Radiated Emission Far-Field Propagation with Multiple Ground Stitch Locations Within a Printed Circuit Board

Mark I. Montrose
Montrose Compliance Services, Inc.
Santa Clara, CA USA
m.montrose@ieee.org

Abstract— **Printed circuit boards (PCB) exhibit numerous properties related to the development of common-mode currents created by digital components. When multiple devices are switching simultaneously, a complex RF field will propagate into free space from package bond wires, the silicon die itself, via implementation, poor transmission line routing, the physical board edge, interconnects, and numerous other areas generally not recognized by the designer. The majority of electromagnetic interference (EMI) is associated with a poor power distribution network (PDN). Do we really know where the EMI is coming from with a fully populated assembly and what are the effects of increasing the number of ground stitch locations to chassis ground to disrupt propagating RF?**

Measurement of propagated field amplitude is easy related to regulatory compliance standards. Within a complex PCB, one generally does not know the source of the EMI based on only simulation, since many second and third order parasitics are unknown to the designer. This paper investigates overall effects of disrupting the dipole antenna structure of the power distribution network in the far field using numerous source stimuli versus a simple single source stimulus along with simple transmission line routing to a simple end termination.

Index Terms — Radiating field pattern, multiple source stimulation, propagating field, field reception, far field measurement, multiple vias, ground stitching.

I. Introduction

Both power and return planes are in reality transmission lines and must be implemented within a PCB using the same design and routing requirements as signal lines. Both must be terminated. If poor or improper termination exists, wave reflections will occur that can then be observed as a propagating RF field, be it the edges of the board reflected back to the silicon package and radiated from internal bond wires, or interconnects consisting of various configurations [1].

For efficient propagation of an RF field, an antenna is required, either dipole or loop. When analyzing which antenna structure applies to power and return planes, assuming return is at 0V potential and power is bouncing, an efficient dipole antenna is created.

Within this paper, we examine only the dipole model. For an antenna to be an efficient radiator, the physical length of both the driven and return element must be at a particular wavelength, or permutation of a wavelength based on both frequency and harmonics of a propagating signal.

Between every power and return pin on digital components and their relative location to the edge of the PCB, a physical dimension exists. This physical dimension creates a dipole antenna.

There are generally hundreds of power and return pins along with many digital components in a real-world PCB. Depending on the physical location of these pins within the device package, and the magnitude of current consumption during an edge switching event, a propagating wave will be created with each signal transition (source or sinking current). This propagating wave travels away from its respective pin radially. Depending on the frequency and wavelength of this signal, relative to physical dimensions on the PCB, the dipole configuration within the PDN will be energized. An RF propagating field at 3 meters (far-field) is examined based on the grounding points provided. Grounding disrupts the efficiency of the internal dipole antenna present within the PDN planes.

Past simulations investigated RF field propagation in the *near-field* whereas this research examines RF propagation in the *far-field*. Analyzing near-field RF EMI and relating these to the far-field, when applied toward compliance standards, provides little value to engineers.

II. Simulation Model

The configuration of the PCB is shown in Fig. 1. First, we examine propagating field plots in the far field with 100 stimulus sources all in phase. Each stimulus creates a propagating wave on the power plane which phase add/subtracts with all other propagating stimulus caused by transmission line reflections. These reflections occur from other power and return pins in the plane and

978-1-4244-5623-9/10/$26.00 ©2010 IEEE

the physical edge of the PCB. The physical distance between two points is the driven element of a dipole antenna structure and will radiate RF energy based on the self-resonant frequency of the antenna.

At frequencies in the GHz range, planes will radiate nicely. At lower frequencies, typically below 1 GHz, phasing effects are minimal with regard to EMI since RF wavelengths are larger in physical size.

After baseline field plots are determined, we then incorporate ground stitch locations on the 0V plane to a single point reference, or chassis ground, to disrupt the efficiency of the dipole antenna's return leg, or ground element, thus making the planes an inefficient radiator at higher frequencies. In a typical PCB, the return plane is generally bonded to a metal chassis, thus the analysis herein represents a real-board with many power and return pins spread throughout the assembly.

PCB Characteristics

- Dimensions: Lx=30.48 cm (12 inches)
  Ly=10.16 cm (4 inches)
- Distance between power and return plane:
  h=0.127/0.254/0.508 mm (5/10/20 mils)
- Dielectric constant: $\varepsilon_r = 3.3$
- Loss tangent: $\delta = 0.002$
- Copper thickness: 0.7 mils = $17.5e^{-3}$ mm (0.5 oz)

Power Plane Configuration (Fig. 1)

- Total number of stimulus points: 100 located randomly with regard to x- and y-axis.

Return (Ground) Plane Configuration (Fig. 2)

- Number of ground locations: 70 (14 x 5 matrix)
- Spacing between ground locations: 2 cm
- Radius of stitch: 1 mm (implemented as vias).

Stimulation Characteristics

- Stimulation source: 1 mA/via
- Stimulation frequencies: 1.8, 2.45, 3.0, 5.0 GHz
- Simulation software: HFSS

It is well documented that a PCB radiates a propagating field based on any number of stimulation sources. Examples of stimulation sources include lead bond wires internal to component packages, improper implementation of decoupling capacitors that in turn allows a poor power distribution network to be present, board edge radiated emissions, transmission line routing with impedance discontinuities that results in a signal integrity problem (which in turn develops common-mode currents), ground loops between circuits that are a significant wavelength of existing RF energy present, along with many other items not detailed herein.

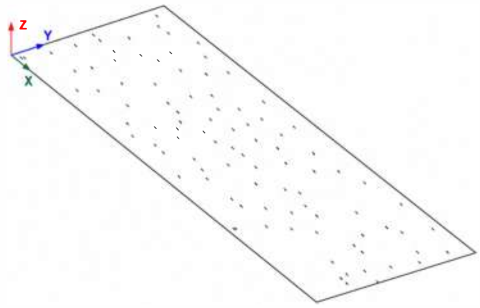

Fig. 1 Power plane with 100 randomly located via stimulation sources

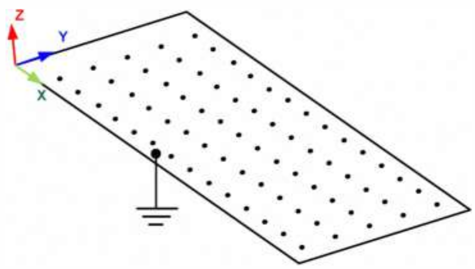

Fig. 2 Ground plane with 70 ground stitch locations

III. Discussion

Two primary causes of unintentional EMI are the result of: (1) a poor power distribution network (PDN) and, (2) ground loops containing RF spectral currents propagating between components and interconnects. This paper investigates only the effects of implementing an optimal grounding methodology in the return plane to minimize the magnitude of a propagated RF field at 3 meter distance from the PCB, not a poor PDN or lack of an optimal decoupling methodology.

The most efficient radiating antenna within a PCB is a dipole. Dipoles radiate an RF field at a significant wavelength of a specific frequency or physical dimension between circuit elements. If a component sources or sinks current from the power/return planes, relative to the physical edge of the PCB, the stub between component pins and the edge of the board emulates a dipole antenna. If we short out one leg, or element of the dipole, RF propagation is thus minimized. This stub (dipole) antenna is illustrated in Fig. 3 [1].

If the return plane is at 0V potential with regard to chassis ground, incorporating ground stitches anywhere in the return leg of the dipole antenna will cause this efficient radiating element to become inefficient. The more stitches, the smaller the physical dimension of the ground element of the dipole antenna. It does not take many ground stitches to disrupt the efficiency of a dipole operating at higher frequencies, generally in the GHz range for high-technology products.

IV. ANALYSIS

Comparing simulated data from multiple stimuli all in phase and operating simultaneously both without ground stitches (Fig. 4) and with (Fig. 5), interesting field propagating patterns will exist at 3 meters. If we have an antenna in the far field for signals in the GHz range, can we be confident that the propagating wave being measured is in reality, the actual maximum RF field intensity if the magnitude of the propagating wave falls outside the beamwidth of the transducer, which is usually a horn antenna when measuring GHz signals?

The radiated field plots are significantly different, which is expected. The fields disrupted by ground stitches on the PCB board result in unique radiation patterns. With many ground stitches on a PCB, or multiple bonding of the return plane to a single point reference (i.e., 70 in this paper), it is impossible to manufacture a PCB and measure it in an anechoic chamber. Also, how does one physically, or visually, observe actual RF field patterns as measured by an antenna? HFSS is known to provide accurate field propagation results.

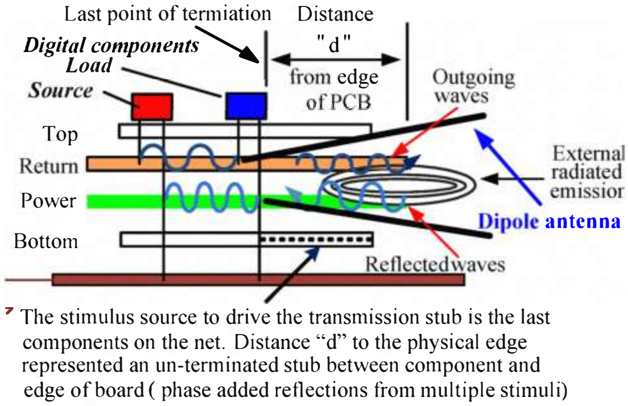

Fig. 3 Reflected RF waves internal to a PCB between power/return pins and to the physical edge of the board

The test configuration is a realistic application of a high-technology PCB with numerous stimulus sources, thus we can assume Figs. 4 and 5 are accurate representations of the RF propagating field at 3 meters.

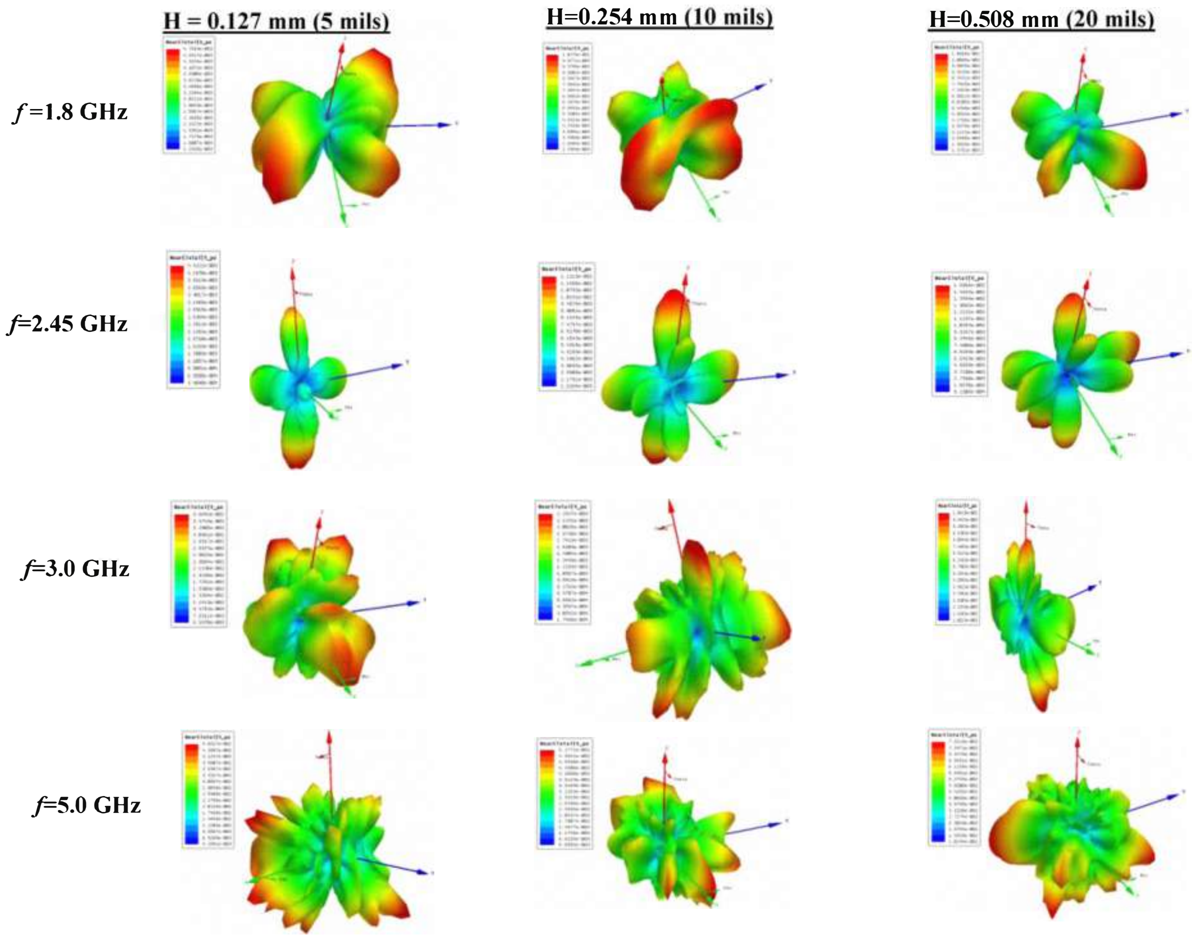

Fig. 4 Plots of the radiated field at 3-meter distance parallel to the PCB

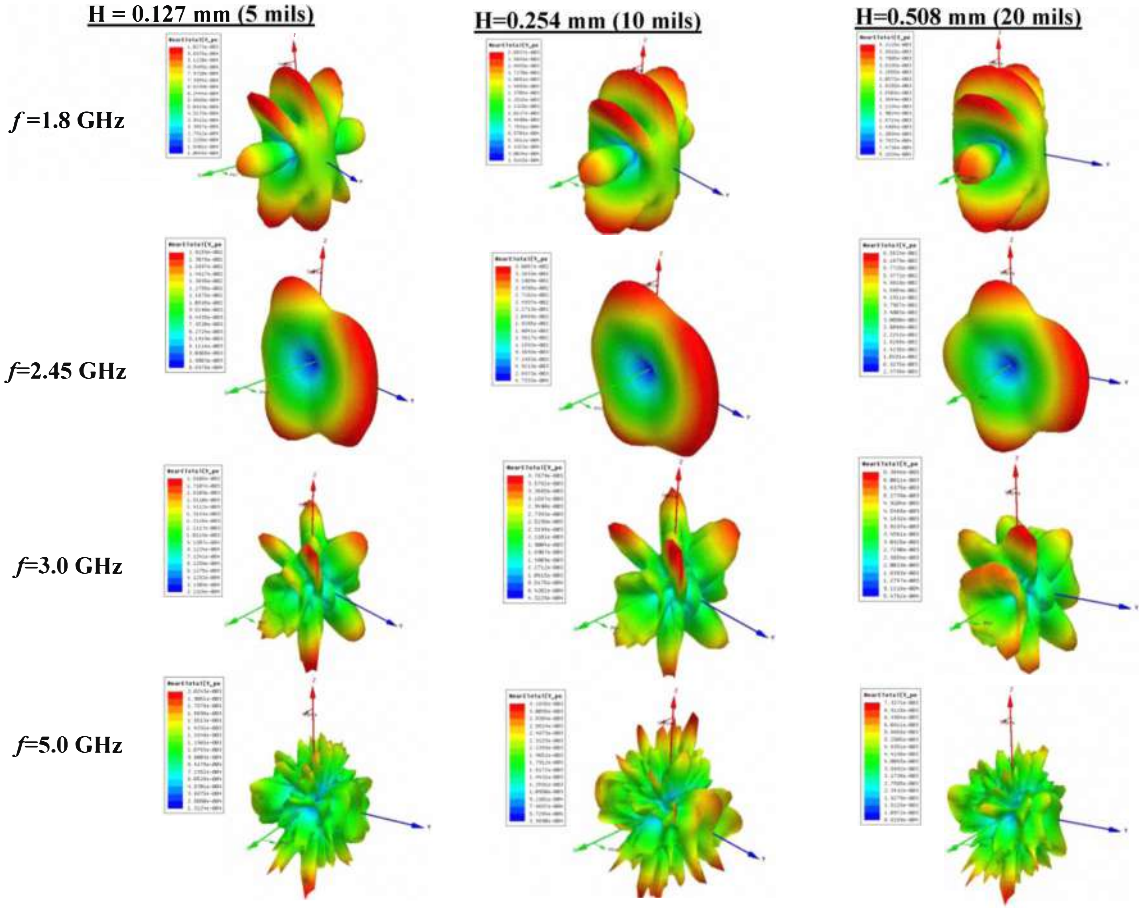

Fig. 5 Plots of the radiated field at 3-meter distance parallel to the PCB with ground stitches

V. CONCLUSION

All printed circuit boards contain numerous RF field stimulation sources, namely the power and return pins on digital component or signal traces traveling through a via.

With each edge transition sourcing or sinking current, either the power or return planes will have a finite amount of SSN injected into the planes if a poor power distribution network is present, or lack of optimal decoupling. Between the power/return pins and the physical edge of the PCB, a dipole antenna exists with wave reflections on the planes. Propagating waves either phase add or subtract which establish an RF propagating field. Planes must be treated as transmission lines and terminated.

With ground stitches, the field pattern of the propagating field will be significantly altered when measured in the far field. Depending on the physical dimensions of the PCB, we easily notice that the planes are efficient radiators at a particular frequency, and the effects of providing ground stitching significantly changes the radiation pattern. What we observe in the far-field are many dipole antenna on the PCB radiating simultaneously to create a complex propagating field.

To make the dipole antennas present on the power and return planes, by virtue of having digital components on the PCB, we must convert the antenna to an inefficient radiator by shorting out one of its two elements, in this case, the ground element to chassis ground.

Higher frequency propagating fields have smaller wavelengths that appear differently to an antenna. If only one stimulus source was analyzed, we would probably see only a single planar wave instead of real-life complex field patterns.

REFERENCES

[1] Montrose, M. I., LIU En-Xiao. "Radiation emission effects from multiple via stimulation within a printed circuit board", 2008. *Asia-Pacific Symposium on EMC & 19th International Zurich Symposium on EMC,* pp. 176-179.

[2] Mishra, S.R.; Kashyap, S. "Structure of EM field at an open field site". *1988 IEEE International Symposium on EMC*, pp 94-98.

[3] Thomas, D.W.P., et. al. "Model of the electromagnetic fields inside a cuboidal enclosure populated with conducting planes or printed circuit boards," *IEEE Transactions on EMC*, 2001, pp 161-169.