Analysis on the Effectiveness of Image Planes within a Printed Circuit Board

Mark I. Montrose
Montrose Compliance Services
2353 Mission Glen Dr.
Santa Clara, CA 95051-1214
(408) 247-5715

ABSTRACT

To analyze the effectiveness of image planes based on clock edge rates in addition to radiated emissions from a printed circuit board, the following is examined.

1. Effects of an image plane on reducing radiated emissions;
2. The effects of distance spacing between a clock trace and an image plane;
3. The effects of microstrip and stripline on radiated emissions;
4. The effect of a gap / slot in an image plane related to radiated emissions

Results of this analysis are from measurements taken on a specially designed printed circuit board in the frequency domain. A companion paper, *Analysis on the Effectiveness of Clock Trace Termination Methods and Trace Lengths on a Printed Circuit Board* uses the same printed circuit board with signal functionality recorded in the time domain.

INTRODUCTION

This paper presents an evaluation related to printed circuit board (PCB) layout that has a significant impact on every electronic device, regardless of clock speed or sophistication of technology. This paper corroborates two previously published papers related to image planes [2] and [3]. Using image plane theory, we investigate different applications of use and problem areas that exist if improper implementation is provided in a PCB layout, along with the effects of distance spacing between a trace and a plane.

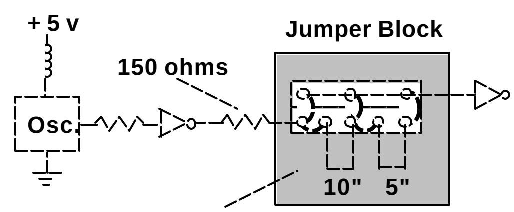
Practicing engineers need to understand fundamental issues related to image planes and trace lengths that affects compliance to domestic and international requirements for radiated emissions. Most design engineers have a limited time to design and manufacture a product which includes modeling and computer simulation. If a product fails radiated EMI tests, rework is required based on ones knowledge of how image planes, and their use, relate to each other.

TEST SETUP

A PCB was designed for flexibility in analyzing different image plane configurations. A driver (74xx04) injects a clock signal down a trace to an active load located on a socket to allow ease of logic family changes. Different edge rates were investigated - 1.0 ns for 74F04 and 5 ns for 74LS04. Each edge rate is sensitive to transmission line lengths for signal functionality purposes as well as creation of RF currents.

Shunt jumpers are provided to change the length of the trace route. Trace lengths vary from 3 inches (electrically small for signal propagation; time for the signal to travel from source-to-load and load-to-source) to 18 inches (electrically long which allows ringing and reflections to exist). Additional programmable trace lengths include 8 inches and 13 inches, which are not discussed herein. In the initial configuration, 15" inches of the trace are not located over a copper (image) plane so that configurable image planes could be added later. It should be noted that the use of these jumpers described above for PCB configuration change could be responsible for small impedance discontinuities in the test signal path, thereby exacerbating emissions over those emissions that would have been experienced under ideal conditions of no path discontinuities.

A perfect clock signal was observed on the spectrum analyzer in the frequency range 30-1000 MHz. Comparisons between configurations are based on a fixed test setup without regard to specification limits for Class A or B products. The test setup is shown in Figure 1.



No image plane provided on the PCB
Copper tape used to create an image plane

Figure 1. Schematic representation of test fixture

By routing the 5" and 10" traces over bare FR-4 material, simple modifications to the board allowed for four image plane configurations to be tested using copper tape as a configurable image plane. For each configuration, radiated emissions are recorded to observe effects related to

1. free space (no image plane)
2. a microstrip plane at 0.0014" away from the clock trace (top side of the board)
3. a microstrip plane at 0.062" away from the clock trace (bottom side of the board)
4. stripline (image planes both above and below the trace)
5. segmented stripline (plane with moat or absence of copper)

A spectrum analyzer is used to measure RF emissions as it relates to distance separation of the trace to the image plane. Various clock edge rates are used to change the spectrum profile of the radiated emissions.

German, Ott and Paul [2] demonstrated the effects of placing a conducting plane adjacent to a trace simulating an asymmetric dipole. Their results showed that a reduction of 16 dB occurred with this plane. This reduction is due to differential-mode and common-mode currents flowing on the signal and ground-return traces by inducing currents on the conducting plane which canceled the emissions from the source currents.

German and Dockey [3] examined a trace embedded in a PCB, stripline configuration with a ground plane below and a shunt plane above (symmetrical distant spacing). The copper plane was both isolated from, and then connected to the referenced ground plane. A significant reduction in emissions was observed at most frequencies while an increase in emissions occurred in very narrow bands [4], thus validating the need to connect the ground plane to a reference source. This occurs only if the ground plane is connected at a single point. If these peaks occur at a harmonic of the system clock frequency, other design and layout methodologies would be required to reduce radiated emissions. If the copper plane is connected to the ground plane around its periphery, radiated emissions are decreased by up to 30 dB. This type of implementation, 100% grounding on the periphery, however, is not possible in a PCB layout.

ANALYSIS OF TEST DATA

Baseline measurements were taken for both 74F04 and 74LS04 with 18" trace lengths. Plots of this data in free space (trace without an image plane) is shown in Figures 2a and 2b. As observed, the 74F04 with the faster edge rate created a larger spectral distribution of clock harmonics than the slower 74LS04 per the equation $f_{max} = 1/(\pi * t_r)$ where t_r is actual edge rate. This equation does not take into account harmonics that are created from the oscillator (20 MHz). These plots also justify the need to use the slowest logic family possible in PCB design.
Due to limited space allocation in this paper, only worst case emissions from the 18" trace, 74F04 is presented. By similarity, this analysis applies to all configurations. During the test, copper tape was applied to the PCB to create the image plane.

Plots for all configurations are shown in Figure 3 (ground plane *unconnected* to ground reference) and Figure 4 (ground plane *connected* to ground reference). Significant improvement, 10 dB to 20 dB, was observed when the image plane was referenced to ground. As expected, the ungrounded image plane did not provide a return path for RF currents, which in turn allowed for greater RF emissions to exist.

The physical and mathematical analysis as to why an image plane performs best when adjacent to a signal trace is described in great detail in [3].

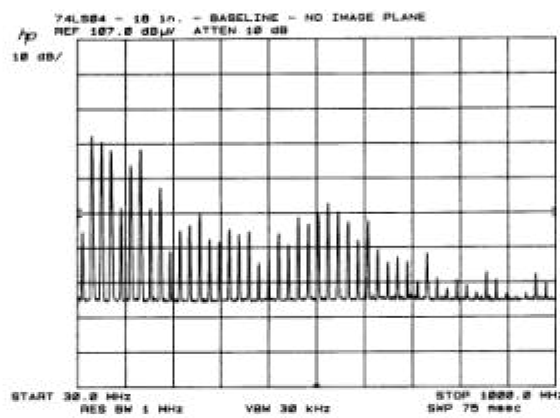


Figure 2a. Baseline Measurements - *74LS04* 18" trace
No image plane

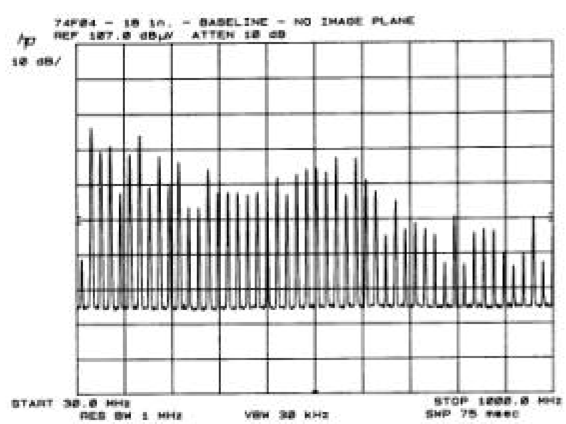


Figure 2b. Baseline Measurements - *74F04* 18" trace
No image plane

Image Plane - *Not Connected* to Ground

Image Plane - *Connected* to Ground

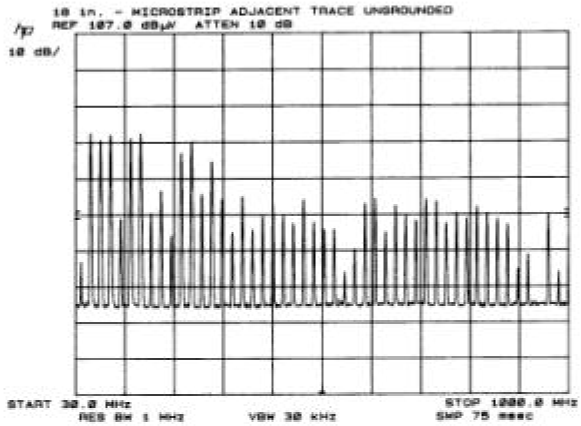


Figure 3a. Microstrip Adjacent (0.014") to 18" Trace

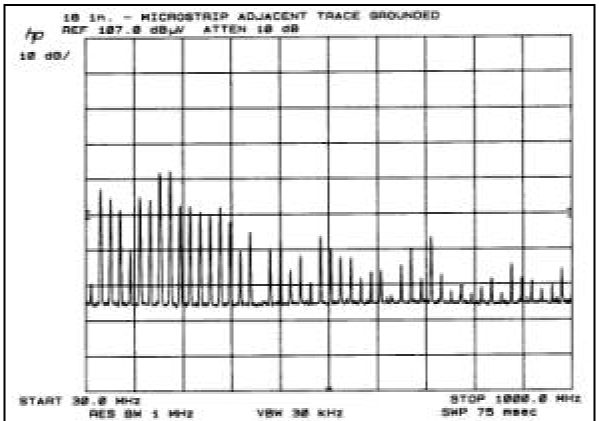


Figure 4a. Microstrip Adjacent (0.014") to 18" Trace

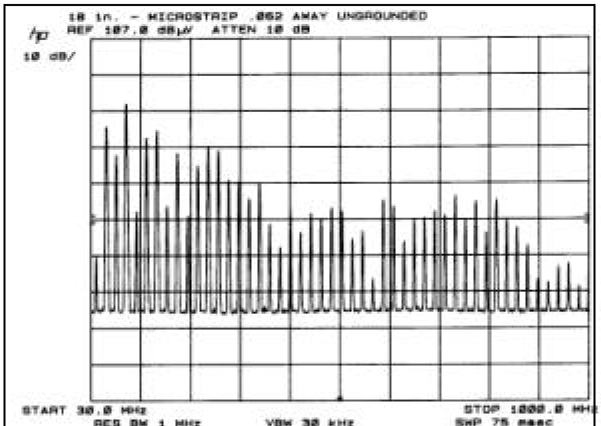


Figure 3b Microstrip 0.062” Away from 18" Trace

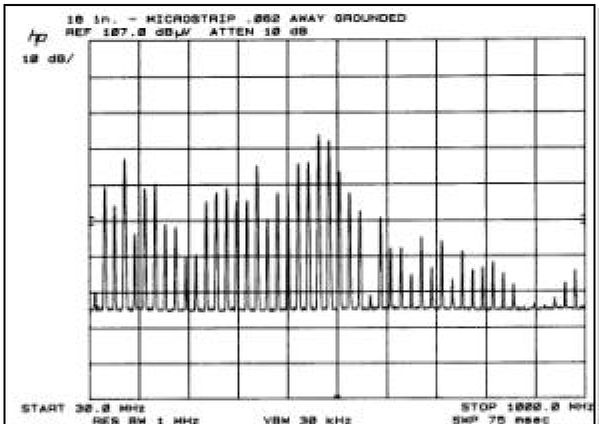


Figure 4b Microstrip 0.062” Away from 18" Trace

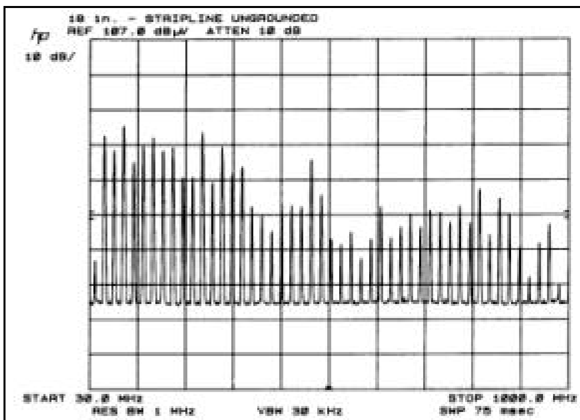


Figure 3c. Stripline Surrounding 18" Trace 0.014 / 0.062 "

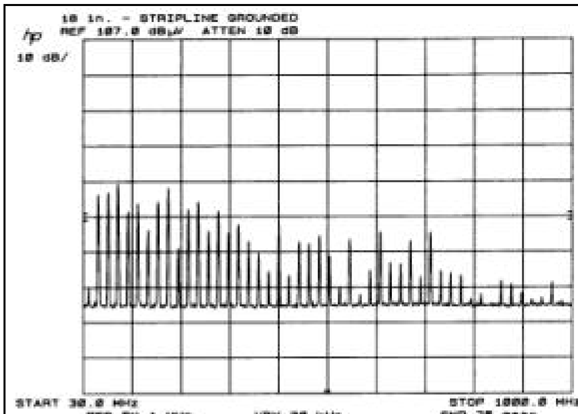


Figure 4c Stripline Surrounding 18" Trace 0.014 / 0.062 "

Notice that for the “not connected” to ground reference plots, Figures 2b, 3a, 3b and 3c, all four are almost identical. This illustrates what can occur if an image plane is not referenced to the main signal ground reference. No reduction in radiated emissions exist.

Comparing the two microstrip configurations, Figures 4a and 4b, the difference between the plane adjacent to the trace (0.0014") and the plane 0.062" away is caused by increased partial mutual-inductance between the trace and image plane. Close distance separation decreases ground-noise voltage and the resulting common-mode currents on the ground plane, hence a better plane.

The difference in the behavior of microstrip and stripline is related to the return current path. For microstrip, the return path is spread out in the ground plane much more than stripline. For stripline, return current is not spread more than 2-3 times the spacing (*h1*) between trace and plane [4], shown in Figure 5. If the trace is at the edge of the board (plane), the return path is disturbed and an increase in RF emissions will exist.

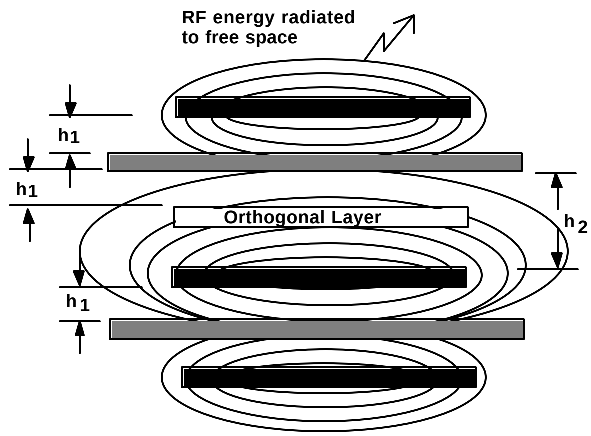
The spectral amplitude spikes for certain frequencies noted in the plots is due to the resonance of the image plane and where the image plane was connected to the main ground reference point of the circuit. One ground point was located physically closer to the clock driver from that of an alternate ground connection.

USE OF IMAGE PLANES

The concept that a bare piece of copper internal to a PCB will always provide an image plane for RF return currents is not uniformly true. When the tape was connected to ground reference, tied to the supply return, a significant reduction in radiated emissions was observed. The same results were recorded whether a ground connection was made at a single point or multiple location. The image plane contains large amounts of RF currents. Once the return path was provided, almost all differential-mode flux in the trace was canceled by the grounded image plane when referenced to the driver’s power/ground pin.

This paper now investigates the effects of distance spacings between the two microstrip configurations and the stripline configuration. It confirms the results in [3] that the closer the image plane is to the trace, the greater reduction in RF emissions. This can best be described by the simple diagram of Figure 5. Flux distribution propagated from the trace are captured by the nearest image plane, *h1* distance spacing away. For the top configuration, the trace is 0.0014 inches away. Increased coupling for the closer image plane allows larger RF flux cancellation to exist especially in the higher frequency range as seen in Figure 4a in the range 400-600 MHz.

If RF current is captured by the nearest image plane, Figure 4a, the plane located further away, Figure 4b (*h2*) will have minimal flux cancellation. This is observed in the plots of Figures 3 and 4. A stripline configuration, in most cases, prevents RF currents from radiating into free space. A microstrip trace located immediately adjacent to an image plane provides the same performance as stripline, since flux cancellation occurs by the adjacent image plane. This is true when using dual stripline stackup (*h1 spa*cing). Stripline performs best, since both routed traces are adjacent to an image plane at *h1* spacing. The net partial mutual inductance between a trace and plane is significantly reduced as the two are brought closer together [2].



Six layer stackup
2 microstrip layers
2 stripline layers

Note: Actual stripline layers are always orthogonal, not as shown above to prevent crosstalk

Figure 5. Flux cancellation for a PCB trace

The image plane caused a reduction in emissions because common-mode radiation is usually predominant in a circuit board. Common-mode currents are created through several mechanisms with ground-noise voltage the predominant cause. Ground-noise voltage can be reduced by (1) lowering the RF current on the trace, (2) increasing the current rise-time (using slower speed devices), or (3) lowering ground-return inductance. The first two methods are easy to achieve. Reducing ground-noise voltage is a more difficult problem.

EFFECTS OF A SEGMENTED IMAGE PLANE

The effects of a segmented image plane are now examined. A slot was made in the plane shown in Figure 6.

The reason a slot was provided in the image plane was to simulate the effects of ground plane partitioning (moating) commonly used in digital-to-analog separation, isolated power planes, and I/O interconnects. When a slot is provided in a plane, the

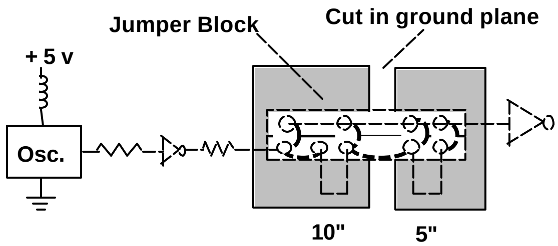


Figure 6. Effects of image plane segmentation

RF return current must travel around the slot in an attempt to return to its source, in effect producing a loop. This return loop path distance could be significant, with an increase of inductance in the return trace and resultant increase in RF emissions. A visual representation of increased return trace inductance is shown in Figure 7.

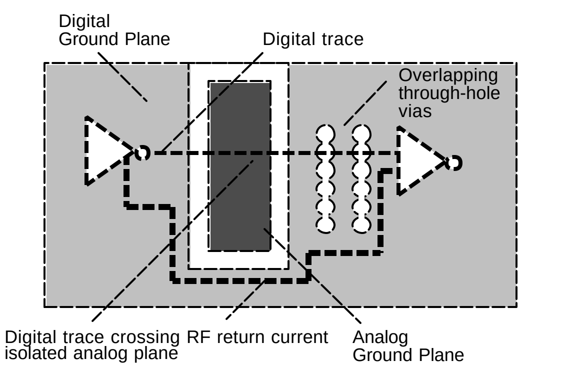
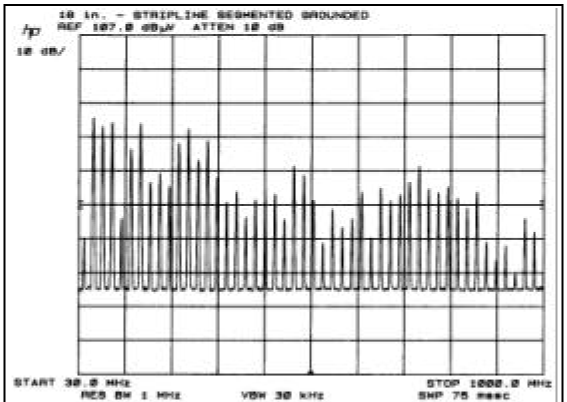


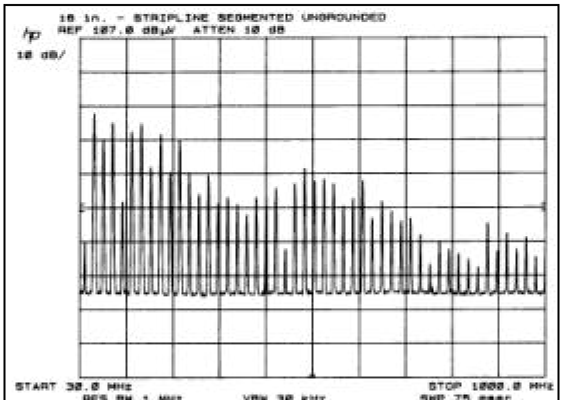
Figure 7. PCB routing of a trace over a segmented image plane

The same results occur when a through-hole component is provided on a PCB. If the air gap (anti-pad) surrounding the via overlaps an adjacent anti-pad, a continuous discontinuity will exist. This is exacerbated if multiple vias are provided in a straight line. Figure 8 shows the measured results of a stripline configuration with slot (moat) provided.

Excessive perforation of power/ground planes, especially anti-pads for vias and plated through holes not only cause EMI radiation, but also results in signal integrity problems due to the inductive nature of such discontinuities. Hence, the quality of the power and ground planes as image planes must be maintained.



Stripline *not* connected to ground reference
Segmented



Stripline connected to ground reference
Segmented

Figure 8. Measured Effects of a Slot in a Stripline Configuration

Use of an image plane, is required to achieve a controlled impedance transmission line. These transmission lines (planes) have very low emission levels only when connected to the main driver's power and ground reference. Connection to the ground reference point allows RF return currents to complete their path of travel. As observed in Figure 8, a fully connected image plane is required for optimal flux cancellation.

MULTIPLE IMAGE PLANES IN A PCB

When using an image plane for enhancement of RF return currents in a PCB, we observe the following.

Optimal performance of extra-high-speed clock traces are achieved when they are routed "adjacent" to a ground plane and "not adjacent" to the power plane. This is one of the basic fundamental concepts of EMI suppression in a PCB.

The reason why multilayer boards provide superior signal quality and EMC performance is because signal impedance control through stripline or microstrip is observed. The distribution impedance of the power and ground planes must be dramatically reduced. These planes contain RF spectral current surges caused by "logic crossover," momentary shorts, and capacitive loading on signals with wide buses. Central to the issue of microstrip (or stripline) application is understanding flux cancellation that minimizes (controls) inductance in any transmission line. Various logic devices may be quite asymmetrical in their pull-up/pull-down current ratios. This means that flux cancellation is enhanced between the signal and the ground planes rather than the power planes. With this situation, use of the power plane as a flux cancellation control may not present an optimum condition, resulting in signal flux phase shift, greater inductance, poor impedance control, and noise instability. Use of the ground plane for optimal signal reference is thus preferred. [1] [5].

SUMMARY

Knowledge of how a PCB functions in the frequency domain is important to today's products. As observed, image planes only provide optimal performance when referenced to the driver's main ground reference. Distance separation of a trace layer to an image plane also determines the amount of flux cancellation that will occur. Any discontinuities in the RF current return path can present the same environment as if no image plane existed at all. Routing signal traces over isolated planes will result in increased RF emissions.

Proper design implementation techniques related to image plane grounding, layer stackup and trace routing is necessary for compliance with radiated EMI requirements.

ACKNOWLEDGMENT

Thanks to HADCO Corporation for providing the printed circuit board used in this research project. Additional acknowledgment to Robin Simpson, PCB designer who performed the board layout, Bill Baldwin and W. Michael King for their technical review, and Whittaker Communications, Inc., for use of their facilities and support to conduct this research.

REFERENCES

[1] Montrose, Mark I., **Printed Circuit Board Design Techniques for EMC Compliance**, New York: IEEE Press, 1996.

[2] German, R.F., H. Ott, C.R. Paul**. Effect of an Image Plane on Printed Circuit Board Radiation**. Proceedings of the IEEE 1990 International Symposium on Electromagnetic Compatibility, pp. 284-291.

[3] German, RF., R.W. Dockey. **New Techniques for Reducing Printed Circuit Board Common-Mode Radiation**, Proceedings of the IEEE 1993 International Symposium on Electromagnetic Compatibility, pp. 334-339

[4] S. Daijavad, W. Pence, B. Rubin, H. Heeb, S. Ponnapalli and A. Ruehli, **Methodology for Evaluating Practical EMI Design Guidelines Using EM Analysis Programs**. Proceedings of the IEEE 1992 International Symposium on Electromagnetic Compatibility, pp. 30-34.

[5] Technical explanations provide by W. Michael King.