Impeding Interference

Panasonic improves signal integrity design for a remote surveillance camera using electronics design software from Ansoft.

By Hiroshi Higashitani, Panasonic Electronic Devices Co. Ltd., Japan
Aki Nakatani, Ansoft LLC

In a unique network camera device that permits remote visual monitoring for surveillance and security applications, Panasonic used a standard Ethernet connection to transmit video and audio signals, allowing remote monitoring from any location. The camera was designed to rotate, pan and zoom by commands issued by the user. Within the control electronics in the camera body, three module printed circuit boards (PCBs) were connected by a high-speed, low-voltage differential signaling (LVDS) channel with ribbon cables and associated connectors. After building and testing the initial prototype, the designers of the LVDS network camera realized that device performance would be suboptimal, causing them to face a difficult choice: They could either re-spin and test or adopt a new design approach that involved advanced simulation. With a critical deadline looming, management decided that simulation was the best choice.

Working together with Panasonic, Ansoft created a methodology that enabled engineers to simulate complex high-speed PCBs and meet challenging noise and performance specifications. The team used a reference design board for a consumer electronics device as an example to illustrate how to accurately predict and suppress board resonances and resulting radiated emissions.

Panasonic engineers modeled the full LVDS channel using a combination of HFSS, Nexxim and Ansoft Designer software. The channel included three PCBs (video, mechanical controller and central processing unit) and two Molex® FFP/FPC surface mount connectors. Additionally, the team used the HFSS tool to extract Full-Wave SPICE and S-parameter models for the PCBs. Similarly, they created W-element and 2.5-D planar models for the connectors using Nexxim and Ansoft Designer software. The engineering team then inserted the individual models into a circuit simulator to form the complete channel. With the full channel assembled, the circuit simulator then provided a channel

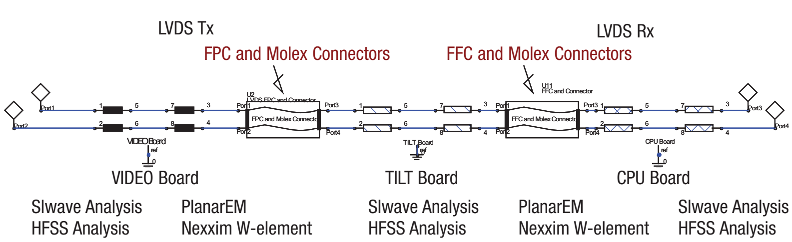
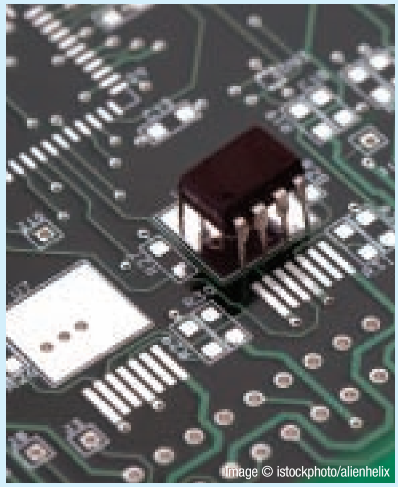


Diagram of PCBs and connectors



Vias are plated holes that connect copper tracks or traces from one layer of a PCB to another.

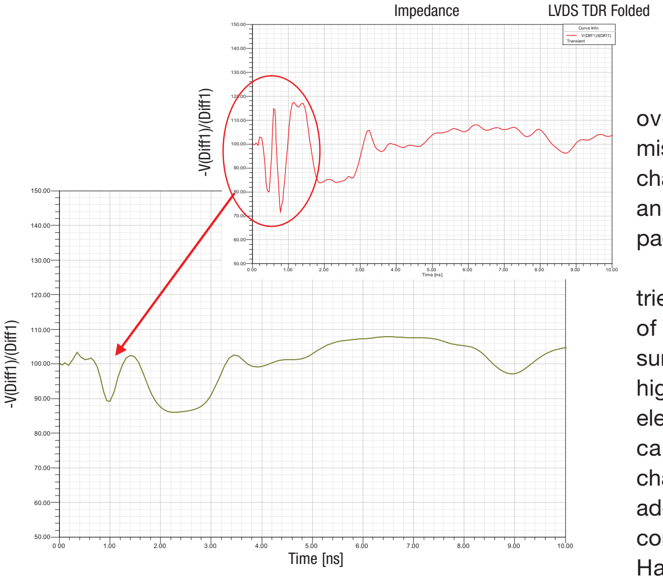
Pads are surfaces on PCB boards to which components can be mounted.

Traces are the electronic pathways that transmit signals from one component to another.

Timing skew occurs when a clock signal travels along traces and reaches its component destinations at different times.

Today's printed circuit board (PCB) designers face competing challenges of smaller, higher-density applications coupled with high-frequency and high-speed signaling. A multitude of standards now exist that utilize high-speed serial signaling. The higher speeds give rise to greater demands on PCB designs to meet signal integrity (SI), power integrity (PI) and electromagnetic interference (EMI) specifications. The challenge becomes especially acute for low-cost commercial devices in which traditional signal-integrity design rules may be ignored in exchange for a board with fewer power and ground planes or a higher-density design with less than optimum signal routing.

SI, PI and EMI design once were considered separate disciplines, each with its own design rules, analysis methods and measurement techniques. A more modern approach is to recognize that there is a strong interdependence among the three and that optimum board design requires an integrated approach. A signal-integrity problem, for example, may lead directly to an EMI problem. This article illustrates the new approach, and the important design considerations, through an overview of an LVDS application.



A comparison of the impedance maps for the original and improved designs for a full LVDS channel; in the later map, impedance is greatly improved.

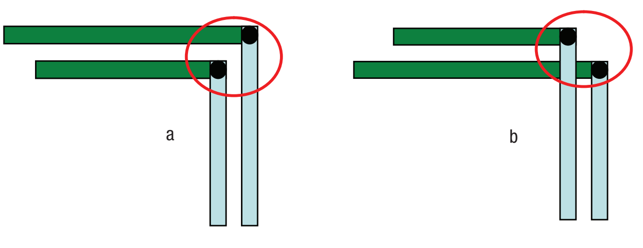
impedance map, similar to time domain reflectometer results. The system's initial design had a significant impedance problem along the video board.

Upon further examination of the video board's layout, the team found that a pad and via were the root cause of the impedance problems. The impedance mismatch was the result of a step change in the width of the trace located near the via. Beyond the impedance mismatch, the team determined that the original trace routing would also lead to skew.

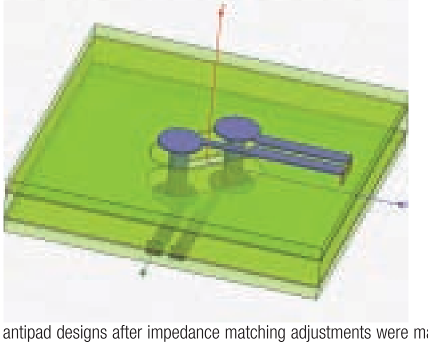
Panasonic engineers addressed the skew and impedance mismatch in two steps. First, they reconfigured the trace routing so that the total length of each trace in the differential pair was equal; this was accomplished by overlapping the traces. They resolved the impedance mismatch, on the other hand, by eliminating the width step change in the routing to the via and by optimizing the pad and antipad radii. This was done by parameterizing the pad and antipad geometries in HFSS software.

Once they identified optimal routing and via geometries, the engineers focused on the impedance peak of one of the connectors. Polyamide strips were placed on the surface of the connector over certain sections. With their higher permittivity, the polyamide strips caused the local electric fields to be more tightly concentrated. Hence, the capacitance of the transmission line increased, and the characteristic impedance fell. Finally, Panasonic engineers added a common mode noise filter to the circuit to reduce common mode signals while permitting differential signals. Having made these changes, they generated a second impedance map and found that the impedance variations were significantly reduced.

By addressing the signal integrity problems in the PCBs and the connectors, the team confirmed that they had improved the channel's electromagnetic interference performance. In the initial design, the LVDS signal was scattered whenever it encountered an impedance discontinuity. The scattered energy had to go somewhere; some of it scattered back toward the transmitter, some of it coupled to other propagation modes, especially common mode, and still other energy coupled into parallel plate resonant modes within the PCB. This energy could then radiate to produce unwanted EMI. Solving the SI problem, therefore, had a direct effect on the radiated emissions of the system. Experimental measurements of the camera's radiated emissions before and after the modifications clearly showed a reduction as well. By adopting a circuit and 3-D electromagnetic cosimulation approach, the design team saved about two months on a second prototype build and about one month on lab measurements. ■



Traces before (a) and after (b) skew matching adjustments were made



Pad and antipad designs after impedance matching adjustments were made