Analysis on the Effectiveness of the 20-H Rule Using Numerical Simulation Technique

Er-Ping Li, Wei-Liang Yuan
EMC Division
Institute of High Performance Computing
1 Science Park Road, Science Park II
#01-01 , Singapore 117528
eplee@ihpc.nus.edu.sg

Jiang Yi[+], Li Le-Wei[++]
+Temasek Lab
++Department of Electrical
and Computer Engineering
National University of Singapore
Singapore 119260

Mark I. Montrose
Montrose Compliance Services, Inc.
2353 Mission Glen Dr.
Santa Clara, CA 95051-1214
mmontros@ix.netcom.com

Abstract

With increasing demand for higher operating frequencies, printed circuit board (PCB) designers are faced with numerous obstacles during the design cycle. The 20-H rule is one layout technique that is recommended to minimize radiated fields propagating from the edges of a PCB. These fields, either magnetic or electric, may corrupt adjacent cable assemblies, sheet metal enclosures and aperture openings. The magnitude of this design rule is investigated herein using the numerical electromagnetic simulation method, Finite Difference Time Domain (FDTD). In addition, an analysis on whether benefits exist from use of the 20-H rule is examined.

The significance to the field of EMC lies in presenting, for the first time, an exhaustive analysis that took over one year to achieve using various configurations and frequencies. A rigorous analysis indicates that benefits do occur from use of the 20-H rule, but only when implemented in certain applications.

Keywords

Electromagnetic radiation, PCB, 20-H rule, FDTD

INTRODUCTION

The 20-H rule states that the physical size of a power plane in a high density, multi-layer stackup topology must be smaller than its corresponding return plane by a physical dimension equal to 20 times the distance separation between the two planes. The application of this rule-of-thumb, and when it is or is not appropriate has not been well defined due to the complexity of the design concept, theory involved and understanding how to setup a proper model that truly describes operation of the rule. For historical reasons, the 20-H rule was discovered and implemented over 20 years ago by several companies, working in conjunction with each other, to solve specific problems observed with PCB design and layout. This design technique, which was proprietary for years, was first released into the public domain in 1996 [1].

The 20-H rule provides PCB edge termination of propagating waves between digital components that switch large amounts of "peak power current," generally faster than 1 ns.

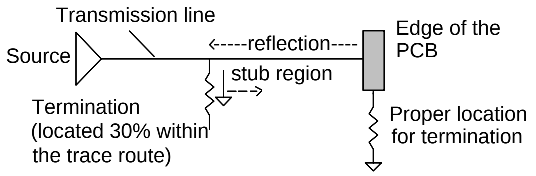
Multiple propagating RF waves, or current surges, are present within a power and return distribution network when *more than one* digital device transitions logic states simultaneously. A bounce condition occurs in the distribution planes if improper decoupling exists. Bouncing the power and return planes causes a propagating field, at RF frequencies, to be developed with the switching wave front. This signal travels in a radial manner from the source location to the edge of the PCB, all at different points of time. When multiple propagating waves reach maximum amplitude (phase addition at the same location in time and space), significant radiated RF energy can occur off the edge of a PCB in various locations (not 360 degrees on all four edges. The 20-H rule terminates this undesired propagating field.

Power and return planes must be treated as transmission lines with wave propagation, reflections and ringing. A propagating wave will return to its source after encountering a high-impedance load (the edge of the board). Basically, a signal integrity situation exists, except this time it involves propagating fields in the dielectric that separates both the power and return planes, similar in nature to a typical transmission line carrying a digital logic signal.

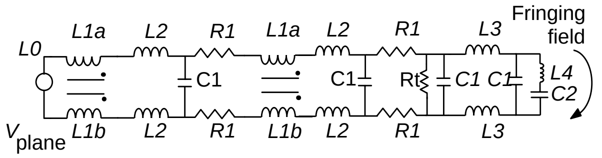
In order to ascertain the magnitude of effectiveness for the 20-H rule, multiple PCB structures were examined. These structures include different size boards, each with a distance spacing of 0-H, 10-H and 20-H. Past research on overly simplified models did not illustrate with any degree of accuracy benefits that may occur. In addition, research focused on only one test frequency and one physical board dimension. A single frequency stimulus that exactly matches the inherent board resonance (creation of a microwave patch antenna, which also happens to be an efficient radiator of RF energy at the stimulus frequency), does not accurately reflect or describe behavior of the 20-H rule. In addition, benefit from use of 20-H rule was not analyzed.

FUNDAMENTAL CONCEPT OF THE 20-H RULE

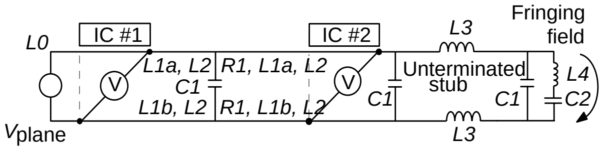
The location of terminators in any transmission line is very important. Figure 1 illustrates both the electrical and physical representation of a power and return plane as they relate to a transmission line equivalent circuit.



Transmission line representation of power and return planes



Power and return plane equivalent schematic



Power and ground plane mechanical equivalent representation

*L*1a and *L*1b = mutual inductance termination
*L*2 = line inductance intrinsic and resudual after flux cancellation
*L*3 = uncanceled inductance of the stub-no controlled image flux
*C*1 = line capacitance
Rt = partial termination, usually 10 times the impedance of the line
*L*4 and C 2 = possible fringing field at unterminated end of the planes

Figure 1. Transmission line equivalent circuits of planes [1]

A PCB with a signal injected into the center of the board will have a propagating wave travel to the edge of the board radially (360 degrees), viewed with the planes forming a transmission line in the vertical, or z-axis (the dielectric).

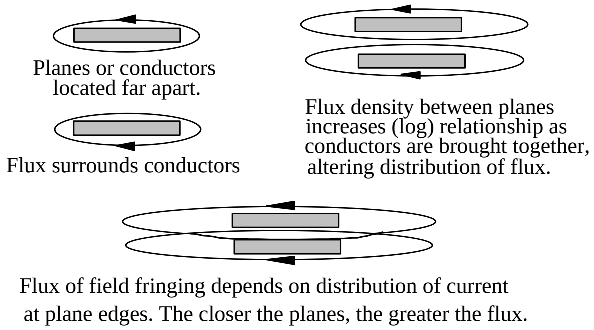
Assume a source signal is now situated at the center of the PCB. Physically under the stimulus, or digital circuit, is a small loop area between the return and power pins where switching current is circulating between the two pins. Planes continue far beyond the localized area of the circuit. With this condition, planes become one large planar stub that causes reflections and resonances throughout the board. These resonances usually do not cancel because of the phase angle skew between the planes. This skew is caused by the spreading inductive characteristics of the planes.

The intended goal of the 20-H rule is to terminate planes locally toward the circulating current boundaries of the flux and RF fields formed by device–power–currents located at the edges of the boards, *not to set an arbitrary dimension*. Even if one were to resistively terminate the "flying" (unterminated) planes, the mechanical geometry of the resistors (which approximate a “1206” sized device) would require undercutting the planes about 20-H or around 0.055 in. (1.4 mm) planar height.

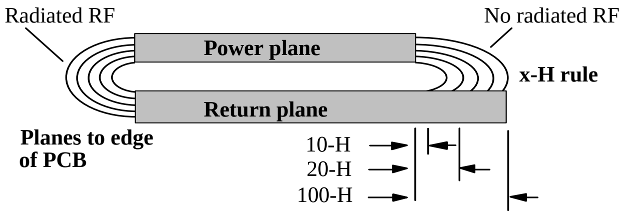
The component technology at the time of first implementation only provided for an undercut distance of the power plane by approximately 100 mils, which is 20 times the distance separate of 5 mils between plane. The phrase “20-H rule” was born, without consideration of the relevance of the application or implementation [2].

The propagational model as an antenna structure for planes can take several forms ranging from a slot or patch antenna formation at various frequencies, to a dipole that is resonance at other frequencies. The formation of the antenna model depends primarily on the configuration of the power current propagational circulation; where and how the planes are terminated.

Use of the 20-H rule increases the intrinsic self-resonant frequency of the PCB, because the physical dimensions of the power distribution network are altered. Since less capacitance is present, there will be a higher self-resonant frequency of operation. This impedance change in the power distribution threshold is first noticed at approximately 10-H, with 20-H representing the approximately the 70% flux boundary. The dimension "H" is the physical distance spacing between the planes within the stackup assignment. Flux boundary refers to the distance that magnetic flux is observed from the planar structure in the near field. To achieve a 98% flux boundary, 100-H is used.



Simplified illustration on how flux surrounds conductors



At 10-H, impedance change of the planes is first observed.
At 20-H, we reach the 70% flux boundary.
At 100-H, we approach th 98% flux boundary.

Closeup of flux coupling between planes once distance spacing becomes very small. Greater distance spacing between planes do not benefit significantly from an undercut power plane.

Figure 2. RF fringing effects from planes [1]

NUMERICAL SIMULATION

In order to investigate the 20-H rule, extensive computing power is required to ascertain the magnitude of emissions over the entire frequency spectrum, not just one stimulus frequency. Use of multiple stimulation frequencies and distance spacing between planes, along with different configurations from 0-H, 10-H and 20-H are examined. From here, we visualize what really occurs, and can make a judgment decision if the 20-H rule is valid.

Special algorithms had to be developed and the FDTD code rewritten to define the problem of propagating waves (frequency domain) in conjunction with transmission line termination (time domain). This modified code is now proprietary and was validated to be accurate with other known propagating field structures and configurations. This custom code had to be executed on high performance mainframe computers, as desktop workstations are unable to handle the amount of computing power and memory required to achieve high levels of accuracy during simulation. Due to the nature of the problem definition, as well as determining the magnitude of the propagating fields off the edge of the board, which are located in the near field, measurement on an actual PCB in an anechoic chamber in the far field would not provide benefit for correlation purposes.

SYSTEM MODEL IMPLEMENTATION

In order to simulate the 20-H rule accurately, a model had to be developed that took into consideration both electric "*E*" and magnetic "*H*" fields present at the same point in time and space. A 2-dimensional model was selected to simulate signal propagation. Figure 1 illustrates the planar structure used for this investigation.

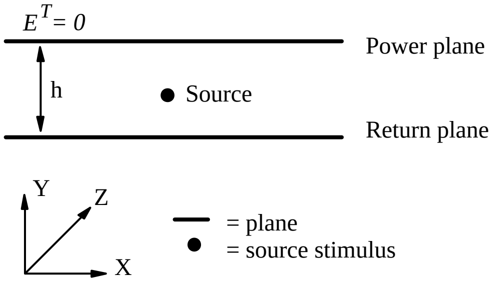


Figure 3. Planar structure model

The following assumptions are made:

1. Both the power and return plane are perfectly conducting and satisfy these conditions in the positions of the two planes.
2. Both planes are infinitely thin in thickness (compared to width and length).
3. Simulation is based on free space.

Due to limitations during simulation, free space was truncated into finite rectangles in which the edges of each rectangle include reasonable perfectly match layers (PML).

Three types of sources were used for simulation– dipole, uniform voltage and Gaussian.

The dipole excitation source is implemented at a single point in the center of the structure, described by Eq (1) and shown in Figure 3.

$$E_{center} = E_o \sin(2\pi k f dt) \ V/m \qquad (1)$$

where: f = frequency of the source, k = time step number after discretization and $dt = \Delta x / 2c_o$ (c_o = speed of light).

The uniform voltage source is illustrated in Figure 4.

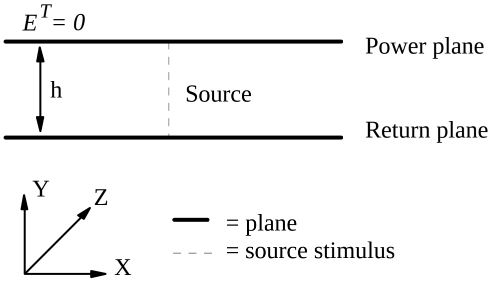


Figure 4. Uniform voltage source

Gaussian impulse is defined by Eq (2).

$$E_{center} = E_o e^{1/2 (t_o k)^2 / a} \qquad (2)$$

where k=time step, and a = spreading function in the time domain.

SIMULTION RESULTS

Electromagnetic distribution

Simulated analysis detailed herein represents the electromagnetic field distribution for both *E* and *H* fields before and after implementing the 20-H design rule. Simulation is based on the physical length of the power plane at 10cm (3.94 inches), with separation between the power and return plane at 0.25mm (0.01 inches or 10 mils). A sinusoidal excitation source of 300 and 600 MHz was applied. Only data from 300 MHz is presented for brevity. Results obtained for 600 MHz had the same properties and nearly identical results thus providing correlated data.

Figures 5 and 6 illustrate the electromagnetic field emitted from the edge of the PCB. Figure 5 shows the *electric field component* (E_x), for 0-H, 10-H and 20-H in the x-axis. Figures 6 shows the *magnetic field component* (H_z) for 0-H, 10-H and 20-H in the z-axis. Size of the power plane was constant, only the ground plane was extended.

It is clearly illustrated that the radiated electric field (*E*) and magnetic field (*H*) after extending the return plane beyond the power plane has absorbed flux propagation from on the power plane by terminating the flux field over a continuous return path (detailed in Figure 2).

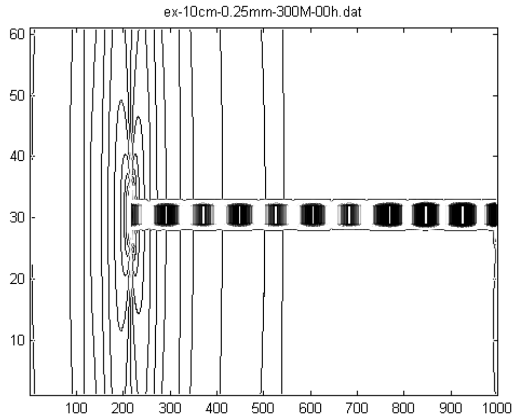


Figure 5(a). ***Ex*** **distribution 0-H at 300 MHz**

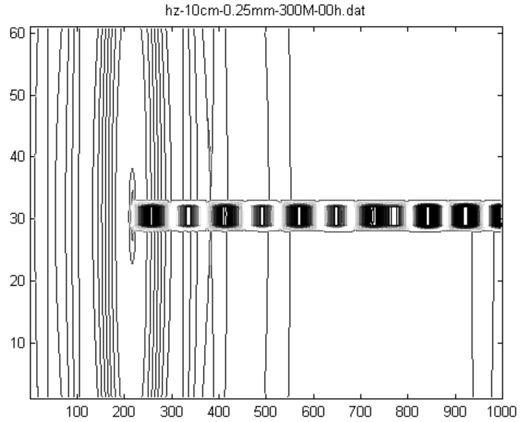


Figure 6(a). ***Hz*** **distribution 0-H at 300 MHz**

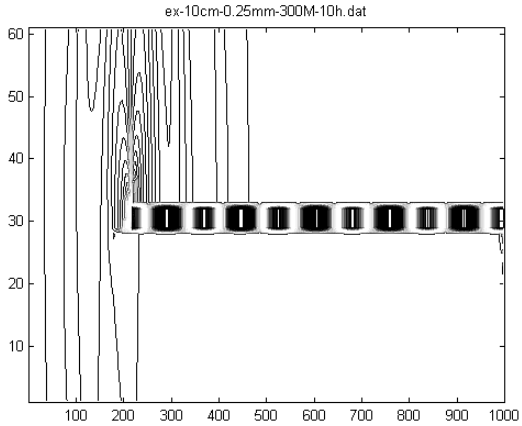


Figure 5(b). ***Ex*** **Distribution with 10H at 300 MHz**

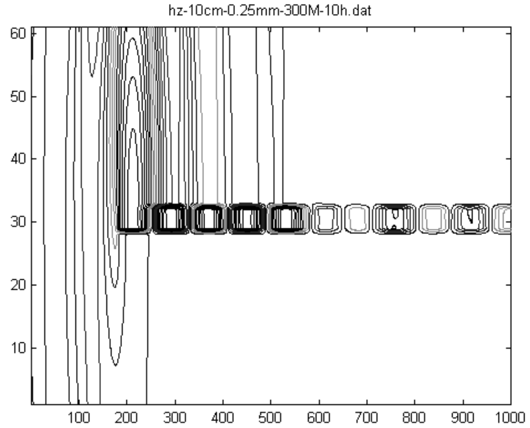


Figure 6(b). ***Hz*** **Distribution with 10H at 300 MHz**

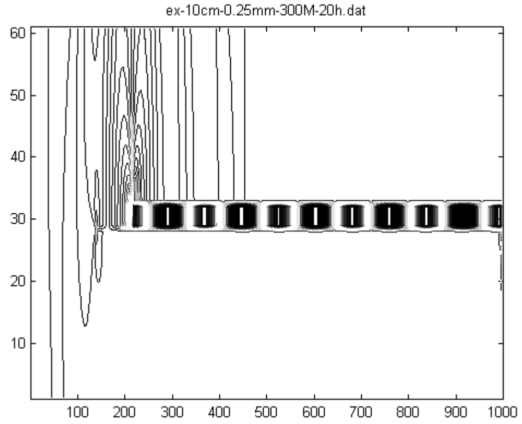


Figure 5(c). ***Ex*** **Distribution with 20-H at 300 MHz**

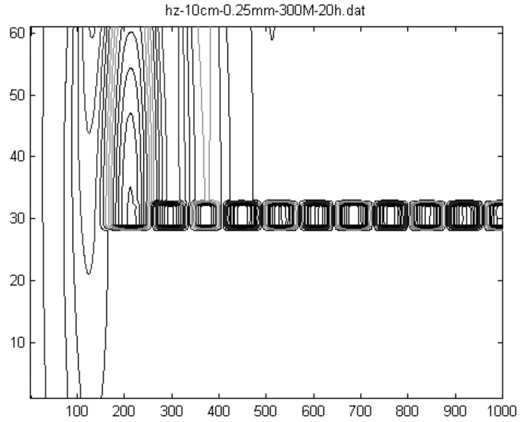


Figure 6(c). ***Hz*** **Distribution with 20-H at 300 MHz**

Terminating flux at the edge of the PCB with an RF return plane results in significantly less RF energy that may radiate to adjacent circuits, metallic enclosure or card guides and cable assemblies.

This simulated analysis validates the effectiveness of the 20-H design rule for this particular PCB configuration– length of the power plane, extent of the extended return plane beyond the power plane, distance separation between planes, and frequency of operation.

However, it is not applicable to every PCB configuration. The effectiveness is quantified in the following sections and when it should not be used.

A key element to remember is not the shape or configuration of the flux from the extended plane, but the *"magnitude"* of the radiated power off the edges of the PCB that may cause harmful interference.

Radiated energy from the edges of the PCB

To access the effectiveness of the 20-H design rule, energy emitted from the PCB edges needs to be quantified. The Poynting vector is selected to represent radiated energy from the edges. The Poynting vector is defined as electromagnetic energy density, expressed in Eq (3).

$$p = E \times H \qquad (3)$$

Hence, total energy radiated from the PCB edges can be expressed as the integration of the energy density P:

$$P = \oint_s E \times H \quad ds \qquad (4)$$

Where E and H is denoted as the electric and magnetic field strength respectively. For instance, the emitted energy from a 2-D structure is the integration of the power density along the edges AB, BC, CD and DA as shown in Figure 7.

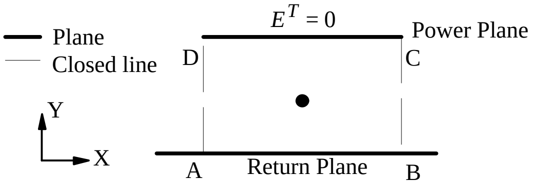


Figure 7. PCB structure for calculation of Poynting vector

Simulation was performed to access effectiveness of various extended plane configurations and source signal frequencies. A dipole voltage stimulus sits at the center of the two planes, defined as $E=1000 \, Sin \, (2 \pi \, f \, k \, dt) \, V/m$, where k= time step in FDTD algorithm, $dt=\Delta t=1.667 \times 10^{-13}$ (s), and f = source frequency (600 MHz). Results are detailed in Table 1 and plotted in Figure 8 for all configurations. Physical dimensions of the PCB had the power plane at 2cm (0.78 inches) long with a separation distance of 0.2mm (0.008 inches or 8 mils).

Table 1. Radiated power vs. return plane (600 MHz)
(*Power plane length* = 2cm, *d* = 0.2mm)

Configuration	Radiated power (P^{-10} watts)
0-H	0.5531
10-H	0.4002
20-H	0.0565

It is obvious from the results in Table 1 that radiated energy is drastically reduced when 20-H is applied for this specific PCB structure. The resonance of the PCB when the extended reference plane *is almost negligible* when compared to the radiated power without any extended return plane. This analysis does not conclude that the 20H design rule is effective for every PCB layout. The following further examines the design rule's applicability, and when this design technique fails.

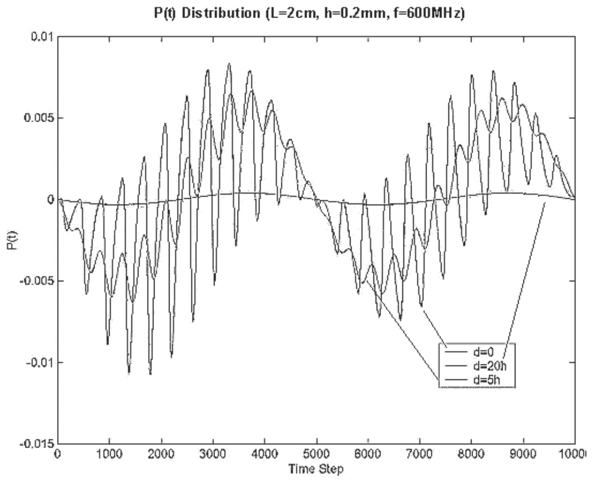


Figure 8. Poynting vector at 600 MHz

Additional simulations were conducted to examine radiated energy with different source signal frequencies (300 MHz, 600 MHz and 900 MHz). *The distance spacing between planes is doubled to 0.4mm.* For each frequency, five configuration of the extended return plane was investigated: 0-H, 5-H, 10-H, 15-H and 20-H. Simulation results are tabulated in Table 2. The respective Poynting vector plots are provided in Figures 9(a), (b) and (c).

Table 2. Radiated energy vs. source frequency
(*Power plane length* = 2cm, *d* = 0.4mm)

Distance of extended reference plane to create	Radiated power (P^{-10} watts)		
	300MHz	600MHz	900MHz
0-H	1.152	1.4000	1.035
5-H	0.692	0.6073	1.023
10-H	0.603	0.6079	1.017
15-H	0.603	0.6079	1.021
20-H	0.602	0.6077	1.023

Table 3 illustrates effects when the *physical length of the power plane is increased* to 4cm (1.6 inches), *with the same separation distance* between the planes at 0.4mm (0.016 inches or 16 mils). Changing the physical size of the power plane examines what happens to the PCB's self-resonance frequency if the board's physical dimensions are altered. If a resonant condition exists, and the stimulus frequency is at any possible combination of wavelength or harmonics, an increase in radiated power should occur.

It is observed in Table 3 that the 20-H rule is not valid when the PCB size is altered to match that of a stimulus frequency. The extended return plane with a larger size power plane did not absorb magnetic flux and electric fields that exist. Rather, more radiated energy is present. It is noted that the physical dimensions of the PCB plays a *significant role* in the effectiveness of the 20-H rule (or any variation: i.e., 5-H, 10-H) when stimulated at a resonant frequency of the power and return planes. The 20-H rule works *perfectly* for board dimensions that do not have any relationship to a stimulus frequency resonant with the planes. For brevity, plots are not provided for this table.

Table 3. Radiation Energy For Different Extending Distance (*Power plane length* = 4cm, *d* = 0.4mm, *f*=300MHz)

Distance of extended reference plane to create	Absolute power (P^{-10} watts)
0-H	1.2
5-H	2.1
10-H	5.7
15-H	23.0
20-H	121.6

CONCLUSION

The significant aspect of this research identifies field structures that radiate from the edges of a PCB when propagating waves are reflected back to the stimulus source after reaching a high-impedance termination (free space). Use of a smaller power plane, or having an extended return plane, significantly reduces radiated electromagnetic fields from the edge of the PCB. This rule-of-thumb is *most effective when the planes are closely spaced together*. The further apart the physical distance between planes, as in a typical 4-layer PCB stackup, benefit from use of 20-H (or any variation) may not be achieved.

The Poynting vector emitted from the board edge is the primary element of concern, not the radiation pattern of flux. If the fields that propagate off the edge of a PCB are small in magnitude, and does not cause harm by coupling undesired energy to other elements, should a PCB design engineer be overly concerned about implementing this design and layout technique? Again, physical size and frequency of operation determines if the 20-H rule (or variation) is required, including stackup dimensions.

ACKNOWLEDGMENT

The Institute of High Performance Computing, Singapore, is acknowledged for their computing resources and personnel in performing research on the 20-H rule.

REFERENCES

[1] Montrose, M. 1996-1st ed., 2000-2nd ed. *Printed Circuit Board Design Techniques for EMC Compliance – A Handbook for Designers,* New York: IEEE Press.

[2] The concept of the 20-H rule was first modeled and implemented by W. Michael King, who also came up with the term *20-H rule,* circa 1980.

[3] Yi, Jiang. 2001. “Printed Circuit Board Design Rule Analysis and Simulation by FDTD Method.” Master’s degree thesis–National University, Singapore.

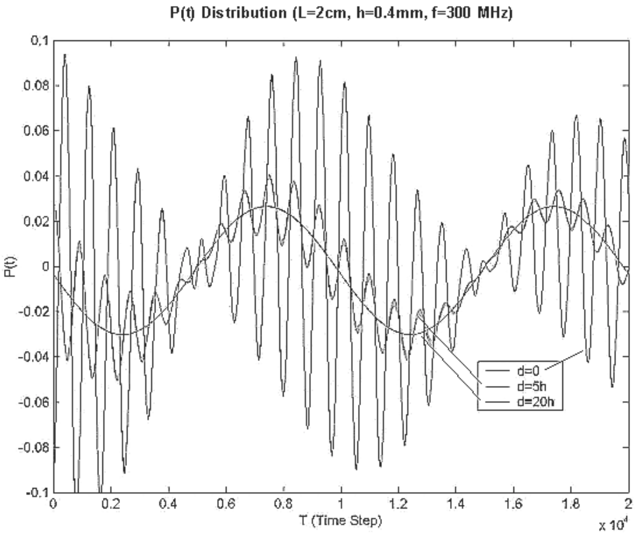


Figure 9a. Poynting vector at source frequency of 300MHz

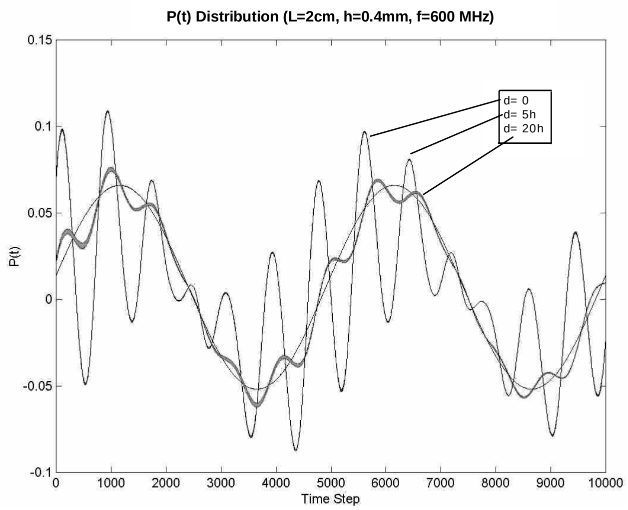


Figure 9b. Poynting vector at source frequency of 600MHz

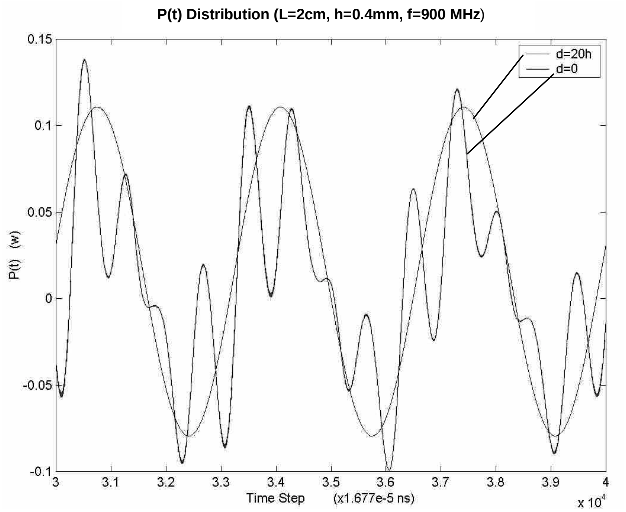


Figure 9c. Poynting vector at source frequency of 900MHz