UltraCAD Design, Inc.

# DESIGN NOTE

## RIGHT ANGLE TRACES

At a recent conference we heard two different speakers talk about right angle traces on circuit boards. They both used defective reasoning. One said right angle turns were OK (which is wrong), the other said they were bad but for the wrong reason! Over two hours were wasted at the conference on irrelevant issues, and the attendees heard too much mis-information. Here is the truth about right angle turns on circuit boards.

### Incorrect Theory:

Consider the trace as shown in Figure 1. It can be shown that the maximum length across the trace at the corner is 1.414 times the normal trace width. When this is plugged into the equation for intrinsic impedance of a trace, it can be shown that the incremental peak change (decrease) in impedance is approximately 12-15% for microstrip and 15-20% for stripline as the trace turns the corner. The argument presented is that this incremental increase in impedance is bad, can set up unwanted reflections, and therefore should be avoided. Consequently, one should avoid right angle trace turns on circuit boards.

Figure 1

Correct conclusion, but for the wrong reason!

### Irrelevant Theory:

Right angle traces on circuit boards haven't looked like Figure 1 since the days of hand-taping mylar! Today, the process starts with a Gerber format file which drives a film process. The lines on the film are created as if they were drawn by a circular dot (this is true whether a vector or laser plotter is used) moved along a path. Figure 2 shows what this looks like when the dot makes a "right angle" turn.

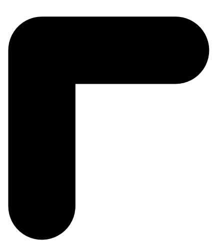

Figure 2

The inside edge forms a sharp right angle, defined by the intersections of two tangents of the dot. But the outside path is curved with an arc whose radius is the radius of the dot itself.

A little geometry, then, shows that the maximum trace width at the corner is only 1.207 times the normal trace width. When this value is plugged into the equation for the intrinsic impedance of a trace, the incremental decrease is only about 8% for microstrip and 10% for stripline. People who have tried to measure the change of impedance at corners (which is difficult to do precisely) have noticed only small perturbations in the signals and concluded that right angle turns have little significant effect on circuit boards.

In fact, however, if the corners of the trace were mitered (as shown in Figure 4), the maximum incremental decrease in intrinsic impedance at each 45 degree leg would only be about 2% for both microstrip and stripline. If the impedance argument is important to you, you gain a significant improvement by mitering the corners. Nevertheless, this is not the primary reason for doing so.

As an aside, note that this argument does not depend on chemical "undercutting" or on "over etching" of the traces during manufacturing. It is inherent in the way the traces are formed on the film during the Gerber and photo finishing processes.

### One Other Irrelevant Theory:

One other speaker said that corners should be chamfered. The reason given was that electrons move like little marbles and they will reflect off the chamfer and "bounce" around the corner. Since they will,

11502 NE 20th, Bellevue, WA. 98004 Phone: (206) 450-9708 FAX: (206) 450-9790

therefore, not reflect backward and cause reflections, chamfered right angle corners are OK on circuit boards.

Frankly we've never seen an electron and we don't know if one reflects around corners like a marble. But it's not relevant anyway!

## Correct Theory:

Consider the trace shown in Figure 3. This trace is very bad on a circuit board and should be avoided. And it doesn't matter if the corners are right angle, 45 degree or chamfered. The reason this trace shape should be avoided has nothing to do with the geometry of the trace itself. It should be avoided because it looks exactly like an antenna. And with fast rise time pulses (remember, the problem is with rise time, not frequency), the trace will radiate badly at this point.

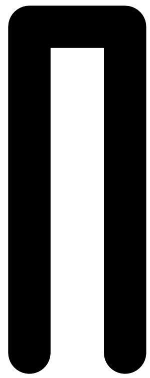

Figure 3

Antenna problems on boards are subtle. They cause radiation into the environment, radiation into other traces, and since antennas receive just as well as they radiate, they are a point of noise susceptibility for their own trace. Many board problems can be traced to signal and clock trace antennas on a board that at first glance look perfectly benign!

**THE REASON YOU DON'T USE RIGHT ANGLE TURNS ON CIRCUIT BOARDS IS THAT RIGHT ANGLE CORNERS BEGIN TO LOOK LIKE ANTENNAS.**

If the 180 degree turn illustrated in Figure 3 is an antenna, the 90 degree right angle turns shown in Figures 1 and 2 begin to look like antennas. Admittedly, they aren't very good antennas, but they are better antennas than are 45 degree corners. And with today's fast rise time pulses, every little bit of protection is important; especially when it takes so little effort to miter the corners of traces. All corners on your boards should be mitered similar to the trace shown in Figure 4 in order to minimize antenna effects. (This issue is not related to impedance, but it is worth noting again that mitered corners also reduce impedance problems at corners.)

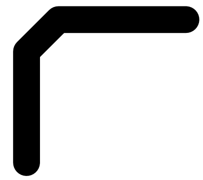

Figure 4

So, if your printed circuit board designer says that right angle corners on circuit boards are all right, or (even worse) that your frequencies are low enough that right angle corners don't matter (remember, it's rise time that matters, not frequency!), get another designer.

*UltraCAD Design, Inc.* routinely miters all corners on all circuit boards. As can be seen from this, and from our other Design Notes, *UltraCAD* is the industry expert when it comes to maximum performance and minimum EMI/RFI radiation and susceptibility on printed circuit boards. If you have had problems in the past with board performance or with FCC compliance issues, give *UltraCAD* a call.