Uncommon Sense

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

When common sense fails, tap into your uncommon sense. While common sense is considered conventional wisdom, uncommon sense is a re-examination of that conventional wisdom. Basically, common sense teaches us that the way it has always been done is the right way, and that's just how things are. Following common sense is usually the safe way to go. But the people who are really making a difference in the world are usually the people who try something new. Tapping into our uncommon sense allows us to take a deeper look at things we often take for granted.

It is remarkable that with all of today's high-performance systems, in which very complicated electromagnetic effects play a dominant role, many of us still hold misconceptions about the fundamental nature of how signals interact with interconnects. In this month's column, I will look at the contemporary ways of addressing an old issue (déjà view, as I call it) and go beyond the design of PCBs.

There is always a debate regarding how a differential pair should be routed. Conventional wisdom tells us that since the two halves of the pair carry equal and opposite signals, a good ground connection is not required as the return current flows in the opposite signal. And tight coupling between the signals is better than loose coupling as it reduces undesirable coupling/crosstalk from aggressor signals. And let's face it, it is easier to route a pair together so that we logical manage the planes, aggressor signals and matched delay simultaneously particularly in complex designs.

Some argue that beyond the fact that differential pairs transfer equal and opposite signals, there are no special requirements that need to be considered when using differential pairs. They should be treated as two single ended signals. The signals of a differential pair don't need to be routed together, should not be tightly coupled and are not required to be routed to the differential impedance.

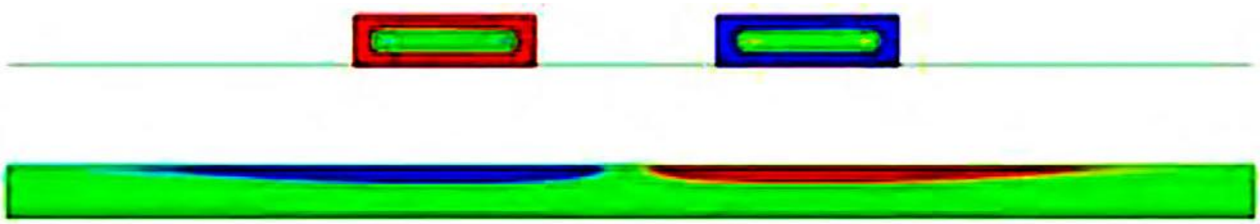

Figure 1: Return current paths of a differential pair (courtesy Ansoft).

Basically, a differential pair is two complementary transmission lines that transfer equal and opposite signals down their length. We assume that tightly coupled differential pairs have no current in the adjacent planes because the return current of one line is carried by the other. That is not correct. On a PCB, the return current path, of each trace of the pair, flows directly below each trace in the reference plane as seen in Figure 1.

If the differential pair is well balanced, then tight coupling will achieve an effective degree of field cancellation. However, if they are not perfectly balanced, then the degree of cancellation is not determined by the spacing, but rather by the common-mode balance of the differential pair. Most digital drivers have poor common-mode balance and therefore differential pairs often radiate far more power in the common-mode than in the differential-mode. In such a case, one gains no radiation benefit from coupling the differential traces more closely together.

According to the FCC Class B compliancy standard, the differential-mode radiation from a microstrip pair, with 20mil separation, should theoretically yield a 40dB radiation improvement at 1 GHz over the radiation one would measure from the same signal routed as a single ended trace. It is the common-mode signal that dominates the radiation and decreasing the pair spacing will not improve this situation.

For a perfectly balanced differential signal, the radiation from one trace exactly cancels the radiation from the other as they are equal and opposite. However, a common-mode signal represents an average of the two signals in a pair. The radiation is identical on both traces and therefore it does not cancel but rather reinforces. To minimize radiation and crosstalk you must think explicitly about the common-mode component of the differential signal—skew creates this common-mode signal.

Arguably, the principal source of imbalance is time delay skew between the two traces. The easiest way to minimize this skew is to match the electrical lengths and to correct any shift immediately, after it arises, by adding length (hence delay) to the shorter trace. Unfortunately, time-delay skew can also be introduced by a variation in the dielectric constant of the glass-resin composite. This, weave induced skew, can be minimized by using materials with a spread-glass weave such as the new Isola I-Speed, I-Speed IS and Tachyon-100G that have been specifically developed for high-speed applications.

The transformation from differential to common-mode also takes place on bends and non-symmetrical routing near via and pin obstructions. In a previous column, Beyond Design: Differential Pair Routing, I concluded that symmetry is the key to successfully deploying differential signals in high-speed designs. Maintaining the equal and opposite amplitude and timing relationship is the principle concept when using differential pairs. Mirror symmetry (as in Figure 2) about an axis, along the interconnects, avoids mode transformation. The symmetry property preserves the signal in the differential-mode which does not radiate. Common-mode noise may have little effect on signal integrity, but will have a more serious impact for EMI.

Mode transformation can also be minimized by reducing the size of any bends in the pair. Any skew introduced, by a bend, should be corrected immediately after the bend so that the majority of the length of the pair is balanced. Also, routing in a stripline configuration has

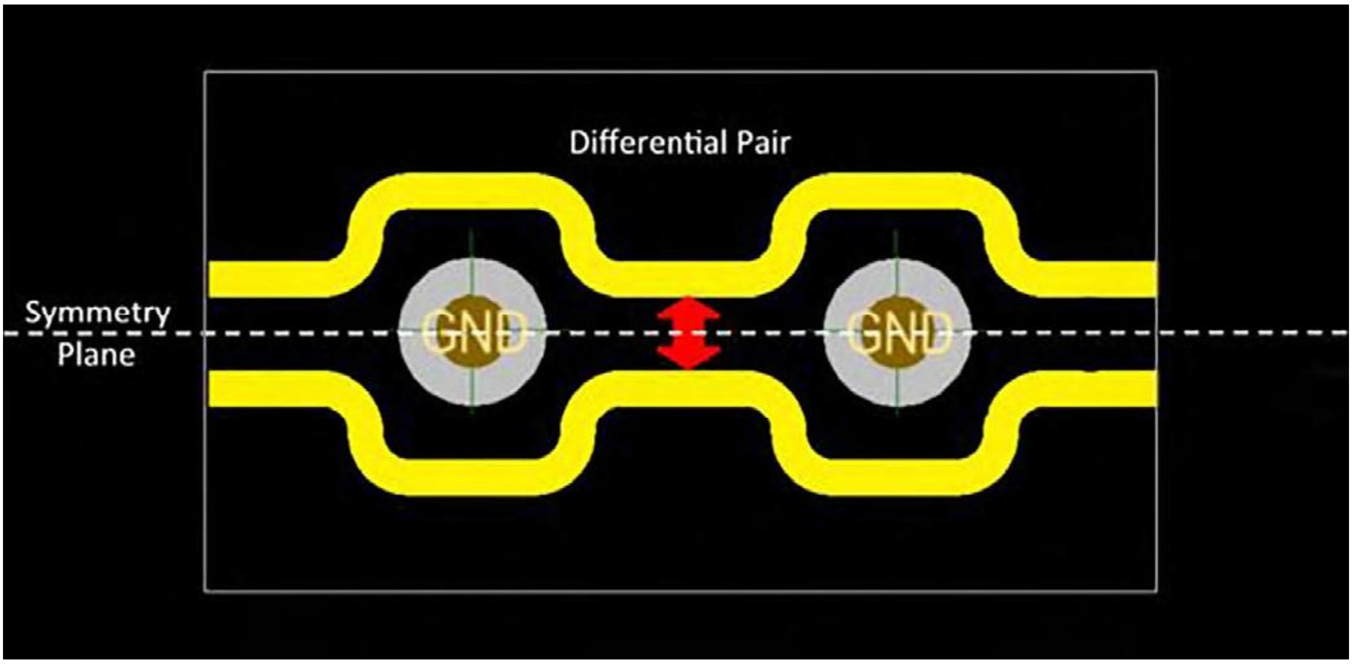

Figure 2: Differential pair symmetry.

equal common and differential-mode propagation velocity which helps reduce far end mode transformation.

It can be seen in Figure 3, that the differential impedance of this particular microstrip pair, levels-off at 100 ohms above 12mils trace clearance (blue curve). This is simulated quickly by multiple passes of the field solver. So, all other factors being equal, the differential impedance will always be 100 ohms regardless of increased spacing. This allows the pair to be split to traverse obstacles (vias or pins) without altering impedance. This curve provides a clear map of the design space and can increase your productivity by efficiently defining the stackup configuration for single ended and differential pairs. In this case, once the separation is greater than 12 mils, the two traces convey single ended signals.

A few signaling standards have both differential and common-mode impedances specified, but many do not. This provides the freedom for the user to set it according to their application. These two impedances are related to the coupling strength of the differential pair. As the traces get closer, both differential and common-mode impedances are reduced.

With loose coupling, there is always the case where the traces must be bought closer together to weave through constrictions such as connector fields, BGA balls, vias and other land patterns. As the traces are bought closer together the impedance drops, so the traces must be made narrower to compensate. The majority of PCB fab shops limit the trace width to 3 mil minimum. And of course, the narrower the trace width; the more the expense.

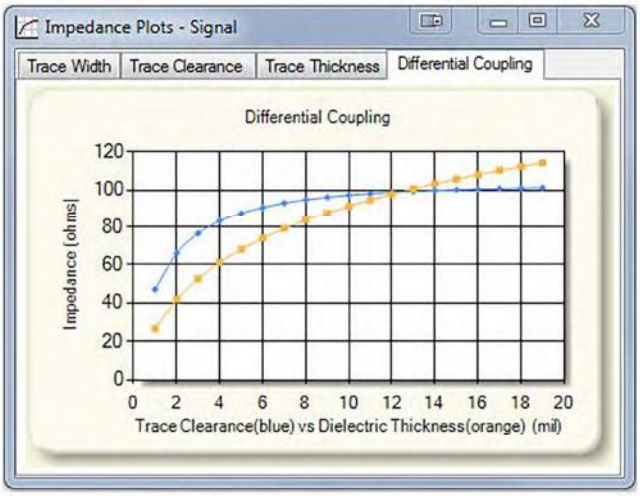

Figure 3: Differential impedance levels-off above 12 mils, simulated by the iCD Stackup Planner.

Also because of the narrower traces, the current is forced into a small width of copper. This increases trace resistance, inductance and skin-effect losses. Contrary to common belief, closely coupled pairs do not improve EMI. This is because it is the common-mode signals from the drivers—the natural imbalance (skew)—that radiates.

In conclusion, to minimize radiation and crosstalk, of a differential pair, one must think explicitly about the common-mode component of the signal. It is not a matter of which is better—tight or loose coupling—but rather which scenario better avoids timing skew that creates this common-mode signal. It is imperative to determine exactly where the differential impedance levels-off. But without a good field solver that simulates signal coupling and flight time, you are really just taking a stab in the dark, which is not good design practice.

Points to Remember

• Conventional wisdom tells us that tight coupling between the signals is better than loose coupling, as it reduces undesirable coupling/crosstalk from aggressor signals.

• On a PCB, the return current path of each trace of the pair flows directly below each trace in the reference plane.

• If a differential pair is not perfectly balanced, then the degree of cancellation is not determined by the spacing, but rather by the common-mode balance.

• Most digital drivers have poor common-mode balance and therefore differential pairs often radiate far more power in the common-mode than in the differential-mode.

• For a perfectly balanced differential signal, the radiation from one trace exactly cancels the radiation from the other as they are equal and opposite.

• The common-mode signal represents an average of the two signals in a pair. The radiation is identical on both traces and therefore it does not cancel but rather reinforces.

• To minimize radiation and crosstalk, you must think explicitly about the common-mode component of the differential signal—skew creates this common-mode signal.

• To minimize the skew, match the electrical lengths and correct any shift immediately, after it arises, by adding length (hence delay) to the shorter trace.

• Time-delay skew can also be introduced by a variation in the dielectric constant of the glass-resin composite.

• The transformation from differential to common-mode also takes place on bends and non-symmetrical routing near via and pin obstructions.

• Mirror symmetry, about an axis along the interconnects, avoids mode transformation. The symmetry property preserves the signal in the differential-mode which does not radiate.

• The differential impedance of a pair, levels-off at a particular coupling point. Beyond this point, the differential impedance will always be the same regardless of increased spacing.

• As the traces get closer, both differential and common-mode impedances are reduced so the traces must be made narrower to compensate.

• Narrow traces increase trace resistance, inductance and skin-effect losses.

• Contrary to common belief, closely coupled pairs do not improve EMI. PCBDESIGN

References

1. Barry Olney, Beyond Design, "Differential Pair Routing."

2. Eric Bogatin, Fundamental Myth-Conceptions.

3. Lee Ritchey, Right the First Time.

4. Yuriy Shlepnev, Analysis of differential line transition from tight to loose coupling, and practical notes on mixed-mode transformations in differential interconnects.

5. Howard Johnson, High Speed Digital Design.

Barry Olney is Managing Director of In-Circuit Design Pty Ltd (iCD), Australia. The company is a PCB design service bureau that specializes in board-level simulation. iCD developed the iCD Stackup Planner and iCD PDN Planner software. Visit www.icd.com.au