RIGHT ANGLE CORNERS ON PRINTED CIRCUIT BOARD TRACES, TIME AND FREQUENCY DOMAIN ANALYSIS

Mark I. Montrose
Montrose Compliance Services, Inc.
2353 Mission Glen Dr.
Santa Clara, CA 95051-1214
United States of America

ABSTRACT

For years, rules of thumb were provided in published literature stating 90 degree corners create radiated EMI. In addition, concerns exist regarding signal integrity for high-speed digital signals traveling down a printed circuit board (PCB) trace. High-speed is defined in this paper as a signal with an edge rate much faster than one nanosecond (1 ns), generally in the mid-to-low picosecond range and greater than 100 MHz. Rules of thumb are stated without justification if they are necessary or whether EMI compliance or signal integrity is jeopardized. These concerns are based on word-of-mouth, theoretical models or the mathematics of Maxwell's equations. Computer simulation of PCB traces with various configurations have been presented in published literature based on models that in almost every case does not represent *real-life* or *actual* electrical parameters found in PCB designs. These parameters include stackup assignments, creation of common-mode energy, component driver models, distance spacing of a trace referenced to a RF return path, or incorporation within a metallic enclosure. Research generally considers only the time *or* frequency domain, not both.

INTRODUCTION

In order to study how transmission lines function, consideration must be given to investigating both time and frequency domain characteristics of the network. When a signal propagates through a transmission line, commonly identified as a PCB trace, the mode of transmission is that of an electromagnetic wave, not voltage, current or electrons. Maxwell's equations describe the characteristics of this electromagnetic wave. A closed loop circuit allows a signal to travel from source-to-load along with a mandatory return path from load-to-source. This circuit will contain both DC and AC (RF) components simultaneously. Design engineers usually consider only propagation delay, frequency of operation, capacitive overheads, dielectric losses, impedance control, and similar parameters during schematic design. When a signal propagates down a transmission line (trace) in the *time* domain, a *frequency* domain component is simultaneously observed with appropriate instrumentation.

The following is examined.

1. Effects of a signal propagating down a transmission line (trace) in the time domain.
2. Effects of trace width and magnetic flux distribution created with various corner configurations.
3. Radiated emissions with and without a RF return path.
4. The frequency at which corners play a significant role in the creation of RF energy.

PCB DESIGN PARAMETERS

Two separate PCBs were used for analysis. The assembly in Figure 1 was designed to simulate a PCB using actual design parameters. These parameters include a double-side board at 0.062 inches thick (15.7 mm) with microstrip trace widths at 5 mils (0.005 inches/0.13 mm), 10 mils (0.010 inches/0.25 mm), and 20 mils (0.020 inches/0.50 mm). There are six corners per trace. Each trace was routed at 90 degree, 45 degree and bend radius (round) for a total of nine traces, each with six corners per trace. The routed length was 18.0 inches (45.7 cm). The impedance of the traces were approximately 150 ohms, 130 ohms, and 110 ohms respectively. These impedance values are typical for a double-sided PCB. This impedance difference between trace and source will be reflected in the test results with instrumentation being at 50 ohms impedance.

The PCB shown in Figure 2 was design to evaluate effects of two corners per trace route using various configurations. All traces were designed to be exactly 50 ohms in order to match the impedance of the test instrumentation. In other words, this is a *specially designed PCB for evaluation purposes only*, and not one that reflects a real-design. Traces on this board were 7 mils wide (0.007 inches/0.18 mm) on a four-layer stackup. This stackup with a ground plane on layer two provides exactly 50 ohms trace impedance. Trace length is 8 inches (20.3 cm). A double-sided or four-layer PCB stackup with microstrip traces is generally never exactly 50 ohms due to physical dimensions and construction requirements required for this particular layer stackup assignment. It is to be noted that the results from this board provide only an intuitive insight into the effects of corners within a PCB. For measurements that are of use to practicing engineers using real construction parameters, the data from the PCB shown in Figure 1 provides greater accuracy.

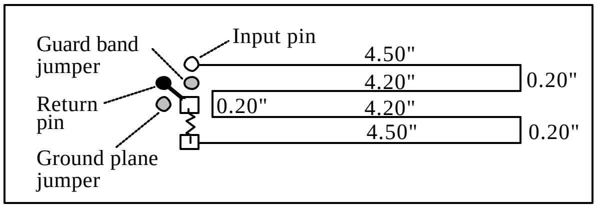
As observed in the time domain plots, an impedance discontinuity of a significant nature occurs at the launch point or location where the network analyzer (PCB #1) and Time Domain Reflectometer (PCB #2) interfaces to the traces through a connector. This "glitch" is identified in the plots. An actual PCB would not have this large impedance discontinuity.

TIME DOMAIN ANALYSIS

When performing time domain analysis, it is necessary to determine if an impedance mismatch within a transmission line will cause signal integrity problems. This concern lies with the known fact that there will be a decrease in *Zo*, the characteristic impedance of the trace. This decrease is detailed by Eq. (1). The inductance of the trace decreases at corners while the capacitance increases. With this knowledge, it must be determined if the impedance change at a corner using a particular routing geometry will cause a functionality concern to exist. Also, when a signal propagates in a transmission line, it does so at a specific velocity of propagation. The speed of an electromagnetic wave through a dielectric material with an effectivity relativity permittivity, e_r of 4.3 (typical value of FR-4 at 1000 MHz) [4] are for microstrip topology 1.65 ps/inch (4.18 ps/cm) and 1.43 ps/inch (3.63 ps/cm) for stripline.

The signal trace routed stripline propagates slightly slower than microstrip as the transmission line is completely surrounded by a dielectric material, whereas microstrip has approximately 50 percent of the dielectric material consisting of air.

$$Z = \sqrt{L/C} \qquad (1)$$



Trace widths: 0.005", 0.010" and 0.020" (0.0013, 0.003, 0.005 r
Corners are at 90 degrees, 45 degrees and round.
Total of nine traces provided for all configurations.

Figure 1. PCB #1.

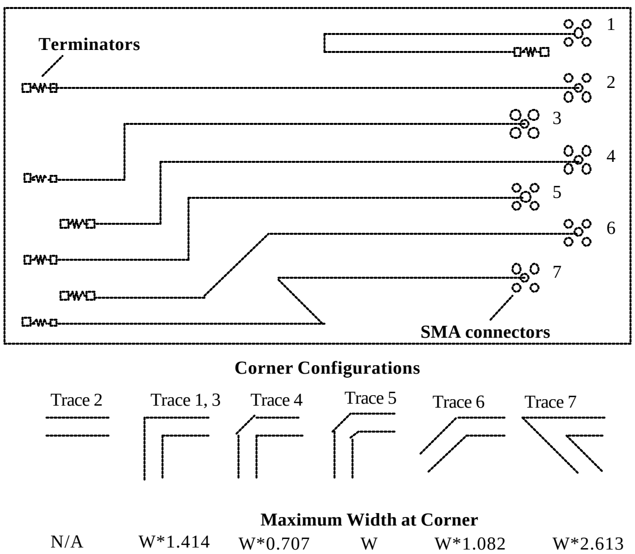


Figure 2. PCB #2.

The primary effect of a right-angle bend, or 90 degree corner (illustrated by Trace 1 and 3, PCB #2), referenced to a RF return path or image plane, is some amount of parasitic capacitance to ground, described by Eq. (2) [5], where w is trace width (inches), e_r is relative permittivity of the substrate, *Zo* is trace impedance, and *C* is in pF.

$$C = \frac{61 * w\sqrt{e_r}}{Z_o} \qquad (2)$$

Assume *Zo* is 65 ohms, (a typical value of a PCB trace), e_r is 4.3, and w is 0.007 (0.18 mm) inches. This results in a capacitive increase of C=0.014 pF, a value that is small enough to not cause concerns for signals propagating through the transmission line below 10 GHz. With such small increase in capacitance of the corner, signal integrity concerns should not exist, unless certain design applications require sub-picofarad values to be included with lumped, distributed capacitance values for consideration with simulation programs. In actual practice, the capacitance from this small increase is only important if one wants to perform simulation modeling requiring an answer of three or four decimal place accuracy. In reality, the increase in capacitance value of a via can be neglected for most applications.

The data taken for both PCB configurations are nearly identical. Only a small difference in impedance discontinuity is present between the three configurations – 90 degree, 45 degree and round. The small difference between corner configuration is *not enough* to cause signal integrity concerns for the practicing engineer. Because of similarity between configurations, it is difficult to identify specific traces for clarity reasons in the plot. To minimize the number of plots that could be provided herein, all three corners are superimposed into one figure, with the worst case configuration provided. All 27 plots were nearly identical.

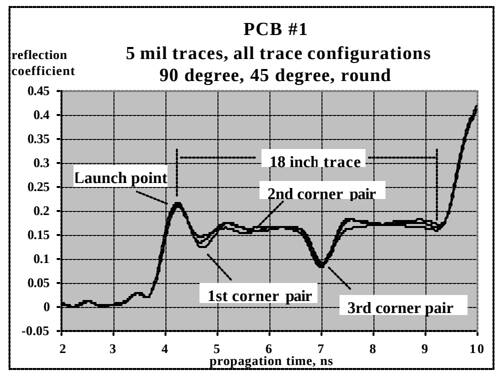


Figure 3. Impedance discontinuities – PCB #1.

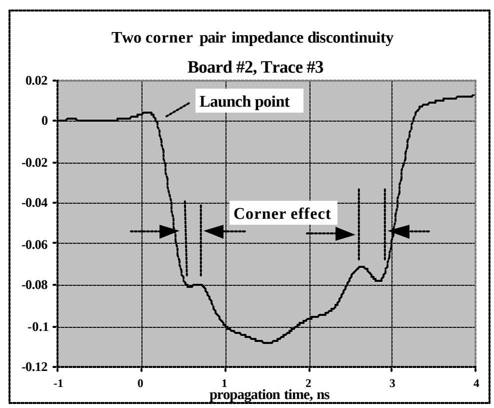


Figure 4. Impedance discontinuities – PCB #2.

It is difficult to show in great detail the impedance discontinuity glitch in PCB #1, using a network analyzer. A TDR for PCB #2 provides a close-up of what the glitch of PCB #1 looks like. For each corner discontinuity, the physical placement of two corners is in close proximity. With the electromagnetic wave traveling at 0.61 in/ps (0.24 cm/ps), the two corners appear as a single discontinuity to the test instrumentation. For an exact value, we divide the total discontinuity of the two traces by two, thus providing us the magnitude of the impedance discontinuity, and approximately how long the discontinuity lasts, within several picoseconds of accuracy.

At each corner, the two glitch combination is approximate 30 ps. Thus, there is approximately a one 15 ps discontinuity for only one corner. The magnitude of the impedance discontinuity, based on Eq. (1), for this very brief time period is approximately 8-10%. If the impedance tolerance of the PCB structure, using FR-4 material is ± 20%, how much concern should an engineer place on impedance discontinuity with a 30 ps or faster edge transition time or 10 GHz spectral bandwidth. With this situation, it is difficult, if not "impossible," to measure impedance discontinuities for corners on a PCB without taking into consideration velocity of propagation of the electromagnetic wave and distance spacing between corners. In order to measure a single impedance discontinuity, or to observe the duration of the signal traveling through the corner, the time delay must be several time constants between corners to acquire optimal signal resolution. This would result in an extremely long trace length in a test PCB to observe this effect, which is not realistic in actual practice.

At time $2td$, (round trip propagation time), the reflection of the signal, measured at the launch point is shown as the reflection coefficient in the plots, defined by Equation (3).

$$r_s = \frac{Z_s - Z_o}{Z_s + Z_o} \qquad (3)$$

where rs = reflection coefficient, Zs = output impedance of the driver and Zo = characteristic impedance of the trace. This is the value that determines the maximum amount of impedance discontinuity that falls within the tolerance range of the PCB as actually measured.

FREQUENCY DOMAIN ANALYSIS

A transmission line requires a RF return path to be present. For PCB #1 (Figure 1), two selectable return paths are provided. These two paths are a ground plane at 0.062 inches away (0.062 inches/15.7 cm) and a guard band using ground fill at 0.005 inch spacing (5 mils/0.13 mm). Depending on how shunt jumpers were configured, each trace was referenced to free space, to the ground plane, or to a guard band located as close as possible to the routed trace, within manufacturing tolerance. These strappable options provide insight on how transmission lines react using various RF return paths configurations, related to radiated emissions. For proper functionality, the value of trace impedance is referenced to a return path. The configuration without a return path simulates a monopole antenna.

With no RF return path, radiated emissions were excessive. With a RF return path, either ground plane or guard band, emission levels dropped significantly on the average of 20-30 at specific frequencies. As reported in [2, 3], flux minimization/cancellation within a transmission line occurs when a RF return path is present. When comparing the RF return path (ground plane) compared to the guard band, nearly identical results were observed. The difference between the two is on the order of 2-4 dB that varied between specific frequencies, and not across the entire spectrum. Thus, only one set of plot is provided herein due to similarity. The guard band did *not* out-perform the ground plane located at a further distance away. It was assumed that a guard , 12 times closer to the trace than a plane on the bottom side of the board would provide enhanced flux minimization performance, which was not the case. Within [3], it was shown that nearly all of the flux present is reduced if a solid RF return path was directly adjacent to the transmission line, similar to that of a guard band or a ground plane a significant physical distance away.

Forty (40) MHz harmonics were injected into the 18 inch (45.7 cm) transmission line in the frequency range 30-1000 MHz. An 18 inch trace has a $\lambda/4$ resonance at approximately 160 MHz, a harmonic of 40 MHz. A baseline measurement was taken to determine the magnitude of the injected signal with a 50 ohm terminated antenna; coax and resistor. This baseline plot, Figure 5, was compared against all trace configurations for actual amplitude of radiated emissions.

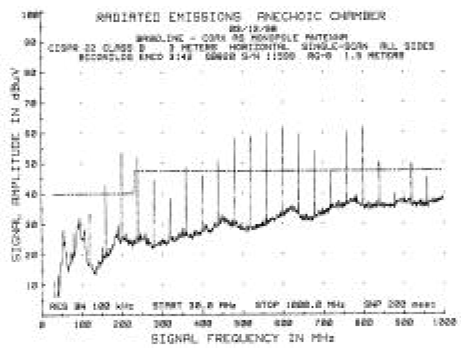


Figure 5. Baseline measurement of radiated emissions

It is noted that the limit line shown in all plots have "no significant meaning!" The limit line was placed within the plot only for the purpose of providing a reference to compare data. The differences between plots were minor, as all plots were nearly identical when compared against other similar configurations. The right angle corner for all three-trace widths is compared against all other right angle corners. The same occurred for the 45-degree corner as with the bend radius. Figure 6 illustrates the worst case plot, the 0.005 inch (0.13 mm) trace, 90 degree corner *without* a RF return path. Figure 7 shows the same trace referenced to ground. Figures 8 and 9 compares radiated emissions for different trace widths, same 90 degree corner configuration. As expected, the 0.005 inch (0.13 mm) trace radiates a significantly greater amount of energy than the other two trace widths.

In Figure 6, radiated energy is observed throughout the frequency spectrum, especially in the lower frequency range. Figures 7-9 has less emissions, as these are referenced to a RF return path (ground plane). These plots validate the need for a closely spaced RF return path, or image plane.

When comparing trace configurations for radiated emissions using PCB #1, mixed results were observed. Each trace width *must be evaluated* against the same width trace in order to make sense of the data. It is not possible to compare the magnitude of emissions between different trace widths using the same corner configuration. This determines if corners radiate above a baseline reference, not absolute magnitude of the signal!

The 0.005 inch (0.13 mm) trace had radiated emissions significantly greater than the 0.010 inch (0.25 mm) and 0.020 inch (0.50 mm) in the frequency range 40-300 MHz. This confirms our analysis that the smaller the trace width, a greater amount of RF energy is present. This greater amount of RF energy is attributed to the

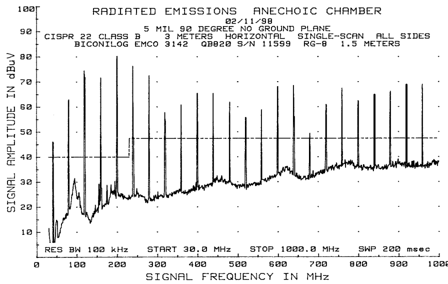


Figure 6. 0.005 inch (0.13 mm) trace - no RF return path

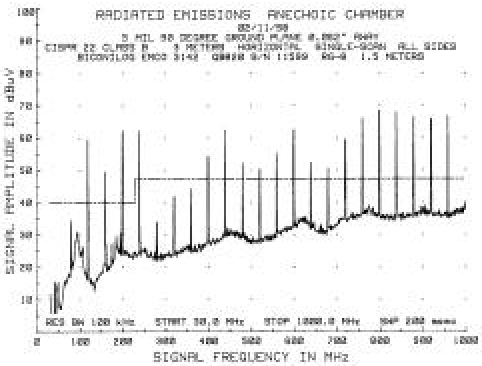


Figure 7. 0.005 inch (0.13 mm) trace - RF return path

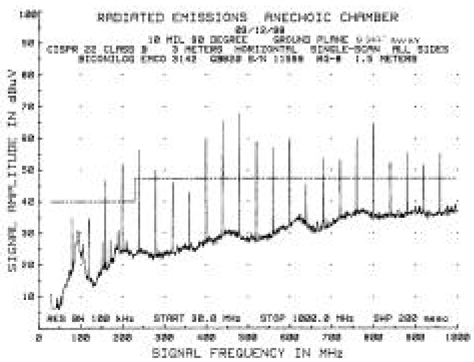


Figure 8. 0.010 inch (0.25 mm) trace - RF return path

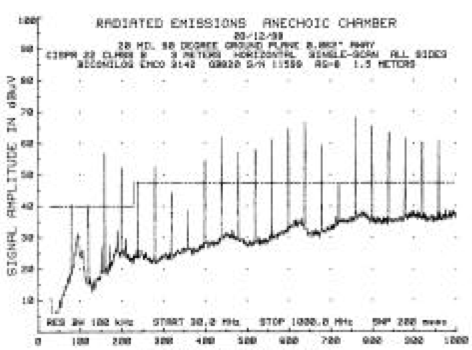


Figure 9. 0.020 inch (0.50 mm) trace - RF return path

impedance mismatch between the transmission line (151 ohms) and signal generator providing RF stimulus input (50 ohms). Other variations within the plots was attributed to difficulties in instrumentation and data acquisition. These differences are considered minor in relation to the overall concept being presented. Throughout the frequency spectrum, the amplitude of measured signals varied making exact analysis difficult. It was observed that all trace configurations had an unusually greater amount of significant radiated emissions from 750 MHz on up.

The magnitude of emissions was not significantly higher with one trace configuration over another. Due to measurement uncertainty, all trace geometries produced radiated emissions that were between 5-10 dB in magnitude above the reference baseline (Figure 5) in the frequency range from 30 MHz to 750 MHz. Above 750 MHz, radiated emissions appeared to be present with significant amplitude. It is concluded that various corner configurations will not start to significantly radiate RF energy until approximately 750 MHz, and then at very low levels, compared to active digital logic that is always present in the design.

CONCLUSION

Time domain (signal integrity concerns): There are no measurable reflections from 90 degree, 45 degree or round corners. In theory, and by mathematical analysis, the impedance of a corner will decrease by a calculable amount. This impedance change is not sufficient to be measured with a 3 GHz bandwidth network analyzer. The velocity of propagation of a signal within the transmission line (trace) is oblivious to the discontinuity unless one designs signals in the upper Gigahertz frequency range or use edge rates faster than 15 ps.

Frequency domain (EMC compliance): Radiated emissions exist, however, measurements up to 1 GHz does not show an increase for 90 degree, 45 degree or round corners that is of any significant amount *greater* than the level of uncertainty of the measurement equipment. The average radiated emissions were approximately 5 dB. The discontinuities within component packages, connector pin-outs, layer jumping of routed traces, vias and common-mode currents within the transmission line will radiate at levels that *far exceed* any measurable effects from any corner configuration. Corners do not appear as radiated emissions until the upper MHz range. The magnitude of the radiated signal measured is minimal. It is difficult, if not impossible to accurately measure radiated emissions from any trace corner configuration.

REFERENCES

1. Montrose, M I. *Printed Circuit Board Design Techniques for EMC Compliance*, 1996. NJ: IEEE Press.
2. Montrose, M. I. *EMC and the PCB – Design, Theory and Layout Made Simple*. 1999. NJ: IEEE Press.
3. Montrose, M. I. "Analysis on the Effectiveness of Image Planes within a Printed Circuit Board." Proceedings of the IEEE International Symposium on EMC, 1996. pp. 326-331.
4. IPC-D-2141. *Controlled Impedance Circuit Boards and High Speed Logic Design*, Institute for Interconnecting and Packaging Electronic Circuits, 1996.
5. Edwards. T.C. Foundations for Microstrip Circuit Design. NY: John Wiley and Sons, 1983.