Analysis on the Effectiveness of Printed Circuit Board Edge Termination Using Discrete Components Instead of Implementing the 20-H Rule

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

Liu Enxiao, Er-Ping Li
Institute of High Performance Computing
1 Science Park Dr.
#01-01 Singapore 117528
engp1643@nus.edu.sg, eplee@ihpc.a-star.edu.sg

ABSTRACT

With increasing demand for higher operating frequencies, engineers are faced with numerous obstacles during the design cycle, especially the layout of a printed circuit board (PCB). Concerns include both time and frequency domain analysis. For time domain analysis SPICE is a favorite choice while for the frequency domain, various field solvers are used, application dependent. There is one design concern that cannot be simulated using exclusively either SPICE or field solvers–the 20-H Rule. The reason why one simulation tool cannot be used is due to termination of an RF propagating field (frequency domain) using transmission line theory (time domain).

The placement of components on a PCB and their physical distance to the edge of the board may cause an undesired RF propagating field to be created. The focus of this research is to analyze propagating electromagnetic fields between a pair of power and return planes behaving as a transmission line. With proper analysis, one can incorporate optimal transmission line termination for RF fields using discrete components instead of blindly implementing the 20-H Rule (provides the same benefit). For unique applications, the 20-H rule may be required but only in certain locations of the PCB.

INTRODUCTION

The 20-H Rule is a rule-of-thumb layout technique recommended to minimize the magnitude of radiated fields propagating off the edges of a multi-layer PCB due to RF fields present between a power and return plane. The validity of the 20-H Rule was demonstrated to be a valid PCB layout technique in [1], but only when implemented under certain operating parameters. Blindly using the 20-H Rule without understanding limitations of use, or when applied to a two-, four- or six-layer stackup assignment may cause this technique to behave as a patch antenna due to board resonances.

A patch antenna allows RF energy to propagate in the *z*-axis. The 20-H Rule is specified for minimizing undesired propagating fields in the "*x*-" and "*y*-axis"! In a multilayer PCB with multiple planes, undesired field propagation from a power/return pair can corrupt adjacent cable assemblies, sheet metal enclosures, aperture openings and the like.

Use of the 20-H Rule is a cost effective technique at the expense of lost routing space on the signal layer adjacent to the power plane. If one can terminate transmission lines to minimize field propagating using discrete components (added parts cost), additional routing becomes available for very dense, high-technology PCB assemblies. This research investigates whether the cost savings of cutting back copper on a plane is more effective than using discrete components to achieve the same goal.

FUNDAMENTAL CONCEPT: 20-H RULE

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 (Fig. 1) [2]. Application of this rule, and when it is or is not appropriate is described in [1] as being valid.

The 20-H rule provides PCB edge termination of propagating waves on power/return planes caused by digital components consuming large amounts of peak power inrush surge current, generally at speeds much faster than 1 ns, and only when board resonances do not match switching frequencies injected into the planes. This rule is to be used when there are multiple return planes, and when the power and return plane pair are utilized in true stripline mode (Fig. 2) [3].

Power and return planes must be treated as transmission lines with wave propagation; reflections and ringing (i.e., signal integrity). A propagating wave will return to its source after encountering a high-impedance load (the physical edge of the board). A signal integrity situation now exists, except this involves propagating fields within the dielectric that separates both the power and return planes, similar in nature to a typical transmission line carrying a digital signal [2].

The intended goal of the 20-H Rule is to terminate power and return planes locally toward the circulating current boundaries of the flux and RF fields formed by device–power–currents located at the edges of the board, not to set an arbitrary dimension. The number "20" is not magical. It is the magnitude of benefit achieved when using this rule-of-thumb.

0-7803-8444-X/04/$20.00 (C) IEEE

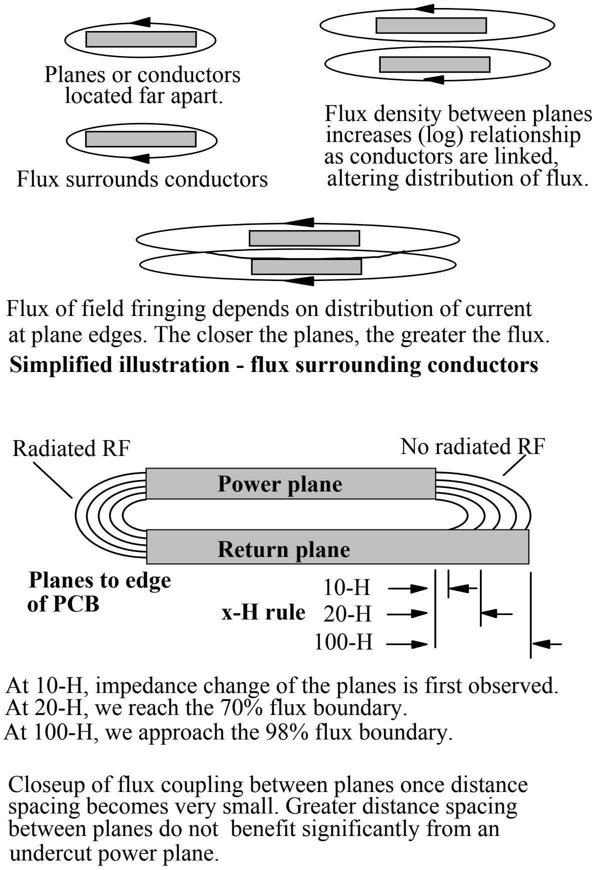
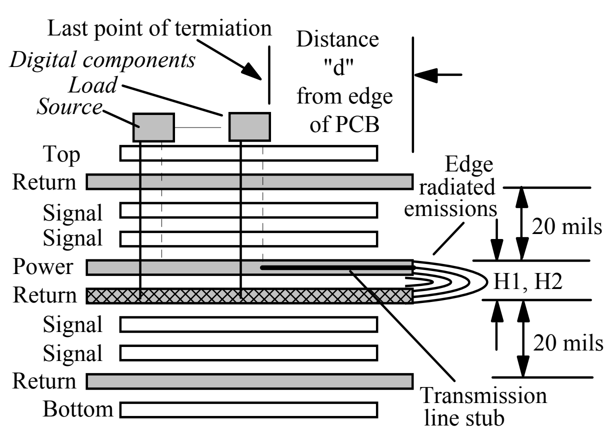


Figure 1. RF fringing effects from planes [2]



Note: The stimulus source is the last load on a transmission line. Distance "d" to edge of PCB represents an unterminated stub trace in regard to the power and return plane.

Figure 2. Board edge radiated emissions (stripline) [3]

TRANSMISSION LINE THEORY

A transmission line must have a source path and return. Power distribution behaves in a similar manner. They consume DC power from a source (power) and use a return (ground). In reality, power/return planes are true transmission lines. For analogy purposes, a source driver sends a signal down a transmission line to a load, and then beyond that load is another trace without termination (stub or the edge of the PCB). The signal will return 100% to its source (reflection) if a high-impedance load is encountered.

When digital components switch logic states, an electromagnetic field is propagated to the edge of the PCB thorough the planes (stub or unterminated transmission line) [3]. This is shown in Fig. 2 while Fig. 3 provides greater detail on stub effects.

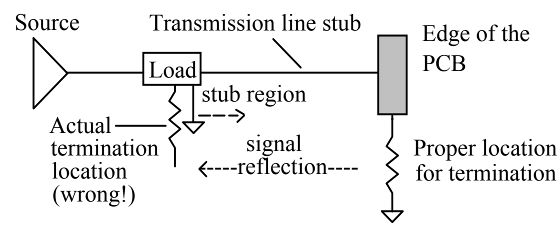


Figure 3. Transmission line stub on planes

Analysis using SPICE is dependent on physical parameters, namely *RLGC*. SPICE cannot simulate a transmission line that is exclusively propagating an RF field. Conversely, field solvers generally do not use *RLGC* parameters to terminate propagating fields. It is difficult to simulate power and return planes as transmission lines incorporating end termination. For this configuration, an *RC* network is examined–*R* to terminate the transmission line and *C* to prevent shorting the power plan to the return plane.

TEST CONFIGURATION AND SETUP

To properly simulate power and return planes as transmission lines, and to determine if greater benefit occurs using discrete components over use of the x-H Rule, the following model was created (Fig. 4).

This model is a four-layer assembly with power and return planes in the middle. To simplify complex simulation, signal/routing layers are defined as a dielectric. Signal traces are omitted. Beyond these signal/routing layers are solid planes that define the boundary limit, as shown in Fig. 2.

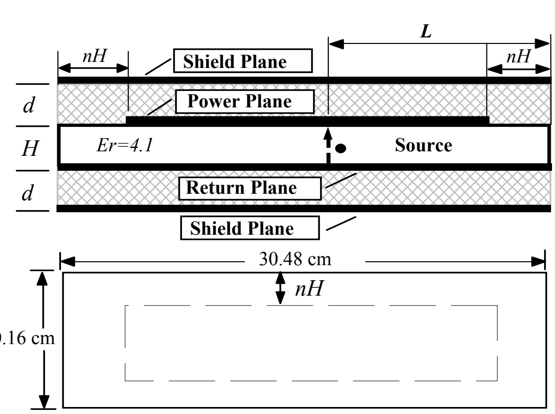


Figure 4. Simulation model

Black dots illustrate the location of series terminators (*RCs*-a series resistor and capacitor).

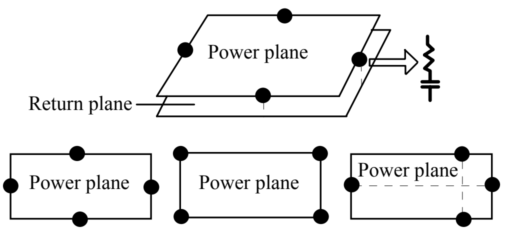


Figure 5. Location of stimulation source

Dimensions and stimulation parameters
Stimulus frequency: 900 MHz, 1.2 GHz and 2.45 GHz
Distance spacing between source and PCB edge:
2.0 cm (0.78 inches)
Distance spacing between power and return planes:
6, 12 mils (0.006, 0.012 inches)
Physical dimension of PCB: 30.48x10.16 cm (12" x 4")
(Typical size of a network interface card)

Stimulation Source
1. Baseband gaussian pulse (impedance calculation)
2. Line source - sinusoidal form (all other simulations)

$$E_{source} = \sin(2\pi fkdt)/H \quad (V/m) \qquad (1)$$

where: f = frequency, k = number of FDTD time steps, H = distance separation, dt = a single FDTD time step:

$dt = 0.05033*10^{-12}$ (s) for $H1$ = 6 mils; and
$dt = 0.97830*10^{-12}$ (s) for $H2$ = 12 mils

The reason why higher frequencies are investigated is due to *real-word* applications of high-technology products using multi-layer PCBs. In a real PCB, the typical physical location of a digital component, relative to the edge of the PCB, is generally 2 cm maximum. Previous investigation of the 20-H rule [1] indicates that this rule-of-thumb layout technique performs best at higher frequencies.

SELF-IMPEDANCE OF THE PCB

As detailed in [4], power/return planes should be considered as a two-dimensional transmission line, also referred to as a radial transmission line. This differs from common signal traces where the signal travels along the axis of conductor while the propagating field generated by the injected signal launches a radial expanding wave. An analytical formula of self and transfer impedances of radial transmission lines with rectangular shapes is also detailed in [4].

The impedance of the power/return plane pair is computed using FDTD [5] (Fig. 6) with L=2cm (0.78") and H=6 mils. At 900 MHz, the impedance is

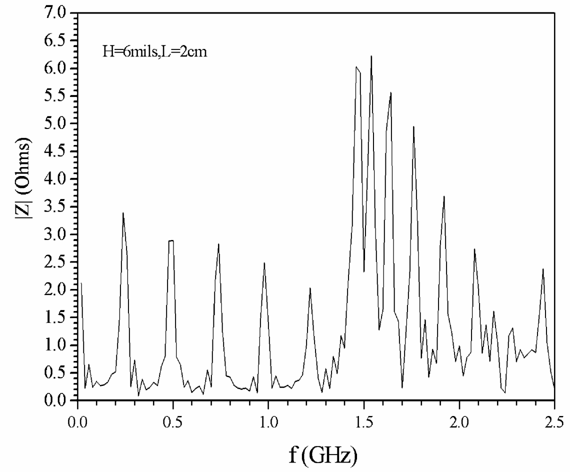


Figure 6. Self-impedance of the PCB assembly

approximately 0.25 ohms; at 1.2 GHz – 2.0 ohms and at 2.45 GHz – 1.5 ohms.

ANALYSIS OF SIMULATED RESULTS

Simulating planes as a transmission line is difficult using commercial SPICE simulators, as SPICE is based mainly on RLGC parameters. A customized version of SPICE is embedded in our FDTD simulator to solve this problem of transmission line termination of a propagating field.

All transmission lines propagate as an electromagnetic (EM) field, which make FDTD an optimal choice of simulation of board edge radiation.

Since we are dealing with an EM field, we investigate actual radiated power physically present at the edge of the PCB through use of a field solver that calculates both electric (***E***) and magnetic (***H***) fields *simultaneously*. A customized version of FDTD permits accurate determination of the Poynting power that emanates from the PCB. When studying the effects of the 20-H rule, the primary area of concerns is harmful coupling of radiated RF energy to adjacent structures (cables, chassis, etc) [1], "*not*" the field strength that one is generally concerned with when attempting to measure very low level EMI in the far-field within an anechoic chamber.

The Poynting vector (or Poynting power) represents actual radiated power, defined as electromagnetic energy density, Eq (2).

$$P = E \times H \quad (\text{watts/m}^2) \qquad (2)$$

Total energy radiated from the PCB edges can be expressed as the integration of energy density P:

$$P = \oint_s E \times H \quad ds \quad (\text{watts/m}^2) \qquad (3)$$

where E and H denotes electric and magnetic field strength respectively. For instance, the emitted energy from a 2-D structure is the integration of the power density along edges AB, BC, CD and DA, shown in Figure 7 [1].

Figure 8 illustrates various test configurations, calculating radiated power from the edge of the PCB.

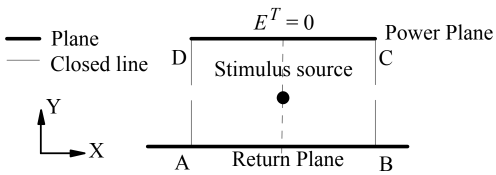


Figure 7. Structure for calculation of Poynting power

In Figure 8 we use the time domain to study edge radiation instead of the frequency domain. Since propagating fields are sine waves, the amplitude of the radiated power will vary with respect to time. This is one advantage why FDTD is an optimal tool, since the peak magnitude over a certain time period is that element which cause EMI concerns.

Only the configuration of *H*=6 mils and *f*=1.2GHz is presented at this time for brevity. Numerical analysis showed that the conclusions drawn from these series of configurations are applicable to other configurations, with minor differences (presented later in this section).

Figure 8(a) compares the Poynting power due to *RC* termination at different locations. When four pairs of discrete components are symmetrically placed at the midpoints of the edges of planes (Fig. 5), the Poynting power is the smallest. Placement of four pairs of terminators described in [6] successfully reduced the input impedance of the power/return planes. When the location of the four terminators is fixed, but the value of the capacitor is increased from 30pf to 100pf, the pointing power remains virtually unchanged. This tells us that the resistor is the important part of the discrete termination (*RC*) pair whereas the capacitor prevents a DC short between the power and return planes.

In Fig. 8(b), terminators are located at the midpoints of the board edges. The capacitor remains at 30pF but with different resistor values. Although a 2-ohm resistor approaches the impedance of the plane pair at 1.2 GHz (Fig. 6), the Poynting power with both 50- and 100-ohm resistors provides the smallest amount of radiated power. Similar to the influence of the capacitors in Fig. 8(a), it appears that radiation is not sensitive to the value of the resistors, unless the parallel equivalent value of all resistors approaches the impedance of the planes.

In Fig. 8(c), we investigate placement of *RC* terminators and how much radiated energy is present that may cause harmful interference to adjacent structures. The Poynting power is the smallest when four pairs of RC terminators are located at the midpoints of edges with R=50 Ohms. For comparison purposes to determine which termination method is

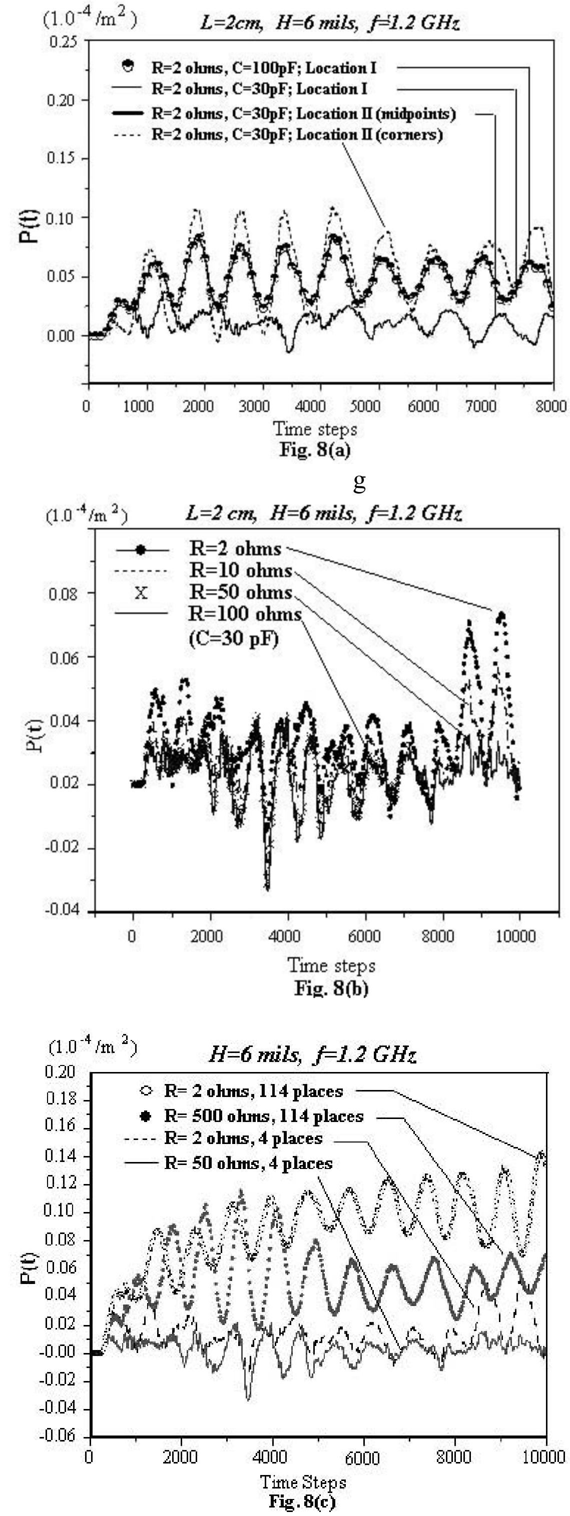


Figure 8. Comparison of Poynting power with terminators at different physical locations

best, placement in the four corners and stitching around the board with 114 *RC* pairs were studied.

The reason why placement of termination in the middle of the edges of the PCB works best versus placement in the corners or multiple locations lies in the disruption of the propagating RF field. A physical dimension is present along the edge of the PCB. This dimension can be optimized as a slot antenna. By shorting out this transmission line in the middle makes this now an inefficient antenna.

Attempting to validate simulated results in an anechoic chamber would provide no benefit as the magnitude of the propagating field off the edge of the board is so small that detection in the far-field is nearly impossible. This is a near-field coupling problem. Accurate measurement in an anechoic chamber would require use of active digital components, which by themselves would skew measurement results.

Table 1 presents edge radiated average power. Using average power (watts) instead of energy density (watts/m^2) is more meaningful to a designer trying to understand the data. Average power, Eq. (4), is obtained by integrating the Poynting vector (power) over the entire enclosed surface of the model, then dividing it by the simulation time (50,000 time steps).

$$\overline{P} = \frac{\int_0^T P(t)\,dt}{T} \tag{4}$$

Table 1. Electromagnetic edge radiated power

Freq. (MHz)	xH Rule	Distance "H" between planes Power (10^{-4} watts)	
		H=6 mils	H=12 mils
900	0H	4.7672	0.9049
	20H	5.0051	1.1694
	RC	15.4151	3.0973
1250	0H	4.9843	6.0609
	20H	2.5566	2.6157
	RC	94.6462	15.95419
2450	0H	7.0882	20.6887
	20H	2.8079	9.9054
	RC	15.3664	38.9587

It is noted that use of the x-H rule *provides significant benefit over use of discrete components* at all frequencies examined with enhanced performance.

When using discrete components to terminate the transmission line stub of the power and return plane pair based upon physical location of the stimulus source relative to the edge of the PCB, a significant increase in propagating energy occurs. This is due to the termination components presenting themselves as a launch point, energized, to behave as a dipole antenna hanging off the edge of the PCB, same as a piece of wire attached to the plane pair.

The plots of field propagation used to calculate the values presented in Table 1 are shown in Fig. 9. These calculated values represent the average power propagating from the power/return plane pair.

SUMMARY

It has been shown analytically that use of the 20-H rule provides "greater benefit for termination of planes over use of discrete components."

Planes are transmission lines that require termination. Results presented indicate a cost effective means of reducing potential or harmful radiated power propagated off the board edge(s) to nearby structures, which in use of the 20-H Rule. The magnitude of the propagating field, both electric and magnetic (Poynting vector), is extremely small and cannot cause significant harmful interference in the far-field, whereas near-field coupling may be a concern if the EM field contains enough power to disrupt other operational circuits.

When evaluating the effects of the 20-H rule, applied to a multi-layer PCB with reference planes both above and below the power/return pair, we need only concern ourselves with those fields that propagate external to the PCB in the *x*- and *y*-axis, not the field structure in the *z*-axis, as *z*-axis fields will be captured by the shield partition of a reference plane. *Z*-axis propagation "is" a concern if one tries to implement 20-H on a 2-, 4- or 6-layer PCB!

The real-world application of this analysis and use of the 20-H Rule is targeted toward high-density, multi-layer PCBs; eight or more layers with 4 or more planes.

REFERENCES

[1] Montrose, Mark I., Er-Ping Li, Wei-Liang Yuan, Jiang Yi, Li Le-Wei. 2002. "Analysis on the Effectiveness of the 20-H Rule Using Numerical Simulation Technique." *IEEE International Symposium on EMC.* pp. 329-333.

[2] Montrose, Mark I. 2000. *Printed Circuit Board Design Techniques for EMC Compliance–A Handbook for Designers*. NY: Wiley/IEEE Press.

[3] Montrose, Mark I., Er-Ping Li, Jin Hong-Fang 2004. "Analysis on the Effectiveness of High Speed Printed Circuit Board Edge Radiated Emissions Based on Stimulus Source Location." *IEEE International Symposium on EMC.*

[4] Novak, Istvan, 1999. "Reducing simultaneous switching noise and EMI on ground-power planes by dissipative edge termination." *IEEE Transactions on Advanced Packaging*, V.22, #3.

[5] Van den Berghe, S., F. Olyslager, D. De Zutter, J. De Moerloose and W. Temmerman, "Study of the ground bounce caused by power plane resonances." *IEEE Transaction on EMC*, Vol. 40, No.2, pp.111-119, May 1998.

[6] Peterson, G. W., J. L. Prince and K. L. Virga. 2000. "Investigation of power/ground plane resonance reduction using lumped RC elements." *IEEE Electronic Components and Technology Conference*, pp. 769-774.

0-7803-8444-X/04/$20.00 (C) IEEE

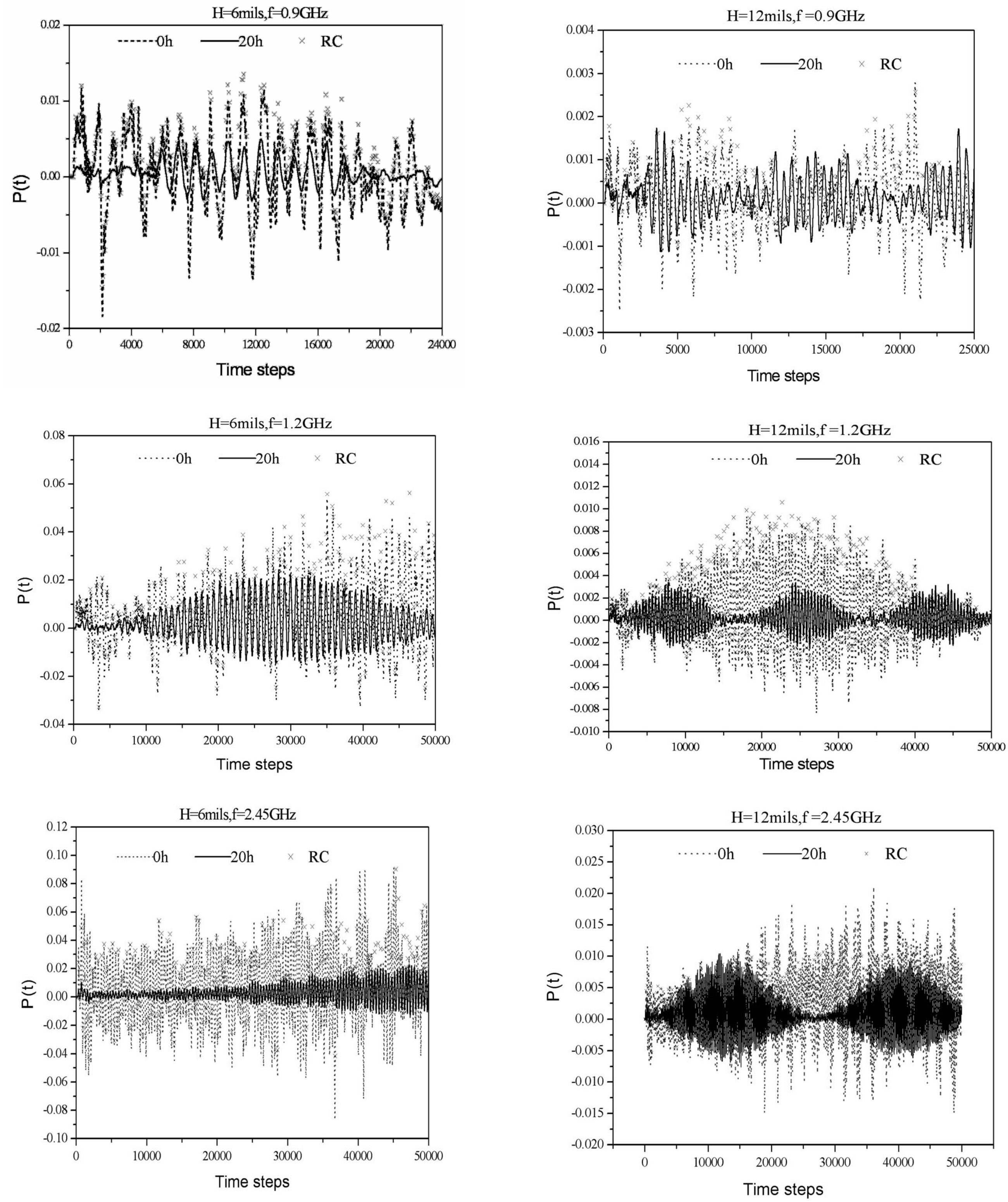


Figure 9. Comparison of EM radiated energy based on the 20-H rule versus discrete RC terminators