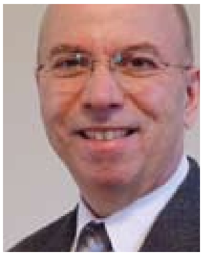

BY LAMBERT SIMONOVICH

Stub termination

Figure 1 illustrates the channel insertion loss of a 30-in. differential channel with differential vias at each end. Using short vias with no stubs (green, top trace), the response rolls off smoothly to −13 dB at 5 GHz. Thick backplane vias with long stubs (red) create unwanted resonances in the channel insertion loss, whereas vias with short or no stubs do not. If these resonances occur near the Nyquist frequency of the bit rate, they will devastate the eye opening at the receiver.

Stub resonances occur when a portion of the signal traversing the active region of a via diverts down into the stub section, reflects off the open-circuited end, and returns later to recombine with the main signal. At a high frequency—the quarter-wave resonant frequency—the round-trip delay from the active region of the via to the end of the stub and back equals a half-cycle. If this scenario occurs, the main wave and the reflected wave appear 180° out of phase, producing destructive signal cancellation. The longer the stub, the lower the resonant frequency is.

You can shorten your backplane's via stubs by back-drilling the vias as close as possible to the active internal-signal layer. This complex and costly process requires that you specify the necessary back-drilling depth for each via using special design features in your artwork. Occasionally, glitches in the back-drilling process leave some vias with longer-than-expected stubs. It is only after the board has been fully assembled and tested in the system that such a problem can show up in the form of a higher bit-error rate.

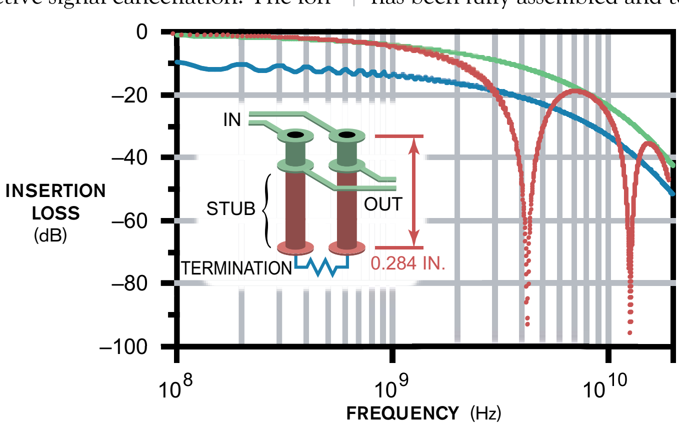

Figure 1 In this simulation, using Agilent's ADS, the channel insertion loss represents two differential vias with 30 in. of PCB etch. This loss deteriorates significantly when you use long via stubs (red), as opposed to short, back-drilled vias (green).

Some vias can never be adequately back-drilled. For example, you must maintain a minimum via barrel length to ensure mechanical stability and good electrical contact to a press-fit connector pin. If that pin connects to a signal layer shallower than the minimum required press-fit via length, the protruding portion of the via creates a longer-than-optimum stub.

Because using via stubs seems unavoidable, what is the alternative? Nicholas Biunno, PhD, principal scientist at Sanmina-SCI Corp, suggests terminating them. Sanmina has developed the new MTS (matched-terminated-stub)-via technology, which embeds tiny metal thin-film or polymer thick-film resistors within a PCB (printed-circuit-board) stackup during fabrication. The technology can terminate a differential-via stub, thus preventing reflections. With one resistive layer at the bottom of your PCB stackup, you can terminate all the high-speed via stubs in your design. Experiment with your favorite field solver to find the right value of resistance. As **Figure 1** shows (blue trace), you can eliminate the resonant notches at the cost of additional flat-loss attenuation.

This stub-termination technology looks like a promising alternative to back-drilling, resolving many of its limitations. Combining it with silicon that can accommodate the additional signal loading may extend the life of traditional copper interconnections for the next generation of Ethernet standards beyond 10 Gbps.EDN

Lambert Simonovich, previously a signal-integrity and backplane architect at Nortel, provides innovative signal-integrity and backplane solutions at Lamsim Enterprises Inc. You can visit his Web site at http://lamsimenterprises.com. Howard Johnson will return next month.