Manage EMI in PCB design: EMI sources and solutions

Minoru Ishikawa - April 02, 2015

With ever-faster interfaces on PC boards, managing electromagnetic interference (EMI) is one of the biggest challenges designers face. There are many possible causes of unwanted emissions. Here are a few examples that can cause EMI issues:

- High-speed trace crossing a reference plane gap?
- High-speed trace routed near a plane edge?
- Reference plane changes for a high-speed trace

Using design rule checks is a quick way to locate potential sources of EMI. Though rule-based checking is not perfect, it provides many user benefits. 3-D EM simulation is commonly used to simulate EMI phenomena. However, the simulation results do not identify the cause of radiation, they only show you how the EM fields are behaving. The source of the radiation usually cannot be found even through close examination of the simulation results.

Rule Checking to Eliminate EMI Sources

Let’s look at an example of using rule checks to locate sources of EMI. The PCB in Figure 1 has three FPGAs, four DDR3 memory devices, and four DDR2 devices. One of the most common sources of EMI is a high-speed trace crossing a gap.
Microstripline is widely used for high-speed traces. It has a reference plane in the adjacent layer of the trace. The return current goes just beneath the signal current (see Figure 2). This trace will have a very stable impedance.

When a trace crosses a gap of reference plane, as in the PCB in Figure 1, the return current needs to detour around the gap. When the gap is too big or wide, the signal is reflected at the gap. (Figure 3)

The looped current forms a loop antenna, which is, unfortunately, an efficient radiator. To find the traces crossing gaps, a rule can be written and executed using a tool such as Mentor Graphics HyperLynx DRC. The rules are simple to write and eliminate the need to prepare more complex IBIS models. In addition to increased performance of the rule-based method, it is much easier for less experienced users to use.

Ease of use, high performance, and no need for models allows the user to run the rule checks as many times as they want over the course of the design. Instead of running simulations at the end of the design (top illustration in Figure 4), designers can run the simulation during and after each phase of the design. This catches problems very early and allows quick and inexpensive rectification of the EMI issue.
3D EM Simulation
Rule-based checking is quick, accurate, and can be run by non-expert users. Still, 3D EM simulation is useful but often requires EMI experts.

Figure 5 shows two differential pairs (A and B) crossing two plane gaps (blue arrows). In this design, the upper pair (A) has gaps closer to its receiver and the driver, and the lower pair (B) has gaps in the middle of their traces. In this case, 3-D electromagnetic simulation is required.

The S-chart in Figure 6 shows the plot for S(21) that was obtained from a Nimbic nWave simulation result. It shows that the channel B (red line) has significant insertion losses at lower frequencies. This indicates strong resonances occur, since the return current lost its path in the middle of the
way. Trace B will cause a more critical EMI issue.

Figure 6: Impedance plot of each of the two differential pairs, A and B.

Conclusion
As shown, rule-based verification and 3D EM simulation complement each other. Rule based EMI checking allows known EMI generators to be searched for and identified. Then, the offending layout can be fixed early in the design when it can be done quickly and inexpensively. For more analysis, full 3D EM simulation can help designers locate difficult issues so they too can be rectified.

Also See
- Build low-EMI prototypes: The essentials
- Manage EMI from high-speed digital interfaces
- PCB design: Reducing errors & increasing efficiency
- Minimize EMI in automotive environments