Slots in Planes

Don't Use 'Em!

Douglas Brooks

Has this ever happened to you? The board is finished. The traces are tightly routed and you have done a magnificent job of confining them to the fewest possible number of signal trace layers. Then the engineer calls and tells you he has forgotten one little thing--just one little net that happens to run horizontally across the entire board! It will take *hours* to tweak hundreds of other traces to fit this one in. But, just one little slot in an adjacent ground plane would make room for it and you would be done in minutes. And, after all, it is the engineer's fault, not yours, that you have to create this little slot!

Tempting, isn't it? Well, don't do it! This article will give you three reasons why slots in planes are to be avoided on high speed boards.

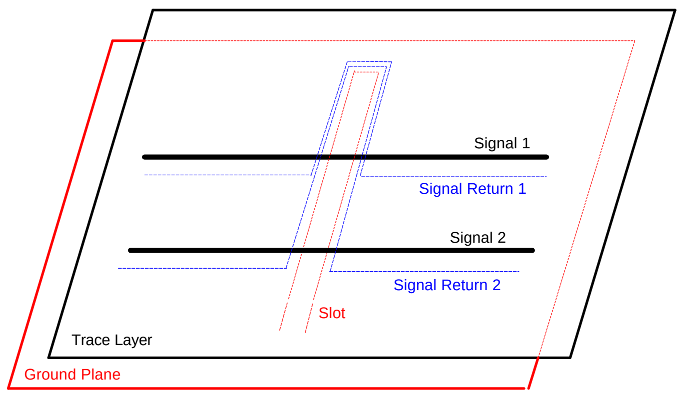

Figure 1.
Allowing signals on trace layers to cross over slots on adjacent reference planes can cause problems in impedance control, EMI radiation, and crosstalk.

Impedance Control:

Signal traces begin to look like transmission lines to signals with very fast rise times. The problem with transmission lines is that reflections, and therefore noise and false triggering, can occur if the characteristic impedance of the line is not controlled over its entire length. If there is a discontinuity in the impedance of the line, a destructive reflection can be caused by that discontinuity.

The characteristic impedance of a trace is determined by its geometry, one element of which is the distance between the trace and an adjacent plane. If all other things are constant, but the distance to the plane changes, the impedance will change at that point and a reflection is likely to occur.

Consider Signal 1 in **Figure 1**. It is referenced to the ground plane along its length except where the trace crosses the slot in the plane. In high speed designs, the return signal for Signal 1 will be on the ground plane directly under the trace for reasons I gave in my January column (Footnote 1). But the slot interrupts the path of the return signal, and it must find a way around the slot. This discontinuity causes an obvious change in geometry that in turn causes a change in impedance. An easy analogy would be if we cut a coaxial cable and spliced it as shown in **Figure 2**. We intuitively know that this is bad practice! So is allowing an impedance controlled trace to cross a slot in a plane!

EMI Noise:

In my January column I also pointed out that one source of EMI radiation is the "loop area" defined by a signal trace and its signal return path. Since destructive radiation can be directly related to loop area, we want to keep loops as small as possible. The case of a signal trace directly over a plane is an excellent example of controlled loop area. The return signal is tightly coupled to the trace and the loop area is very small.

It is clear from Figure 1, however, that if there is a discontinuity in the plane, the signal return path must necessarily move away from the trace. Where it actually goes can become an interesting effort in speculation. The return signal might, for example, find its way through nearby bypass caps to a different plane. In this case we might get lucky and the practical effect of the slot might be small. On the other hand, the return signal may have to travel all the way around the slot. In this case the loop area could be relatively large, causing serious EMI problems. The point is that the path of the return signal is uncontrolled, with subsequent unknown consequences. That's why we don't want to create the situation in the first place.

Crosstalk:

When two traces are adjacent to each other, they can couple unwanted (noise) signals into each other. This cou-

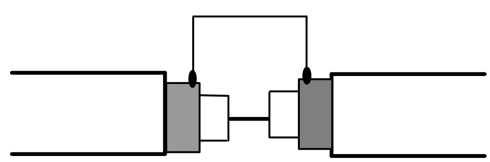

Figure 2
Splicing a coax cable in this manner would likely cause severe reflection prob-

This article appeared in Printed Circuit Design, a Miller Freeman publication, March, 1999

pled noise is called "crosstalk" (Footnote 2). The degree of coupling is inversely related to the square of the distance between the traces. To a large extent, the further the traces are spaced apart the better.

In Figure 1, Signal 1 and Signal 2 are spaced well apart from each other. In a well controlled design their return paths would be directly underneath their respective traces and would necessarily be spaced the same distance as their signals. But if there is a slot in the reference plane, then the return paths have to find a way around the slot. Over this distance they might be *very* close together, even possibly congruent! This creates obvious potential for crosstalk between the signal returns, and under certain circumstances the crosstalk coupling might be *very* high.

Tough Troubleshooting:

It sometimes is not apparent or recognized that there is a slot in an internal plane, especially to an engineer or technician who didn't design the board. After the prototypes are built, the engineer discovers that he has unwanted reflections on a trace, EMI radiation, and crosstalk problems. He checks the traces and verifies that the impedance control guidelines (trace thickness and spacing) have been met, the routing is seemingly good, and the crosstalk control guidelines (trace separations) are correct. The effects of slots are really subtle and difficult to recognize. It doesn't occur to the engineer that the problems are related to the planes rather than to the traces. Some engineers spend an enormous amount of time and, unfortunately, *never* find the real problem in the board design.

Responsible designers don't do this to their associates!

FOOTNOTES:

1. See "Loop Areas: Close 'Em Tight," PC Design Magazine, January, 1999
2. See "Crosstalk, Part 1: The Conversation We Wish Would Stop," November, 1997, and "Crosstalk, Part 2: How Loud Is It?", December, 1997