Return path discontinuities and EMI: Understand the relationship

Minoru Ishikawa - June 11, 2015

It’s conventional wisdom that a solid, continuous return path provides a better result in electromagnetic compatibility (EMC). This article discusses the relationship between return path discontinuities and EMC.

A quality signal channel has a nice, uniform trace and a continuous return path from driver to receiver. Disruption to the return path introduces noise, and is typically caused by:

- Changing the reference plane(s) along the signal path
- Discontinuities within the reference plane

There are two modes of high-frequency current flow, illustrated in figure 1.
**Figure 1: Normal mode (top) and common mode (bottom) high-frequency current flow.**

*Normal mode:* This is the simpler mode. Current goes along a closed circuit loop, so the total current along the loop becomes zero. The loop is small/narrow enough, so the radiation from the incident current is canceled by the return current. The return path of via microstrip or stripline is just beneath/above plane(s) of the trace signal. Thus, when a trace crosses over a plane gap it breaks this condition.

*Common mode:* Noise power goes through both of the traces and, lacking an appropriate, closely spaced plane, something like the enclosure can become the return path. The noise induced by the currents on the signal traces is not canceled by a nearby return current, so strong radiation could occur. This physically larger circuit can act as antenna, so it may cause EMI as well as an EMS (electromagnetic susceptibility) issue. The common-mode noise source could be the reference plane discontinuity mentioned in Normal mode.

Figure 2 is a simulation result of a microstrip signal trace that crosses a slot on the ground plane. The driver is on the left side of the slot, connecting to both the signal trace and the ground plane with matched impedance. The current goes from the driver to the left end of the trace and returns on the ground plane. Due to the slot, the return current on the plane spreads along the gap, and some current is shown at the edge of the board. As can be seen, the discontinuity affects the normal mode current flow.

![Image of return current around a plane slot](image-url)

**Figure 2: Return current around a plane slot (modeled with Mentor's Nimbic nWave).**

**A differential pair and its currents**

For higher-frequency signal traces, a differential pair is used. Differential pairs carry the signal and the opposite phase of the signal so that they can combat common mode noise or induced noise. All of the SerDes signals, such as PCI Express or Serial ATA, use differential pairs.

Many differential pairs use a differential impedance of 80 to 100 ohms. Since the paired signals are coupled tightly, a differential pair is far more tolerant of reference plane discontinuities.

Let’s look at two types of differential pairs: Figure 3a has narrow spacing between traces and thinner dielectric layer, resulting in 73 ohms impedance for each trace. In Figure 3b, each trace has an impedance of about 50 ohms. Both circuits have 100 ohms differential impedance, but the one in
Figure 3b has stronger coupling between the traces. Figure 3a has stronger coupling to the reference plane. A typical design will be somewhere between these two extreme cases.

![Figure 3](image1.png)

**Figure 3: In (a), the trace impedance is about 73 ohms; in (b) it is 50 ohms.**

Figure 4 shows current on the differential pair and on the plane of the wider trace spaced example. Though the traces are a differential pair, it acts like two single-ended traces that have current flowing in opposite directions to each other.

![Figure 4](image2.png)

**Figure 4: Differential pair crossing a gap.**

Title-1
At both sides of the plane gap edges, the return currents couple to the traces as they lose the reference path of the plane (see Figure 5). Differential pairs have a robustness for plane gaps; however, as shown in Figure 6, the electric field measured at 5mm above the board is much more significant than the field of narrow trace spacing (see Figure 7). In both Figure 7 and 8, strong radiation (both electric field and magnetic field) is seen at the gap area, especially the wider spacing example.
Figure 5: Return currents on the reference plane couple to the traces (blue and green arrows in the middle).

Figure 6: Electrical field at 5mm above the plane. Significant field emission can be seen in the middle and from the whole area of the plane.
Differential pairs should not cross gaps or splits in their reference planes. However, as can be seen from the above simulations, this can be done without catastrophic consequences. These types of effects are difficult to identify using a traditional signal integrity (SI) simulation with a 2D field solver. However, the current deviation and radiation effects can be captured using a 3D field solver. Once these types of structures have been characterized, instead of re-simulating them every time, it makes sense to check for these problematic items by inspecting the whole board. Such checks can be automated using full-board electrical rule checking engines. Figure 9 is an example of a differential pair crossing over a plane split. In this case, the gap crossing appears to be completely avoidable, and the layout can easily be modified to eliminate this crossing.
Summary
Reference planes are always part of the signal transmission path. Whether they are intentionally placed reference planes close the signal traces, or an unintentional return path like the equipment chassis, current will always flow through these reference planes. The tighter the current loop, the less EMI is produced. Verification of this phenomenon can be done through simulation in a 3D field solver, requiring significant computation time. A more efficient way to eliminate return path discontinuities from a design is to use a rule-checking tool to analyze board geometries to find problems in a matter of minutes. Rules can be created based on the results of 3D analysis, in addition to best practices and other design experience.

Also See
- The EMC blog
- ESD measurements enhance simulations
- Getting EMC design right the first time

Want to learn more? Be sure to attend DesignCon 2016, the premier conference for chip, board, and systems design engineers. Taking place January 19-21, 2016 at the Santa Clara Convention Center, DesignCon 2016 will feature technical paper sessions, tutorials, industry panels, product demos, and exhibits.
Get updates on Twitter, Facebook, & DesignCon Central.