"Silence is Golden"

8 ways to reduce noise on your next PCB

by Dallas A. Dean

Previously published by CMP Media LLC
Printed Circuit Design, Manhasset NY, 1999

Silence is Golden

8 ways to reduce noise on your next PCB

by Dallas A. Dean

Electromagnetic interference (EMI)—i.e. noise—is a common problem that every designer faces when designing printed circuit boards in the computer industry. There are many details that need to be addressed in order to produce a clean board. In a phrase, "the chain is only as strong as the weakest link." And many times, even well designed boards can be improved. A "checklist" of the dangers that each board designer should be aware of at the beginning of a new PCB project is included below. With proper planning, designers can avoid these problem areas, thereby improving board layout and minimizing EMI radiation levels.

Keys to noise reduction

• ***Proper bypass procedures for the ICs and active devices are critical.*** The strategic placement and orientation of components on the board is very important. Some components—especially magnetic devices (filters)—may have more magnetic fields in one direction than another. By placing them at 90 degrees from each other, the fields cancel and radiation can be reduced. Any excess radiation can cause EMI on the board. Proper separation of switching devices from the magnetics also can reduce the noise radiation. The key is the proper placement of components and routing of PCB traces.

• ***Signal rise and fall times of the active devices and clocks must be controlled.*** This can be done with proper terminations to prevent overshoot. If the resistive termination is too high, the signal can become under-damped, causing a faster rise-time and thus overshoot. Improper terminations can produce serious noise spikes, thus increasing the EMI problems.

• ***It is also critical to keep digital lines separate from signal lines.*** Signal paths must be kept away from I/O lines to prevent noise coupling. Switching noises that may exist on digital lines can be radiated over to the signal lines and add to the EMI problem on the signal lines onto the cable.

• ***It is essential to keep differential traces as short as possible to minimize the inductance.*** This can influence the buildup of common mode voltages. A short trace has fewer emissions than a long one. A fat trace has fewer noise emissions than a thin trace. And these lines must be symmetrical and matched for impedance. Do not allow the trace widths to change or the traces to wander from each other. This will cause an impedance mismatch and introduce return loss problems. Signal traces should be kept close to the return path. Return or ground planes should be as solid as possible, free from gaps, and thus create no current loops.

• ***Minimize crosstalk and common mode signals for the best EMI suppression.*** Different signal paths should have the proper separation from each other to reduce any radiation on the board. Balance of the traces is very important

FOR INDIVIDUAL USE ONLY
PCD MAGAZINE'S COPYRIGHT
PCD MAGAZINE'S COPYRIGHT AND DISTRIBUTION

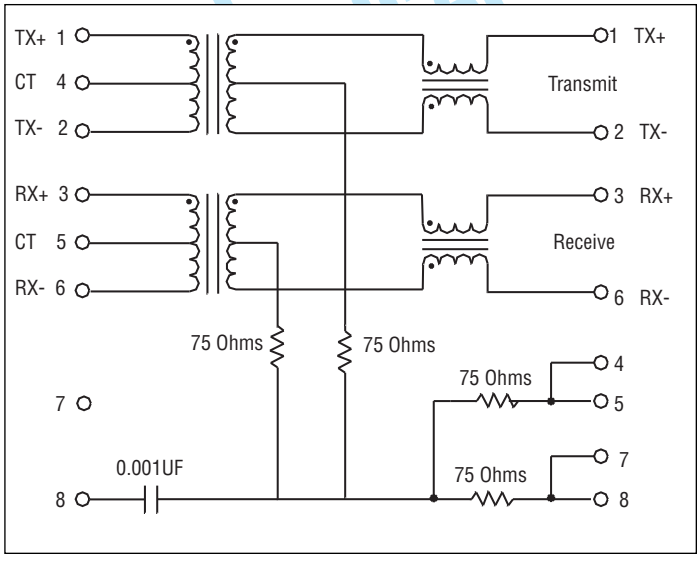

Figure 1—*Common Mode Termination*

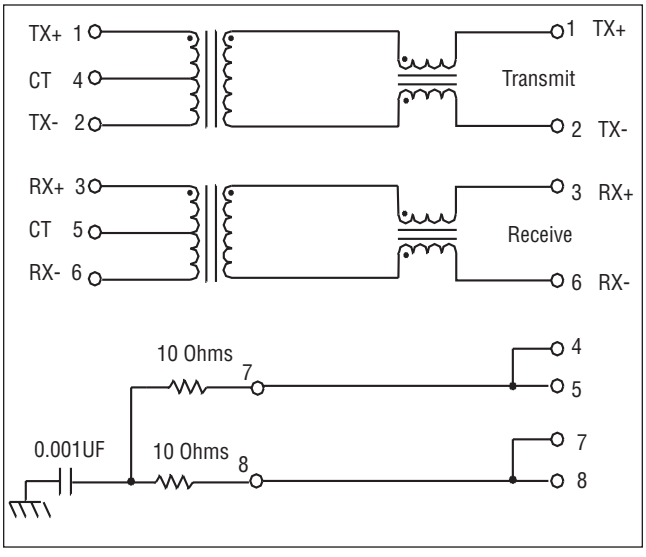

Figure 2—*Abbreviated termination*

Silence is Golden
continued

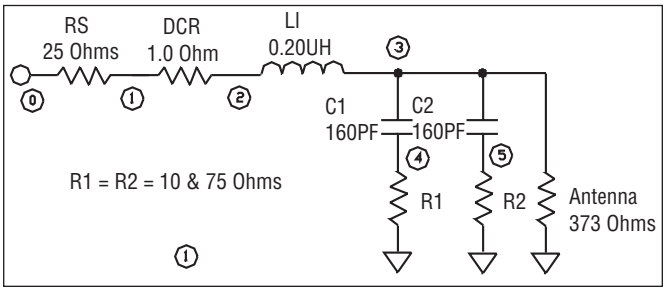

Figure 3—*Model for unused pairs*

in keeping the common mode noise to a minimum. The transformer primary and secondary centertaps are decoupled to ground for the best common mode rejection below 50 MHz. Three resistors (typically 50 ohms) are used to terminate the unused cable pair's common mode impedance. These unused pairs can act as antennas with no termination. Some success has been achieved by varying the resistances between 0 and 100 ohms. This is covered later in the article with curves to show the effects. The transformer primary inductance limits the low frequency response and the leakage inductance and shunt capacitance limit the high frequency response above 50 MHz. Smaller values improve the return loss at the higher frequencies, thus extending the bandwidth.

• ***Avoid extending the ground plane under the magnetic devices to the RJ45 jack.*** This promotes coupling for noise. Non-media signal lines should not be in this area. All differential pair data lines to the RJ45 connection do not require a return path.

• ***Chassis ground can be extended to the interface module to provide additional shielding.*** It should not overlap an analog or digital ground. There should be a gap between the chassis and signal grounds to reduce any coupling. Partition analog power and ground planes. Apply 45-degree trace routing in lieu of 90-degree bends—a common practice. (Forty-five degree bends avoid electric field concentrations that 90-degree bends have.) This also reduces crosstalk.

• ***Run differential or balanced lines as a set of parallel traces.*** Consider the stripline, microstripline, and imbedded microstripline techniques for critical lengths.

With all of the above considerations accounted for, designers should then double-check the schematics to see if any other problems could exist on the road to achieving a noise-free board. One other area to consider is the removal of the magnetic coupling modules and shortening the traces to the RJ45 connector. This can be achieved by integrating the module magnetics into the connector jack. This also enables the size of the board to be reduced.

As an aid to reduce the EMI on a board that may be marginal, the replacement of the RJ45 connector with an integrated magnetic connector composed of a common mode choke—such as the PulseJack developed by Pulse—can reduce the EMI considerably. This is a simple matter of "plug and play." Employing the integrated transformer/common mode choke connector reduces the trace length by eliminating the discrete component.

In an ongoing effort to reduce EMI, there have been many methods used to bypass the noise to chassis ground. This is especially true for the unused wire pairs on the cable between ports. For 10/100 BaseT, these would be cable pins 4, 5, 7 and 8.

One such method most commonly used is the "Common Mode Termination." A schematic is shown in Figure 1. Pins 4 and 5 are shorted together and isolated from the 2KV capacitor by 75 ohms. Pins 7 and 8 are treated in the same manner. Thus, these two "antennas" are decoupled to ground. As an added feature, the center-taps of both

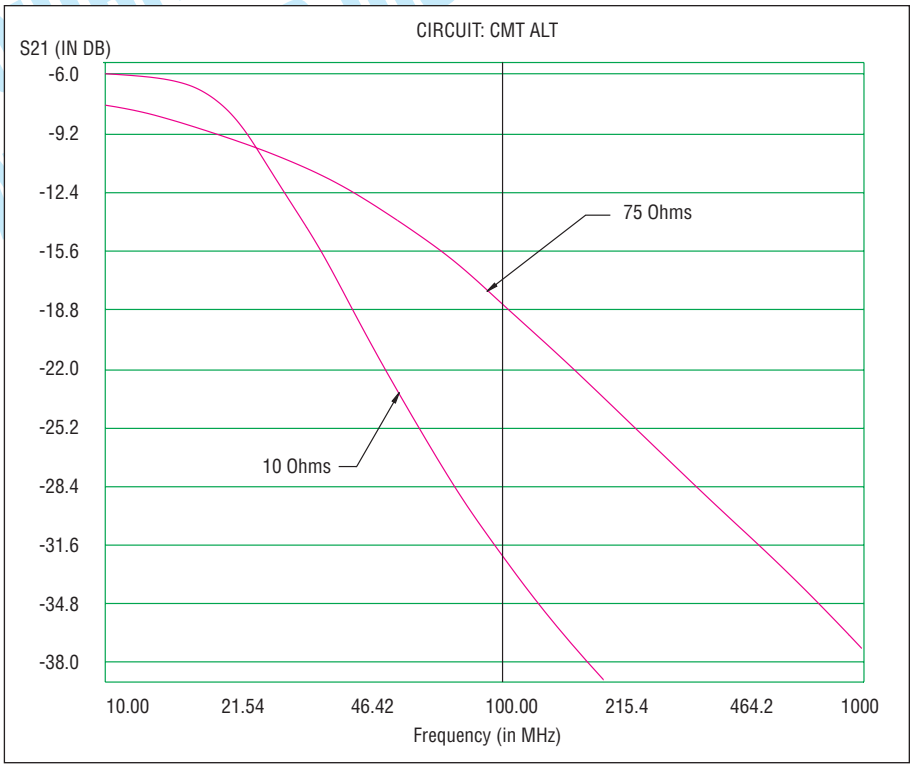

Figure 4—*Graph of unused pair analysis*

PCD PERSONAL USE ONLY
PCD PERMITS COMMERCIAL
DISTRIBUTION

the transmit and receive channel transformers also are decoupled to ground in the same manner.

After modeling the two unused twisted pairs in the cable to view the effect of resistor values on the noise attenuation, one finds that the optimum resistance appears to give the best results using 10 ohms between the unused pairs and the coupling capacitor to ground. See **Figure 2**.

The model used for this evaluation is as shown in **Figure 3**.

The attenuation for each resistive case is shown in **Figure 4**. Note that the 10-ohm resistors affect the attenuation roll-off by reducing the attenuation up to approximately 20 MHz, and then the roll-off is a sharper fall. If the attenuation at 100 MHz is compared for both the 10 and the 75-ohm cases, the 10-ohm version provides about -l2.5 db more attenuation.

Another method of improving the noise reduction, especially at the higher frequency ranges, is the use of the center wire common mode choke in the transmit channel. See **Figure 5.**

The coupling capacitance of this device is balanced from each side to the center wire. Being in shunt with the secondary of the transformer, it aids the distributed capacitance of the transformer secondary winding to better control the return loss of the transmit channel. It also provides a notch of attenuation at the higher frequencies. This frequency range occurs between 700 to 900 MHz. This range can also be controlled by process control. See **Figure 6** for a typical response curve.

The above checklist of areas to consider and avoid was developed to speed the design of a clean board and reduce the time to market. The additional ideas provided—such as improving the EMI by reducing the common mode signal—will only enhance the final outcome by fine-tuning the reduction of noise harmonics. ■

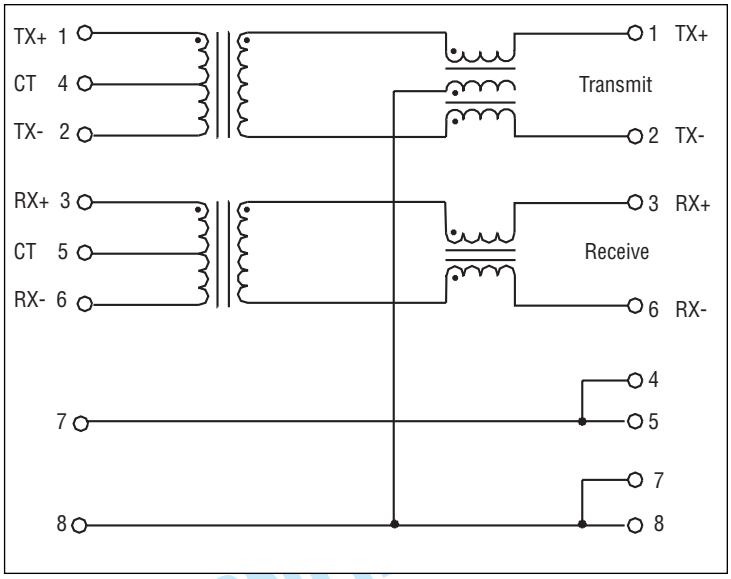

Figure 5—*Common mode center wire*

Dallas Dean is the electrical staff engineer for the Emerging Technology Group at Pulse, a Technitrol company. A 30-year veteran of Pulse, Dean holds a B.S. degree in electro-mechanical engineering from Lowell Technological Institute (Lowell University). He can be reached via e-mail at dallasdean@pulseeng.com.

FOR INDIVIDUAL USE ONLY
PCD DESIGNATES COMMERCIAL
PUBLICATION AND IS FOR DISTRIBUTION

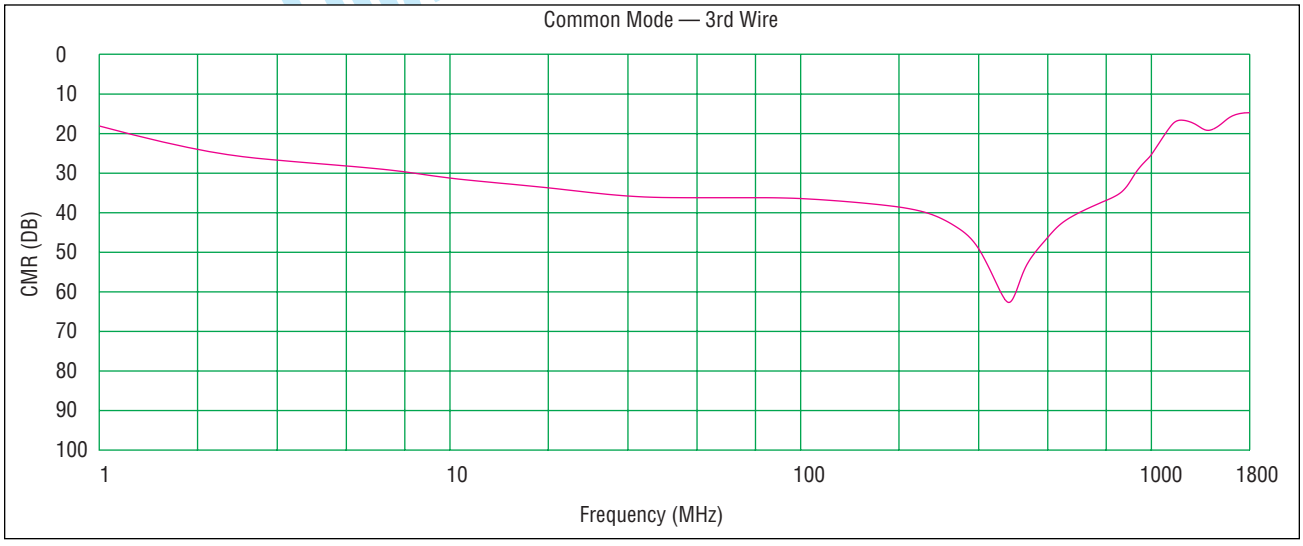

Figure 6—*Results using common mode center wire*