BY DOUG SMITH

Crossing the river

Cross a river without a bridge, and your clothes get soaked. Cross a split-plane gap with a high-speed signal, and your whole development schedule gets soaked.

As an EMI (electromagnetic-interference) specialist, I have known for years that a high-speed signal crossing a split or gap in its solid reference plane generates EMI. The first thing I do when troubleshooting a PCB (printed-circuit-board) design is shine a light through the board to look for splits. That trick immediately spots any traces that cross the splits. Of course, I can see only those splits that completely penetrate all the planes in the circuit, but it's surprising how many times this simple check pays off.

A split in the planes causes an impedance discontinuity in the signal path crossing the split. The discontinuity reflects energy back toward the source. It turns out that the split reflects only higher-frequency components of the incoming signal. Removing those components degrades the rise time of the remaining signal that crosses the split.

To demonstrate rise-time degradation, I fabricated a small PCB with two straight traces, each about 15 cm long (**Figure 1**). The board has a signal-ground-power-signal stackup, typical for a four-layer design. Trace A, a 50Ω microstrip configuration, routes on Layer 1 over solid ground metal its whole length in. Trace B uses the same configuration but crosses a break in the planes. The break is a narrow, 5-cm-long gash. The break penetrates both solid reference planes. The auxiliary output of my Agilent Infiniium scope drives the traces one at a time. **Figure 1** superimposes the two results, showing significantly different rise times (300 versus 400 psec). The faster signal is the one over a solid ground plane. If your plane splits exceed 5 cm, expect even more pronounced effects than these ones.

Two explanations for the rise-time degradation come to mind. One is that the split in the planes forms an impedance discontinuity that reflects energy, representing the difference between the two results. Another explanation, possibly oversimplified for this case because of the circuit dimensions, involves lumped-element equivalent circuits. The lumped-element view supposes that signal current flows on Trace B straight across the gap and that an equal but opposite returning signal current flows in the adjacent ground plane. At the break, the returning signal current must pass around the ends of the break to either side of the signal trace. It cannot leap across the gap. The diversion of returning signal current away from the main signal current creates inductance, which filters out higher-frequency components of the incoming signal.

Whichever way you think of it, any significant slowing of your signal rise or fall time can lead to a number of undesirable effects.

Next time you analyze a PCB layout, keep in mind that crossing splits in the planes can cause problems with radiated emissions, immunity to external signals, crosstalk, jitter, and degraded rise and fall times. Don't get soaked by this one! EDN

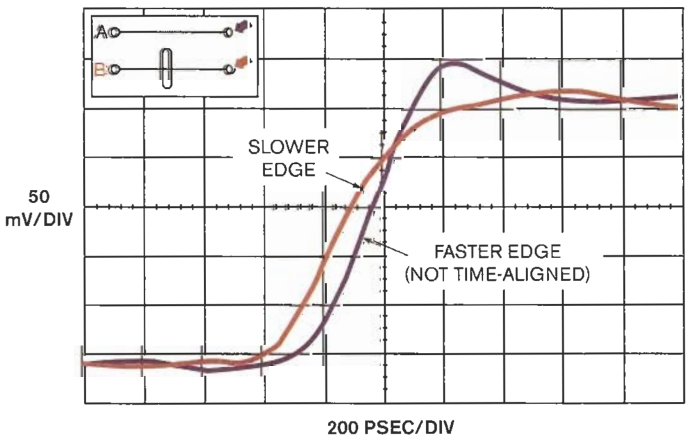

Figure 1 A signal crossing a plane split (red) has a slower rise time than the one with a solid, continuous reference plane (blue) (adapted with permission from http://emcesd.com/tt2005/tt010105.htm).

Doug Smith is an independent consultant specializing in EMI/ESD (electrostatic-discharge) design. Visit his Web site at www.dsmith.org. Howard Johnson will return after a summer break.

+ www.edn.com/signalintegrity