RF/Microwave Connectors on Printed Circuit Boards

By Gary Breed
Editorial Director

This month's tutorial takes a look at the practical matter of characterizing, then installing, an RF connector on a printed circuit board

When mounting an RF/microwave or high speed digital connector to a printed circuit board, the transition from the connector body and inner conductor pin to the printed traces is often the source of excessive mismatch. The discontinuity in size, shape and surrounding conductors results in an area with a characteristic impedance that can be much different (usually lower) that the system impedance of 50 ohms (RF/microwave) or 75 ohms (data or video).

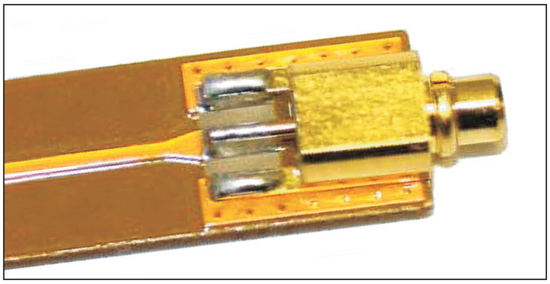

Figure 1 · Connectors must provide a transition from a round coaxial transmission line to the planar microstrip or stripline structure of a p.c. board.

An additional challenge is the transition from the round coaxial structure of cables and their connectors, to the planar stripline or microstrip structure of signal paths on a p.c. board. The connector shown in Figure 1 demonstrates one approach to solving the problem. The connector body is designed to minimize the VSWR "bump" caused by the change from coaxial to planar transmission lines, but the pin that is soldered to the board has as added vertical thickness, and almost always will require a solder pad that is wider than a stripline with the desired characteristic impedance.

Reference [1] offers a good description of the problem and typical solutions. Although the author covers the issue from the perspective of 75 ohm BNC connectors, the information applies to 50 ohm systems and other connectors, as well. As shown in Figures 2 and 3 on the next page, one solution is to modify the ground metal layer under, and adjacent to, the connector. Wider spacing between the signal conductor and the ground/shield conductors increases the characteristic impedance in the area where the required solder pad is much wider than a normal 75-ohm microstrip line. Figure 2 illustrates the problem by showing the relative widths of the solder pad and microstrip lines on the top metal layer. Figure 3 shows how the top metal and the metal of the next lower layer are modified to create a region with higher characteristic impedance.

An alternate solution is to mechanically machine the p.c board to create an air space adjacent to the solder pad. This lowers the effective dielectric constant of this part of the board, which will increase the characteristic impedance without changing the pad and trace widths.

Top-mounted connectors present a different set of structural variations that affect the impedance match between the connector and the p.c. board traces. This type of mounting offers greater mechanical strength than edge-mounting, and is the only option for installing connectors in locations other than at the edge of a board.

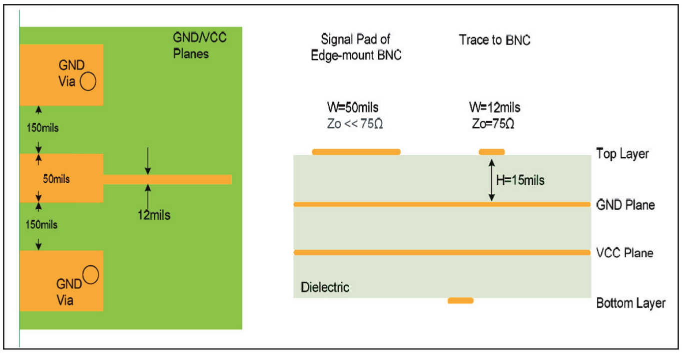

Figure 2 · Cross-section of a p.c. board "landing pad" area for an edge-mounted BNC connector, with no compensation for impedance mispatch (adapted from Ref. (1).

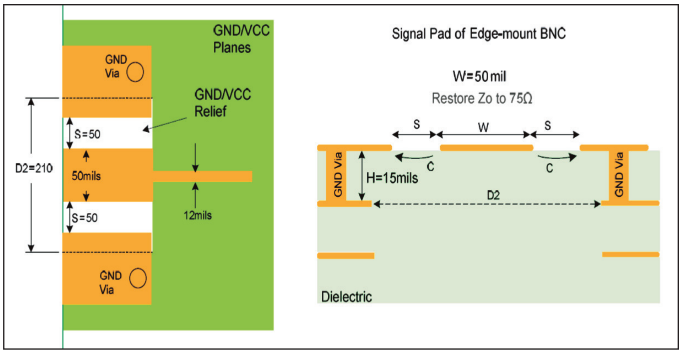

Figure 3 · Cross-section of a p.c. board "landing pad" area for an edge-mounted BNC connector, with the ground metallization modified to provide an improved match to a 75 ohm (adapted from Ref. (1).

Surface-mount connectors will perform similarly to edge-mounted connectors, with round-to-planar transitions and impedance variations due to solder pad size. However, through-hole mounted connectors must have vias to provide both signal and ground connections. As shown in Figure 4, the length of the via has an inductance, and the gap between the via and intervening metal layers has a capacitance. The average values will determine the characteristic impedance of the via "tube," but the layered structure means that the impedance will vary along the length of the via.

If the space between the layers is small relative to the wavelength of the highest frequency desired signals, the variation will have little effect on performance. But as frequency increases, the effects will become increasingly apparent as increased VSWR and its attendant loss, and time-domain reflections that can affect the modulated waveform of RF signals, or the waveform shape (and eye closure) of high speed digital signals.

At 2.45 GHz, with typical FR-4 material dielectric constant of approximately 4.4, the thickness of a p.c. board is in the range of 1/17 wavelength. At this frequency, problems would not be significant. However, a common rule of thumb is that wavelength-related problems will arise when dimensions are in the range of λ/10. As the operating frequency rises above 2.45 GHz, designers should be prepared to implement compensating techniques to avoid performance issues with through-hole connectors.

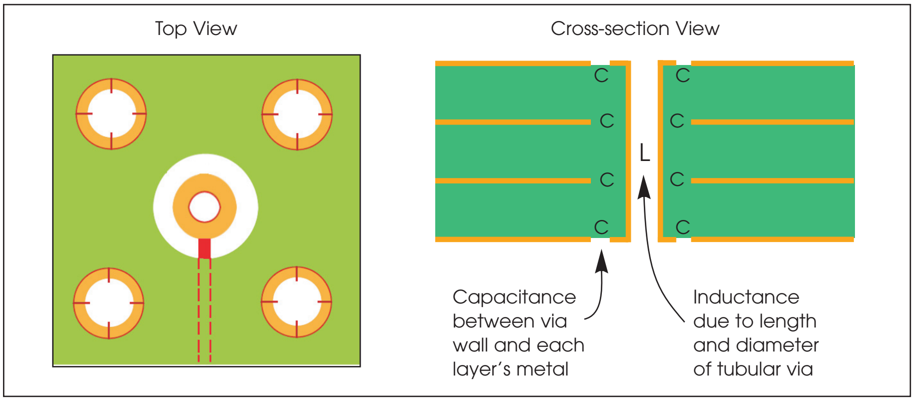

Figure 4 · Thru-hole, top-mounted connectors have an impedance that is affected by vias, mid layers and metal layers of the p.c. board.

Of course, the magnitude of the problems will vary with the connector type and mounting method as well. Well designed and precision manufactured surface-mount connectors are available with performance specified into the tens of GHz.

Summary

At high frequencies and fast edge rates, the interface between an RF/microwave or high speed digital connector and the printed circuit board can be the most critical location in the signal path. Designers must be aware of the potential problems that can arise, and be prepared to use appropriate techniques to prevent their occurrence.

References

1. T-K Chin, National Semiconductor Corp., "Optimizing BNC PCB Footprint Designs for Digital Video Equipment," available at: www.samtec.com/technicallibrary/white_papers.aspx

2. "PCB Design Guide," Trompeter Electronics, (available from several sources—do an Internet search by title + Trompeter)

3. S. McMorrow, J. Bell, J. Ferry, "A Solution for the Design, Simulation and Validation of Board-to-Board Interconnects," *High Frequency Electronics*, Jan. 2005.