If you have as many as 16 differential line pairs, and you must go through a connector to terminate the differential signals on a daughter card, what is the best signal-to-ground ratio and pattern to use? In this case, the connector is a high-density pin connector. If the differential impedance is 100 Ω, do I need a special ground pattern, as the signals go through the connector, to maintain the differential impedance close to 100 Ω?

Sal Aguinaga

Thanks for your interest in high-speed digital design. There is no general formula for the number of grounds required; it depends on the spacing and sizes of the connector pins and how they are bent. Here are a few general rules you may want to consider:

Put the two elements of each differential pair on the same row of the connector. This placement ensures that the elements have the same pin lengths and go through the same pattern of elbow bends. On a synchronous bus, if you have enough time to wait for the crosstalk to settle, you may not need to isolate the differential pairs from each other. If isolation between pairs is required (for a good, low-crosstalk signal or a clock or asynchronous interrupt signal), place the pairs so they are never adjacent to any other pair. (This arrangement implies that you are using at least as many grounds as signal pins to separate the pairs and probably more.)

The differential impedance of most open-pin-field connectors is probably a little higher than you want. You can measure this impedance using a pair of test boards on which you can mate the connector halves. The boards don’t use any traces. They can be solid copper with holes drilled for the connector pins. Ground all the pins that you need grounded for your application. Use two RG-174 50 Ω coaxial cables to route a differential, 100 Ω signal into the designated signal-pin pair. Let this signal go through the connector to the far side. On the far side, terminate the signal differentially with 100 Ω. For any signal speed that works with an open-pin-field connector, a 1/8W axial 100 Ω resistor works as a terminator.

Blast in a differential signal from your 100 Ω source. Make a record of the resulting waveform as you measure it at the source. This record should show the source waveform going out and a first reflection coming back (you’ve made a crude TDR measurement). Use a step rise time commensurate with the rise time you will use in the real system. Don’t mess around with fancy 35-psec step edges. They just show you a bunch of fine-structure detail that doesn’t matter in the real system. Now disconnect the coaxial cables from the connector. Place the 100 Ω termination directly across the coaxial cable outputs, with the coaxial grounds tied together. Repeat the measurement.

You should (ideally) see no reflection. Looking at the difference between the first measurement and the second, if the reflected waveform bumps up in the positive direction (same polarity as the step input), the connector impedance is a little too high. If it bumps negative, the connector impedance is too low. Adding more ground pins around the signal pair lowers the impedance. Spacing the signal pins further away from ground raises the impedance.

Here’s a bonus idea: Adding a little lumped-element capacitance from signal to ground on each side will lower the effective impedance. You can implement this idea in the pc-board layout by using larger-than-normal via pads. Getting this idea to work requires experimentation and remeasurement. The “big-pad” concept works when the connector through-delay is less than 1/6 of the signal rise time and the connector acts like a lumped-element inductor (it has too high an impedance).

THE NUMBER OF GROUNDS DEPENDS ON SPACING AND SIZES OF THE CONNECTOR PINS AND HOW THEY ARE BENT.