Time and Frequency Domain Analysis for Right Angle Corners on Printed Circuit Board Traces

Mark I. Montrose
Montrose Compliance Services
2353 Mission Glen Dr.
Santa Clara, CA 95051-1214

Abstract*:* For years, rules of thumb were provided in published literature stating 90 degree corners create radiated EMI. In addition, concerns exists 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. These rules of thumb are stated without justification if they are necessary or whether EMI compliance 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 an RF return path, or incorporation within an enclosure. Research generally considers only the time *or* frequency domain, not both.

Introduction

In order to study how routed traces perform within a PCB, consideration is given to investigating both the time and frequency domain of a circuit. When a signal is sent down a transmission line, commonly referred to as a trace, the mode of transmission is that of an electromagnetic wave, not voltage or current. This electromagnetic wave exists based on Maxwell's equations. A closed loop circuit allows a signal to travel from source-to-load along with a return path from load-to-source. This circuit contains both DC and AC (RF) components. 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.

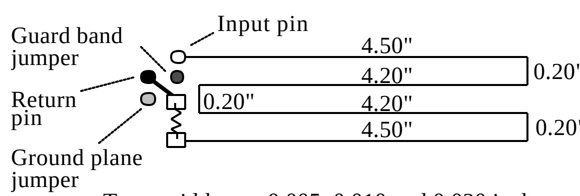
1. Effects of a signal propagating down a PCB trace in the time domain.
2. Effects of trace width and magnetic flux distribution created with various corner configurations.
3. Radiated emissions with and without an RF return path.
4. The frequency at which corners play a significant role in the creation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

PCB Design Parameters

Two separate PCBs were used for analysis. The assembly in Figure 1 was designed to simulate an actual PCB using real-life parameter. These parameters include a double-side board at 0.062 inches thick (0.02mm) with microstrip trace width at 5 mils (0.005 inches/0.0013mm), 10 mils (0.010 inches/0.003mm), and 20 mils (0.020 inches/0.005mm). 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 of each trace was 18.0 inches (457.2mm). The impedance of the traces were approximately 150 ohm, 130 ohms, and 110 ohms respectively. These impedance values are typical for a double-sided PCB.

The PCB shown in Figure 2 was design to evaluate the effects of only two corners per trace using various routed configurations. All traces were designed to be exactly 50 ohms in order to match the impedance of the test instrumentation. Traces were 7 mils (0.007 inches/0.0018mm) wide and situated on a four layer stackup designed to give exactly 50 ohms trace impedance. Trace length is 8 inches (203mm). A double sided, or four layer PCB with microstrip traces can never be exactly 50 ohms due to the dimensions required for layer stackup assignment along with physical construction requirements. It is to be noted that the results from this board provides only an intuitive insight into the effects of corners within a PCB. For measurements that are of use to engineers using real construction practice, the data from the PCB shown in Figure 1 provides greater accuracy.

As observed in all time domain plots, an impedance discontinuity of a significant nature occurs at the launch point or location where the network analyzer interfaces to the traces through the interface connector. This "glitch" is identified in the plots. An actual PCB would not have this large impedance discontinuity.



Trace widths are 0.005, 0.010 and 0.020 inches
Corners are at 90 degrees, 45 degrees and round
Total of nine traces required for all configurations

Figure 1. PCB #1

Time Domain Analysis

When performing time domain analysis, it is necessary to determine if an impedance mismatch will cause a signal integrity problem. This concern lies with the known fact that there will be a decrease in *Zo*, the characteristic impedance of the trace. This decrease is related to Equation (1). The inductance of the trace decreases at corners while the capacitance increases. With this

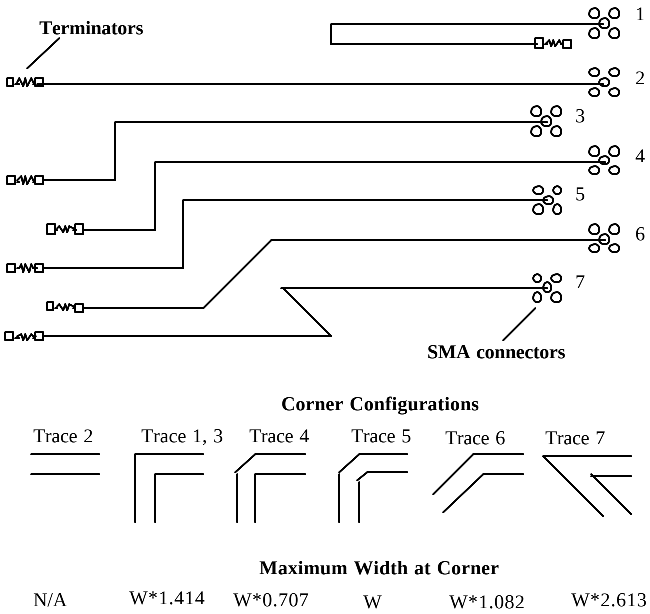


Figure 2. PCB #2

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 through 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] will be for microstrip topology 1.65 ps/in. (4.18 ps/cm) and 1.43 ps/in. (3.63 ps/cm) for stripline. As observed, 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.

$$Z = \sqrt{L/C} \tag{1}$$

The primary electrical effect of a right-angle bend, referenced to a plane, is some amount of extra parasitic capacitance to ground and is described by Equation (2) [5] where w is trace width (inches), e_r is relative permittivity of the substrate, *Zo* is the trace impedance, and *C* is in pF. Assume *Zo* is 65 ohms, a typical value of a PCB trace, e_r is 4.3, and w is 0.007 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.

$$C = \frac{61 * w\sqrt{e_r}}{Z_o} \tag{2}$$

The data in the individual plots are nearly identical indicating that only a minor impedance discontinuity is present between all three corner configurations; 90 degree, 45 degree and round. Because of this similarity, it is impossible to identify specific traces in this paper for clarity reasons. To minimize the number of plots that could be provided herein, all three corners are superimposed into one figure. Observe an approximate 30 ps glitch at each corner discontinuity. Once the impedance discontinuity occurs within the transmission line and the signal has traveled through the corner, the line resumes its designed impedance value which is identified as 0 in the reflection coefficient axis. This means if the transition time is slower than 30 ps (33 GHz) signal integrity concerns are no longer a consideration, related to corner geometry only.

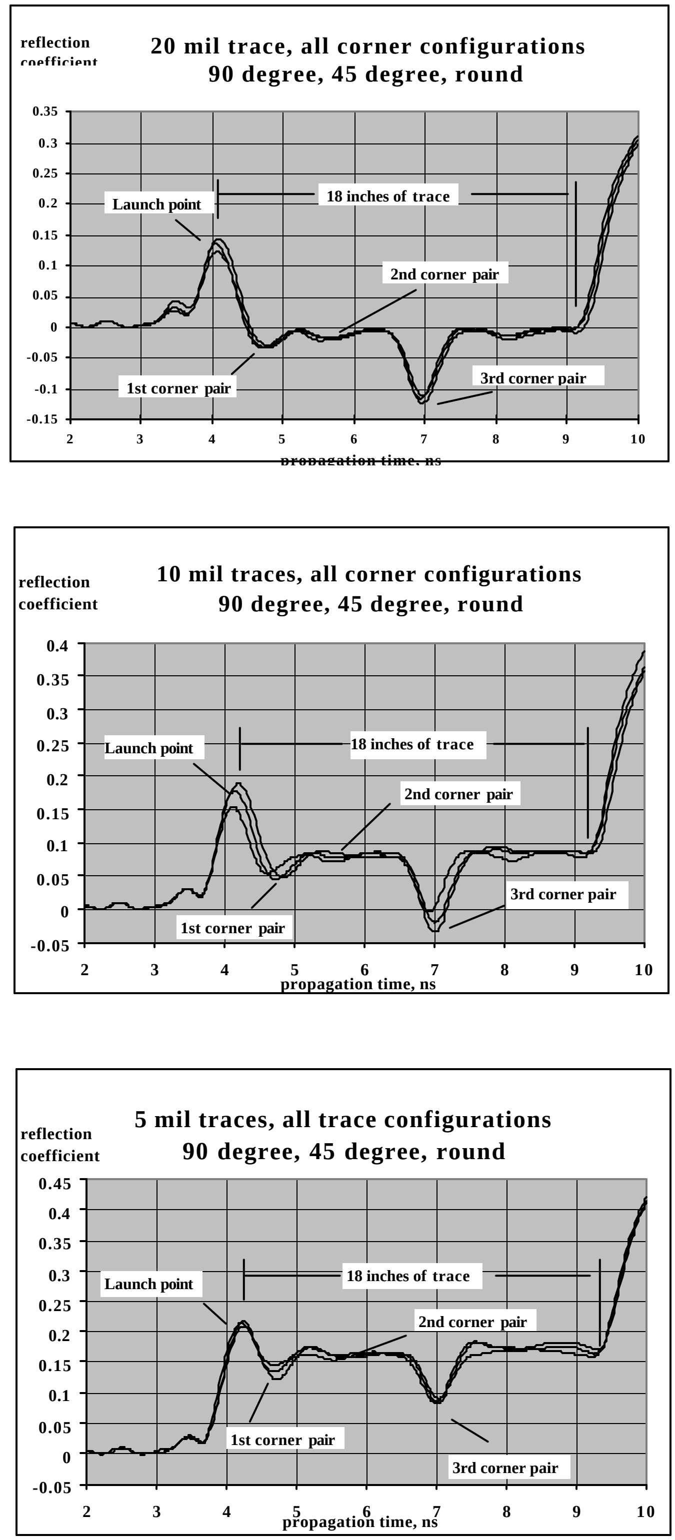


Figure 3. Impedance discontinuities
Various corner configurations plotted together

Notice that for each trace width configuration, 5 mils, 10 mils, and 20 mils, the reflection coefficient baseline value differs. This is because the impedance value of the 20 mil trace (≈110 ohms) is closer to the characteristic impedance of the network analyzer (50 ohms) than the 10 mil trace (≈130 ohms) or the 5 mil trace (≈150 ohms). At time $2td$, (round trip propagation time), the reflection of the signal as 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.

Upon examination of the reflection coefficient value for each trace width and comparing it to Equation (3), take into consideration that we are looking at only ½ of the reflected wave. The value of rs must be divided by 2. For use of Equation (3), Zo=50 ohms and Zo is approximately 100, 130 or 150 ohms, trace width dependent. The coax was approximately 50 ohms (including the impedance of the launch connector. Trace impedance value is approximate die to manufacturing tolerances, hence calculated values of $rs/2$ is almost identical to the plotted value.

A design oversight was accidentally incorporated in board #1 which was discovered during testing. This design oversight had two corners in three separate places in the serpentine trace located in close proximity to each other (0.20 inches), typical of an actual design layout. With the velocity of propagation of an electromagnetic wave at 0.60 in/ps (1.65 ps/in), the two corners appear to the electromagnetic wave as a single impedance discontinuity. This means if multiple corners are physically located in close proximity to each other and the speed of the signal propagating down the trace is fast, a lumped inductive and capacitive load appears to the propagating signal. This lumped load will result in negligible effects related to signal integrity (time domain concerns). The same observation was found with board #2. With this observation, 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.

In Figure 2, board #2, various corners are shown illustrating the maximum width of a corner for different configurations. The maximum trace width is 1.414 (square root of 2) times the nominal width. The effect this has on the characteristic impedance of the trace, Zo, varies (among other reasons) to create a 15-20 percent decrease of nominal impedance. Most PCBs are designed to have a tolerance of +/- 10 percent. A minor impedance glitch is thus presented to the trace. This impedance discontinuity is estimated to occur for less than 15 ps (per corner). With an impedance discontinuity for such a short time period, it is difficult to accurately measure this effect. If it is difficult to measure this effect, should a design engineer worry about signal integrity issues from corners below 30 GHz?

Frequency Domain Analysis

In order for a transmission line to function, a return path must be present. For board #1 (Figure 1), two selectable return paths are provided. These two paths are a ground plane at 0.062 inches away (62 mils/0.02 mm) and a guard band using ground fill at 0.005 inch spacing (5 mils/0.0013mm). Depending on how shunt jumpers were configured, each trace was referenced to free space, to the ground plane, or to the guard band. This strappable option provides insight on how corners react using various RF return paths configurations as they relate to radiated emissions.

A total of 27 plots were taken for the nine configurations of board #1 and seven plots for board #2. The configurations for board #1 were: no ground plane, ground plane 0.062 inches away (bottom side of the board), and ground fill performing the function of a guard band at 0.005 inches away. The guard band was located as close as possible to the routed trace, within manufacturing tolerance. Measured results were expected. With no ground plane (RF return path), RF emissions were excessive. With an RF return path, emission levels dropped significantly on the average of 20-30 dB in the lower frequency range. Both configurations of the RF return path provided similar results. As reported in [2, 3], flux minimization/cancellation within transmission lines occur when an RF return path is present. Locating the RF return plane physically close to the transmission line will provide flux minimization/cancellation. When comparing the RF return path (solid plane) compared to the guard band, nearly identical results were observed. The difference between the two is on the order of 2-4 dB which varied between different frequencies. Thus, only one set of plots is provided herein. The guard band did *not* out perform the ground plane. It was verified that the RF current density distribution radiating off the edge of the trace was equal to the ground plane located some distance away. It was assumed that with a guard band being 12 times closer to the trace than a plane on the bottom side of the board, there should be enhanced flux minimization performance. Within [3], it was shown that nearly all of the flux present would have been reduced if a solid return path was directly adjacent to the transmission line than that of a guard band or a ground plane a significant distance away.

To illustrate current density distribution of a transmission line referenced to a plane, Figure 4 is provided. A reference plane allows RF current to return to its source from the load. Current distribution in microstrip traces tend to spread out within a planar structure, illustrated in Figure 4. This distribution will always exist in both the forward direction as well as the return path. The current distribution will share a common impedance between the trace and plane (or trace-to-trace) which results in mutual coupling due to the current spread. The peak current density lies directly beneath the trace and falls off sharply from each side of the trace into the ground plane structure. [2]

When the distance spacing is far apart between trace and plane, the loop area between the forward and return path increases. This return path increase raises the inductance of the circuit where inductance is proportional to loop area. Equation (3) describes the current distribution which is optimum for minimizing total loop inductance for both the forward and return current path. The current that is described in Equation (3) also minimizes the total amount of energy stored in the magnetic field surrounding the signal trace.

$$I(d) = \frac{I_o}{pH} \cdot \frac{1}{1+\left(\frac{D}{H}\right)^2} \qquad (3)$$

where: $I(d)$ = signal current density (A/inch), Io = total current (A), H = height of the trace above the ground plane (inches), D = perpendicular distance from the center line of the trace (inches).

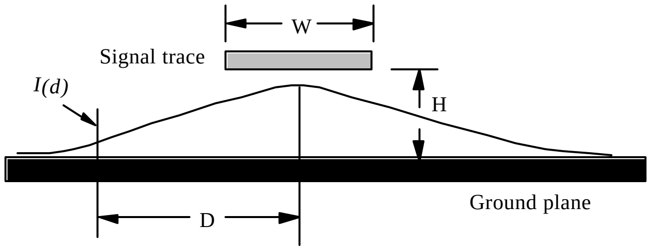


Figure 4. Current density distribution (trace to reference plane)

To achieve plots that are easy to view within this paper, 40 MHz harmonics were injected into the 18 inch (45.7mm) 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. By superimposing plots on top of each other (on a light table) it was easy to observe changes in radiated emissions between corner configurations. A baseline measurement was taken to determine the magnitude of the injected signal with only a 50 ohm terminated antenna; coax and resistor. This baseline measurement allowed the coax to simulate a small monopole antenna. This plot, Figure 5, was compared against all trace configurations for actual amplitude of radiated emissions. To determine the actual amplitude for data shown in Figures 6-9, subtract the magnitude of the baseline plot from the recorded data.

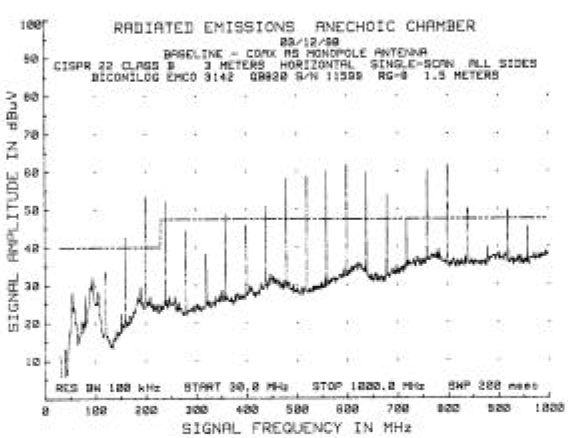


Figure 5. Baseline measurement of radiated emissions

For radiated emissions, a signal generator was set to –10 dBm with the 50 ohm coax connected between the launch pin of the PCB and the end of the termination resistor. Depending on how the board was strapped, various RF return path configurations were possible. It is to be 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 with. It is not possible to include all plots herein, hence one representatives sample is provided. The difference between plots were minor, as all plots were nearly identical when compared against other similar configurations. .Figure 6 illustrates the 0.005 inch trace with no RF return path.. This plot show a significant amount of radiated energy throughout the frequency spectrum, especially in the lower frequency range. The magnitude of these emissions are to be compared to the plot of Figure 5, also with a 0.005 inch wide trace to illustrate the effect an RF return path has on a transmission line.

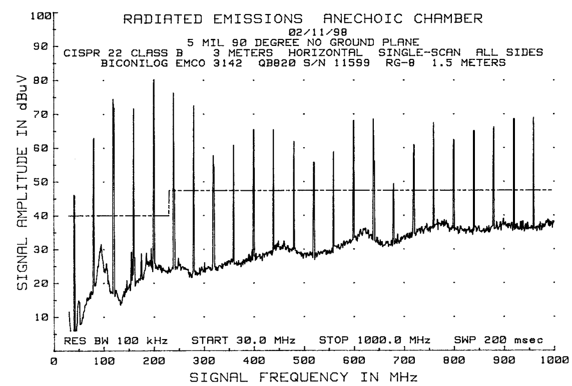


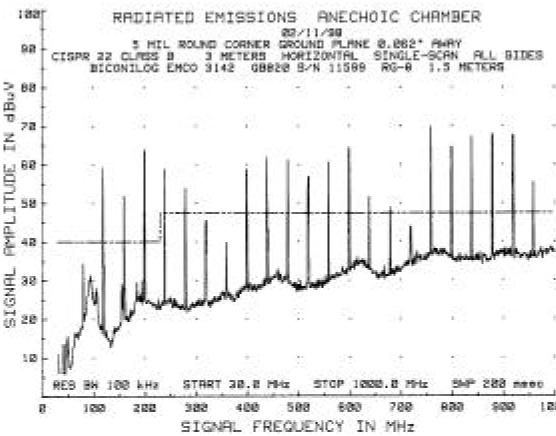
Figure 6. 5 mil trace with no RF return path

Figures 7, 8 and 9 are plots with various trace width and corner configurations. The 0.005 inch wide trace with 90 degree, 45 degree and round corners is shown along with the 0.010 inch and 0.020 inch wide trace. All traces are located over a ground plane 0.062 inches away (bottom layer of the PCB). Data with the guard band was almost identical to the solid ground plane with the exception of ± 2-4 dB for various discrete frequencies.

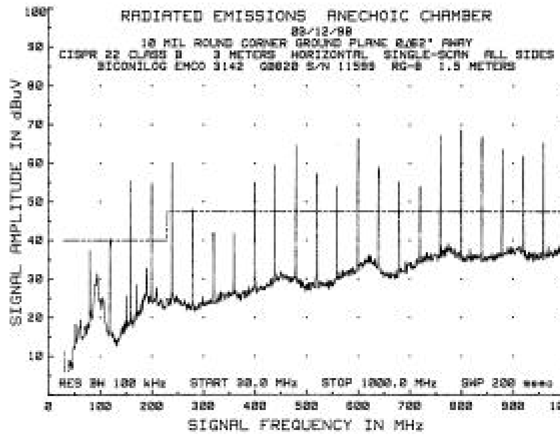
When comparing trace configurations for radiated emissions using board #1, mixed results were observed. Each trace width *must be evaluated* against the same width trace. In other word, it is not possible to compare the magnitude of emissions between three different trace widths using the same corner configuration. This is partially due to parameters related to instrumentation and test setup within the anechoic chamber. The purpose of this measurement is to determine if corners radiate above a reference point, not the absolute magnitude of the signal.

For almost all test configurations, the 45 degree corner radiated *more* than the 90 degree corner by 2-5 dB in the frequency range of 400-600 MHz. The 0.005 inch trace had significant radiated emissions greater than the 0.020 inch in the frequency range 40-300 MHz. The 0.010 inch trace has significant emissions between 100 and 300 MHz, most likely attributed to a change in the test setup and the position of the source cable within the chamber.

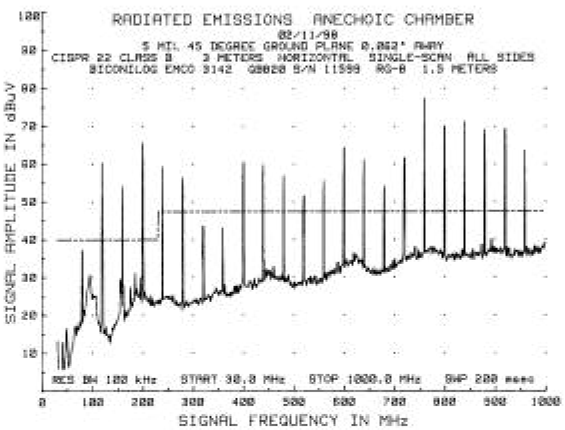
The radiated difference between trace widths can be attributed to the transmission line being an efficient radiator at $\lambda/4$ and $\lambda/8$ for the frequency range investigated in addition to increased trace impedance. Throughout the frequency spectrum, the amplitude of measured signals varied 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.



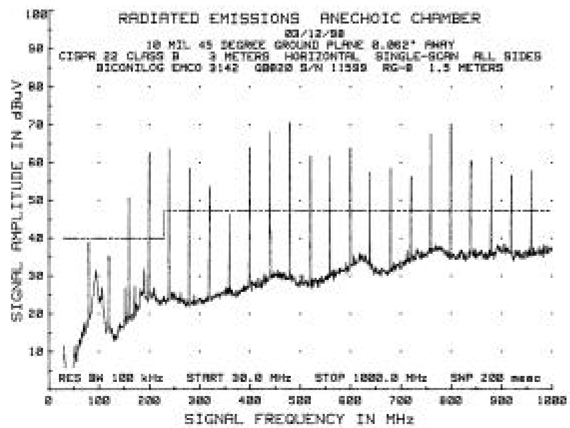
5 mil trace – round corner with ground plane



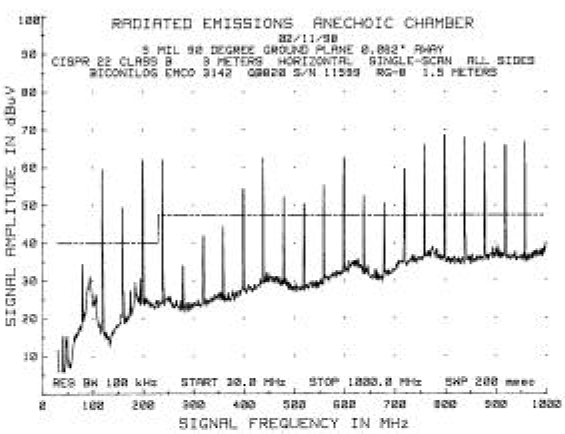
10 mil trace – round corner with ground plane



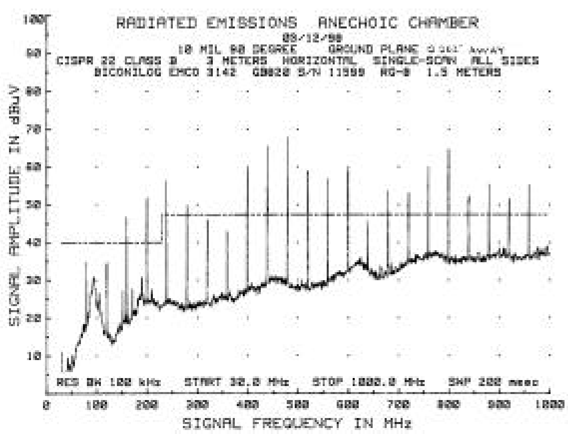
5 mil trace – 45 degree corner with ground plane



10 mil trace – 45 degree corner with ground plane



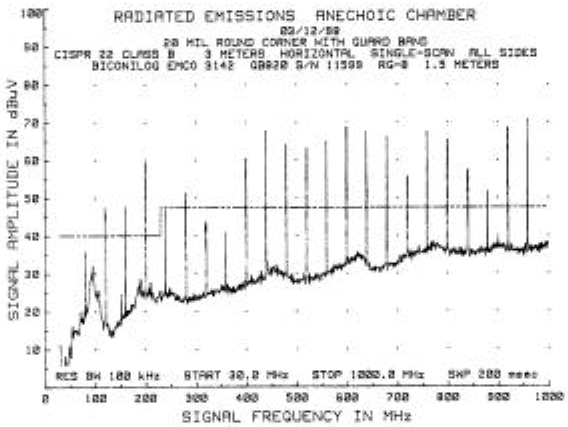
5 mil trace – 90 degree corner with ground plane



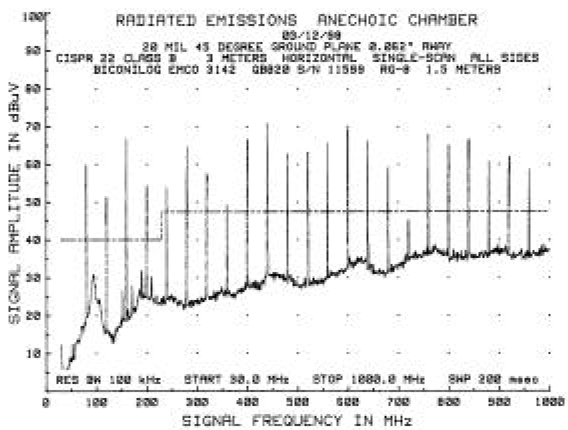
10 mil trace – 90 degree corner with ground plane

Figure 7. Radiated emissions from 0.005 inch trace

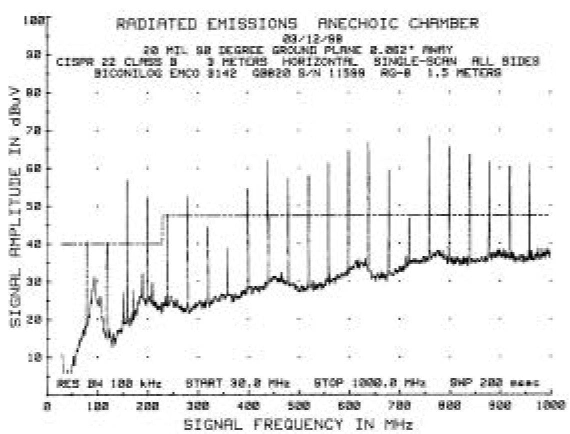
Figure 8. Radiated emissions from 0.010 inch trace



20 mil trace – round corner with ground plane



20 mil trace – 45 degree corner with ground plane



20 mil trace – 90 degree corner with ground plane

Figure 9. Radiated emissions from 0.020 inch trace

For board #2, trace 3 (90 degree corner) and trace 6 (45 degree corner), both radiated slightly higher than trace 2 (no corners). These plots are not shown herein. The magnitude of the emissions were 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 until approximately 750 MHz, and then at very low levels.

Conclusion

For years, assumptions were made stating right angle corners on a PCB will be harmful to signal integrity (time domain) or will radiate RF energy which may compromise EMI compliance (frequency domain). Actual measurements were made to validate these issues should design engineers have concerns with corner configurations during layout.

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: Radiated emissions exist, however, measurements up to 1 GHz does not show an increase for 90 or 45 degree 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, connectors, layer jumping, 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, and even then, the magnitude of the signal is minimal. It is difficult, if not impossible to measure radiated emissions from any trace corner.

Acknowledgment

Thanks to HADCO Corporation for quick-turn manufacturing of board #1 with engineering support and Douglas Brooks of UltraCAD Design for providing the design and a test sample of board #2 [6]. Thanks to Ron Pratt of Hewlett Packard who performed network analyzer measurements used for the time domain analysis). Atlas Compliance & Engineering, Inc., (Mario Baraona and Bruce Smith) provided use of their anechoic chamber for radiated emissions (frequency domain analysis).

References

1. Montrose, M.I. 1996. *Printed Circuit Board Design Techniques for EMC Compliance*. NJ:IEEE Press.
2. Montrose, M.I. 1998. *EMC and the PCB – Design, Theory and Layout Made Simple*. NJ:IEEE Press.
3. Montrose, M.I. 1996. "Analysis on the Effectiveness of Image Planes within a Printed Circuit Board. " Proceedings of the IEEE International Symposium on EMC. NJ:IEEE.
4. IPC-D-2141. *Controlled Impedance Circuit Boards and High Speed Logic Design*, April, 1996. Institute for Interconnecting and Packaging Electronic Circuits.
5. Edwards. T.C. 1983. Foundations for Microstrip Circuit Design. NY:John Wiley and Sons.
6. Brooks, Douglas. "90° Corners, the Final Turn." Printed Circuit Design Magazine, 1998. Vol. 15, #1.