UltraCAD Design, Inc.

DESIGN NOTE

PARALLELISM AND CROSSTALK

Crosstalk between two parallel traces is caused by a combination of two effects:

Mutual capacitive coupling --- a current (I_C) caused by capacitive coupling between the two traces, and

Mutual inductive coupling --- a current (I_L) caused by inductive coupling between the two traces.

The formulas for these two effects are extremely complex and a function of many interacting variables, including the type of PCB structure (e.g. microstrip vs stripline). Many authors find it more instructive to try to estimate the effects with simulation rather than trying to calculate them with formulas. (See, however, our caution in our Technical Note, "MODELING TOOLS AND SOFTWARE".)

When considering the effects of crosstalk, it is important to consider these factors:

- the degree of capacitive coupling between the traces
- the degree of inductive coupling between the traces
- the direction of the coupled signals
- the reflection of the backward coupled signal

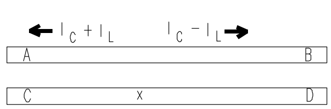

Figure 1

Consider the traces illustrated in Figure 1. A signal propagates down the trace CD from point C to point D. Assume the leading edge of the rise time of the signal is at point x. A signal will be coupled into trace AB at point x that will travel in both directions. The forward coupled signal (called Forward Crosstalk) is proportional to the DIFFERENCE between the two coupling effects (I_C - I_L). The reverse coupled signal, called Backward Crosstalk, is proportional to the SUM of the two coupling effects (I_C + I_L).

In normal PCB applications, the speed of the coupled pulses (i.e. the propagation time for either the forward or backward crosstalk signal) will be the same as that of the driven pulse on CD. For short duration pulses, the width of the forward pulse (forward crosstalk) is equal to the rise time of the driven pulse. The reverse pulse width, however, (backward crosstalk) is equal to twice the propagation time of the length of the coupled (parallel) line. A way to visualize this is as follows: Picture a train moving from C to D whose engine is just passing C and pushes a "bow wave." (analogous to the rise time of the pulse.) A forward crosstalk signal starts at A and moves towards B at the same speed as the engine. But a backward crosstalk signal has also been generated which has just reflected off A and is also moving towards B. As the engine continues towards D, it continues to couple a signal in the parallel line that travels backwards towards A and then reflects back to B again. This continues until the engine reaches D.

The forward crosstalk "bow wave" continues to build over the entire length, so the magnitude of the forward crosstalk signal is proportional to the coupled length. However, since the inductive and capacitive effects tend to cancel in normal PCB applications (since the materials are relatively homogeneous) the signal still tends to be very small. In fact for centered stripline structures the signals exactly cancel and there is no forward crosstalk.

The backward crosstalk signal, however, reaches a maximum value at the point where the coupled length is $t_r/2t_{pd}$ or greater. The magnitude of this signal is hard to predict, but studies done by Digital System Design suggest that a maximum magnitude of about 20% for 2 nsec rise time signals in close coupled lines in microstrip and 12% for 2 nsec rise time signals in close coupled lines in stripline is a reasonable estimate. Increased separation and signal reflections will reduce the size of the signal.

Finally, the device impedance at point A is usually relatively lower than the intrinsic impedance of the trace. So the backward crosstalk (voltage) signal will reflect at point A back towards B with a negative reflection coefficient and a magnitude that will depend on the reflection coefficient:

$$r = \frac{(R_L - Z_0)}{(R_L + Z_0)}$$

The reflection coefficient would normally be relatively high (say .5 to .8).

In normal PCB applications, since the materials are homogeneous, it works out that the inductive and capacitive coupling effects are nearly equal. Therefore, they tend to cancel for forward crosstalk . Furthermore, the immediate backward crosstalk reflection tends to cancel the shorter forward crosstalk pulse. Therefore, forward crosstalk is normally not an issue unless we are talking about very long lines indeed (in which case the effects are very hard to predict). Backward crosstalk may or may not be a problem depending on the impedance of the gates on the coupled line, the termination techniques used, and the specific circuit characteristics of the signals on the coupled line. If the designer has made good decisions regarding the layout of the board, it is entirely possible that backward crosstalk will not be a problem even if arbitrary design rules regarding parallel lengths are violated.

But the special case of signal lines coupling into clock lines (therefore creating potential noise margin problems on clock lines) is of more concern. Clock lines, however, tend to be more powerfully driven and better terminated than other lines, so there is some margin of safety there, too.

Some designers believe that putting a guard band as a shield around traces will help to "shield out" crosstalk problems. It is instructive to note that a guard band necessarily increases the separation between parallel traces by at least a factor of two. Since coupling is an approximately linear function of spacing in microstrip structures and approximately a square function of spacing in stripline structures, the **separation alone** accounts for a two to four-fold reduction in crosstalk! Thus, the effects of a guard band are accounted for in separation distance, rather than electrical shielding, and the space might as well be used for another signal which the designers know from their knowledge of the circuit cannot cause a crosstalk problem. (This is not to say that a guard band should not be used, however, if a controlled signal return path is desired.)
In summary, forward crosstalk is normally not an issue. Backward crosstalk may or may not be an issue. Some ways to minimize the effects of backward crosstalk on a particular line would include:

1. Limit the length of parallel coupling
2. Separate the lines
3. Make sure you know the "real" noise margin of the components you are working with (and that you are not using arbitrary "rule of thumb" guidelines that may not be accurate.)
4. Place lines with higher noise margins adjacent to or between the "driven" lines of interest
5. Pay attention to line termination procedures at both the source and load ends (if the source termination equals the intrinsic impedance of the line, there will be no backward crosstalk reflection.)
6. Note that as a corollary to 5 (above), if a centered stripline structure is used with proper source termination, there will be NO crosstalk at all, neither forward nor backward.)

We have noticed (and are somewhat concerned) that engineers often focus on crosstalk because they sort of understand it and because CAD systems can relatively easily provide "reports" based on arbitrary formulas related to parallel trace lengths. In our experience, the negative effects caused by stubs, impedance mismatches, reflections at stubs, antenna effects, and voltage transients caused by fast rise time switching through inductive trace lengths are all far more significant and cause far more problems than does crosstalk. Yet engineers tend to overlook these other factors because they do not understand them as well. We wish they understood them better!