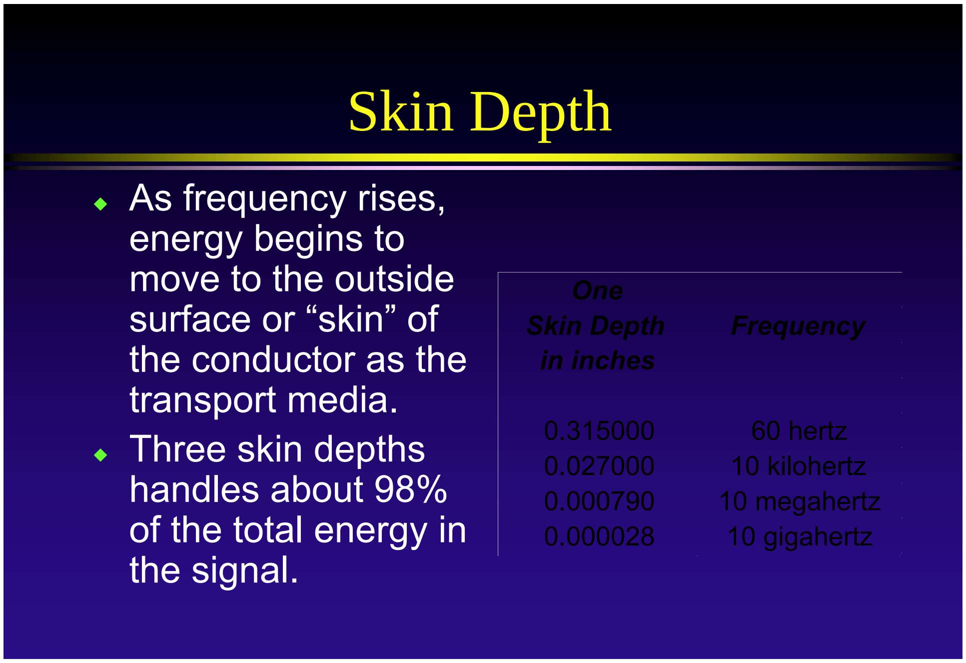
Also, take a look at the electromagnetic fields represented in Figure Four. In this case, we are managing a transition from microstrip to coax. In coax, the ground plane is circular (braid) and evenly spaced. In microstrip this is changed to a ground plane under the active trace. This introduces certain fringe effects that need to be understood, predicted, and considered in the final design. Certainly, this mismatch is a source of return loss and must be minimized to avoid additional noise and signal interference.

Managing impedance in the board, up to the surface level of the board, through a solder joint, into a connector, and back out via coax is not a trivial design problem. Further, impedance is a moving target that can vary with frequency and become harder to manage with rising frequency. Moving signals over larger bandwidths (broadband) using higher frequencies seems to be an established design issue for the immediate future. Even fairly narrow-band applications, such as moving uncompressed CATV data files or voice-over-IP data files, are starting to look like broadband applications with the use of frequency stacking (block conversion).

Conclusion

PCB platform technology needs to play "catch up" ball to get to where the integrated circuit people are now. Continuing rapid advances are needed in the area of high frequency signal management in the PCB and in getting the signal on and off the PCB. Whatever exciting innovation ensues, my prediction is that bandwidth use will continue to be higher than ever, and use of high frequency signals will be the enabling technology to achieve this.

Dale Reed is VP of Marketing at Trompeter Electronics in Westlake Village, CA and can be reached at dale.reed@trompeter.com



Skin Depth

- As frequency rises, energy begins to move to the outside surface or “skin” of the conductor as the transport media.
- Three skin depths handles about 98% of the total energy in the signal.

One Skin Depth in inches	***Frequency***
0.315000	60 hertz
0.027000	10 kilohertz
0.000790	10 megahertz
0.000028	10 gigahertz

Figure Two – Showing the impact of frequency on skin depth of the electrical energy on a conductor.

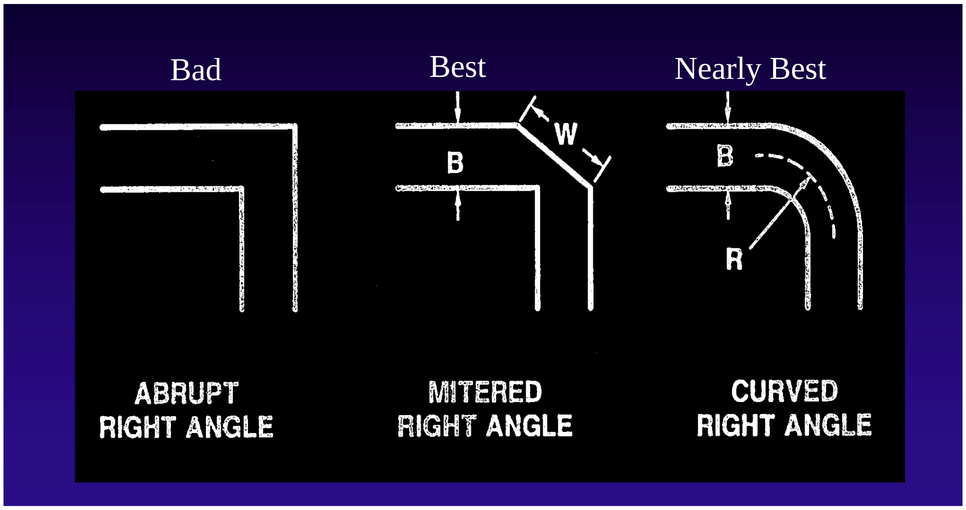


Figure One – Showing various treatments of copper layout patterns to achieve a 90 degree transition of a trace on a PCB.

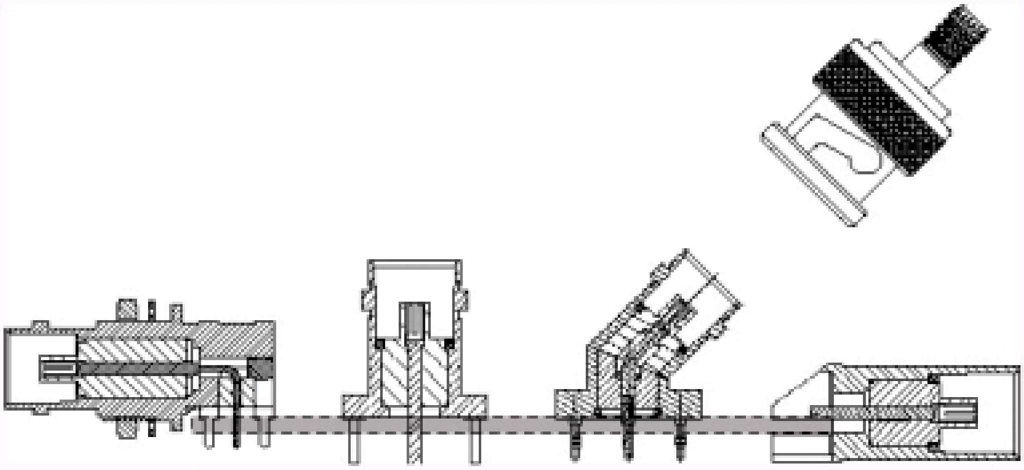


Figure Three – Illustration of several methods of launching a high frequency signal from or to a PCB using coax transmission line technology as the cabling element.

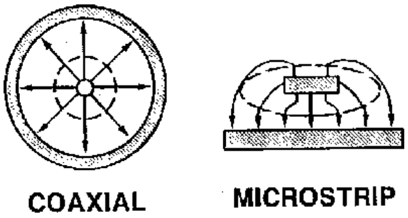


Figure Four – Illustration of the electromagnetic fields that radiate from active transmission lines, the center conductor of coax and the surface trace of a microstrip board design.