2023 IEEE Symposium on Electromagnetic Compatibility & Signal/Power Integrity (EMC+SIPI) | 979-8-3503-0976-8/23/$31.00 ©2023 IEEE | DOI: 10.1109/EMCSIPI50001.2023.10241478

Analysis of Electromagnetic Interference Problems Caused by Split Reference Plane on High-Speed Multilayer Boards

Xin Fang*[1], Chulsoon Hwang*[2], Michelle Liu#[3], Rodrigo Rodriguez Navarrete#[4]
**EMC Laboratory, Missouri University of Science and Technology, Rolla, MO, USA*
#*Tesla Inc., Fremont, CA, USA*
fangxin@umsystem.edu[1], hwangc@umsystem.edu[2], qianliu@tesla.com[3], rrodrigueznavarrete@tesla.com[4]

***Abstract*—Digital/Analog ground partitioning has been used to isolate noisy digital and power current from sensitive analog currents in high-speed multiplayer printed circuit boards. This design, however, breaks the current return path for signal traces that cross the two separated grounds, which causes undesired effects such as signal distortion and radiated emission. Electromagnetic mechanism associated with them needs to be understood to control and suppress these undesired effects. In this paper, equivalent circuit diagrams are presented to explain the current path in a practical camera device with the separated ground. Finally, optimal stitching via locations is determined to provide a good return current path and thus suppress the radiated emission. Numerical simulations are conducted for validation in frequency ranges from 10MHz to 2GHz.**

Keywords—Split reference plane, return current path, multiplayer board, Electromagnetic interference(EMI), cavity resonance, stitching via

I. INTRODUCTION

Electromagnetic Interference (EMI) is becoming an important issue in the design of modern high-speed printed circuit boards (PCBs). Integration trends are squeezing entire systems into high circuit densities, while frequencies are continuously increasing. The routing of traces usually play a crucial role in determining the EMI quality of PCBs. It is well known that a solid ground plane is required to guarantee the impedance match and good return current path for signal traces. However, there are often situations where splitting a solid ground plane is inevitable. These situations are related to a need to keep low frequency currents produced by a noisy circuit from sharing the same conductive return path as currents in a circuit that is sensitive to currents approximately three orders of magnitude lower. The through-hole signal via enables convenient routing and component placement on the PCB and is used for signal interconnections when the signal traces are changing the layers. However, when the high-frequency signal is transmitted through the through-hole via, the signal encounters discontinuities at reference planes and suffers reflections. There will be no clear return loop for incomplete return ground plane design, giving an interconnect with huge loop inductance, and thus strong radiated EMI [1]. Therefore, it is necessary to understand the signal return current path for split ground plane to control and suppress undesired effect.

While [2], [3] include microstrip lines over a split ground plane, only 2-layer boards were considered. [4] has introduced the idea of adding the stitching capacitor across the gap to reduce radiated emission, but it would add another noise peak at certain frequency. Though [5] pointed out return current path in split reference plane in a 4-layer board, the signal trace didn't change layer. Also, the study of [2], [3], [4], [5] were based on simple structures with only signal trace and ground/power plane.

In this paper, we have investigated and analyzed the radiated emission peak below 2 GHz for 6-layer camera PCB with a signal trace go across gap and change reference planes. In the layout review stage, a strong noise peak from the camera PCB was found in GNSS band, which greatly affected the function of GPS nearby. The gap was removed and more stitching vias were added in the next layout generation due to IC operating design requirements. However, it's necessary for further study to explain the noise peak and summarize the criteria on stitching via placement. For the research purpose, the frequency range was extended to 2GHz to show how resonance shift and peak magnitude change. According to this board structure, the power plane is between two ground planes, which can result in the high impedance current path and open edge radiation by the power/ground resonance. To explain the effect of the parallel plane resonance for our 6-layer PCB, we have proposed two cases of equivalent circuit diagram, considering the separated digital and analog ground. The excitation of the power/ground plane resonance is modeled as impedance between adjacent planes of via body. The resonance peak is well predicted by the suggested circuit diagram. We have simulated the total radiated power (TRP) from the PCB, and confirmed that the peak radiation frequencies are well matched to the power/ground resonance frequencies. The circuit diagram is also proven quite well matched to the TRP simulation up to 2 GHz. Besides, we found that the coupling to the power plane is maximized, when the impedance of the power/ground plane is maximized. This is verified with the surface current comparison of the power plane, which increases by 30dB at the maximum coupling. Finally, we have investigated the function of the GND vias, to avoid the radiated emission problems related to the presence of the ground slot. The GND vias provide the return current path of the signal lines even when the ground plane is slotted. There should be a compromise between the design feasibility for manufacture and the guarantee of the return current path, thus the location of the stitching capacitor is designed. Placing vias along the gap with $\lambda/37.5$ distance can suppress the radiated emission power by 22dB.

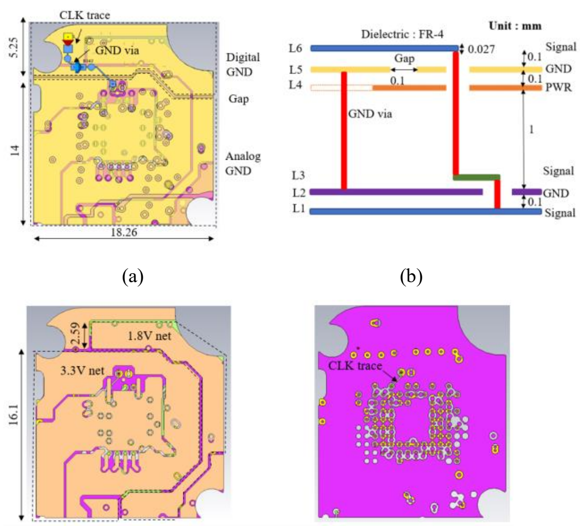
II. PROBLEM DESCRIPTION

Fig. 1 shows a 3D model of the camera board under investigation. The board is a 6 layer PCB with dimensions of

This work is supported in part by the National Science Foundation under Grant No. IIP-1916535.

16.25 mm by 18.26 mm. The dielectric layers are FR-4 and the relative dielectric constant is 4.3. The CLK signal trace goes from L6 to L1 through a signal via with a 50 Ohm termination at the end. A L5-L2 ground via beneath the signal trace provides path for return current. The digital power plane with different power net and L2 is a solid digital ground.

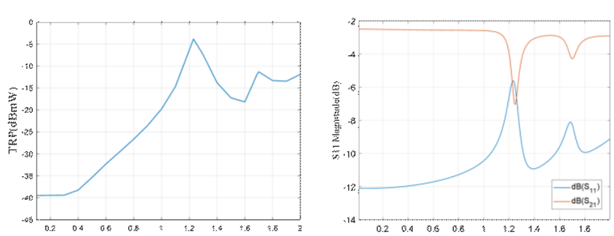
In this system, it was observed that there was a peak at 1.23GHz and 1.688 GHz in TRP below 2GHz when exciting the CLK signal trace, shown in Fig. 2(a). TRP at this frequency is around 20 times higher than the average level, which can cause the performance degradation and critical error by generating noise on nearby signal lines or integrated circuit(IC). S parameter results in Fig. 2(b) show large reflected wave at 1.23GHz and 1.688GHz. Besides, in Fig. 3, at 900 MHz, current mainly return from the GND via and return current concentrate on the top right of the camera board. However, at 1230 MHz, current distributes all over the board and lots of current flows to the bottom of the board, which can result in larger current loop and stronger radiated noise. To explain the reason for the TRP peak and S parameter resonance as well as search effective method for a minimal return current loop and reducing the radiated emission, we built models to investigate the effect of board structure on radiation by simulation.



(a) (b)

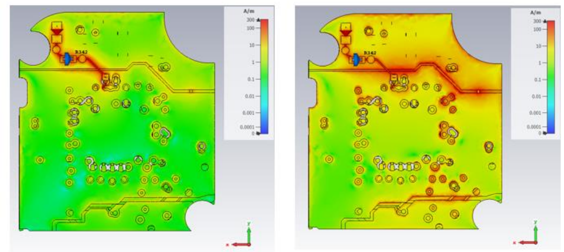
(c) (d)

Fig. 1. 3D model of the camera board under investigation. (a) Top view. (b) Stack-up. (c) Top view of layer 4. (d) Top view of layer 3 and 2.



(a) (b)

Fig. 2. Simulation results when exciting CLK trace. (a) TRP. (b) S parameter.



(a) (b)

Fig. 3. Surface current distribution of different frequency. (a) At 900 MHz. (b) At 1230 MHz.

III. MODELING APPROACH FOR RETURN CURRENT PATH

The through-hole signal via is used for signal interconnections when the signal traces are changing the layers. However, when the high-frequency signal is transmitted through the through-hole via, the signal encounters discontinuities at reference planes and suffers reflections. It is because the reference plane is not continuously guiding the high-frequency electromagnetic waves to support the TEM wave along the signal trace. In addition to the signal reflection problem, the through-hole signal via excites the electromagnetic waves between planes, since the via is passing across the parallel plate waveguide in the multi-layer PCB and via has the parallel plate as a return current path (Fig. 4). Accordingly, depending on the edge termination condition, material and dimensions of the PCB, standing waves appear with multiple resonance frequencies inside the power/ground plane and make a high-impedance return current path. These resonance waves are responsible for the open edge radiation from the PCB, and are major part of the edge radiation from the high-speed and high-density multi-layer PCB. The noise voltage in Fig.4 is the product of return current and power/ground plane impedance.

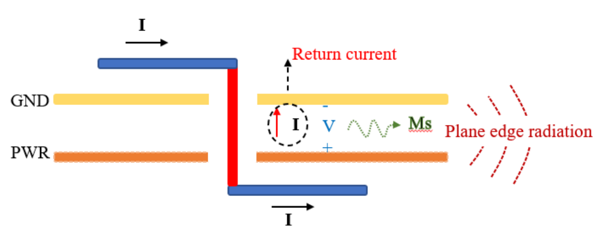
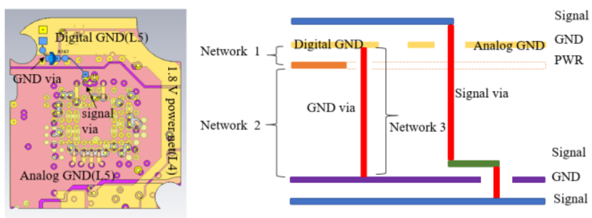


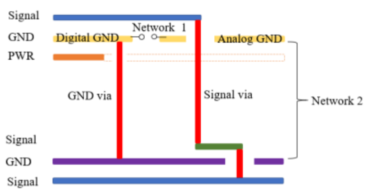
Fig. 4. Power/ground plane edge radiation excited by clock

The camera board has a power plane L4 between L2 and L5 ground planes. As illustrated in Fig.4, high parallel plane impedance can be seen at the resonant frequency, resulting in strong radiated emission. Therefore, the resonant frequency of total parallel plane impedance along the return current path should be the resonance frequency of radiation. In our board, there are more than one parallel plane pair, thus a proper equivalent circuit diagram should be built to get the resonant frequency.

The 6-layer PCB has independent plane pair networks as Fig. 5, since a skin depth of above 50MHz(< 9 um) is smaller than a copper metal plane thickness(>18 um). The through hole via can tie independent networks into a single network by return current path. Fig. 5(b) and Fig. 5(c) show the different independent networks with 1.8V power net as L5 is split as digital ground and analog ground. As shown in Fig. 5(b), for case 1, there are three independent networks when the return current flowing along the GND via and the plane through which the via passes. In Fig. 5(c), for case 2, there are two independent networks when current return from the coupling between digital ground(L2) and analog ground (L5), and the coupling between the gap in L5.



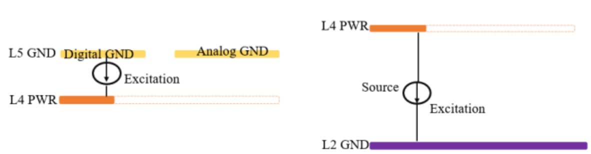
(a) (b)



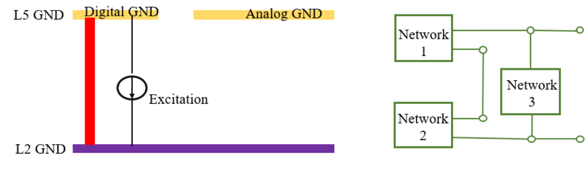
(c)

Fig. 5. (a) Top view of board with 1.8V power net (b)Independent networks of case 1. (c) Independent networks of case 2.

Fig. 6 shows details of each network and the equivalent circuit diagram to connect three networks. As return current is symmetric to induced current for signal via, we can use excitation source to replace signal via to get parallel plane impedance for each network by simulation. The total parallel plane impedance is calculated as Fig.6 (d) by using the impedance simulated from Network1/2/3. Network 1 is formed by the lower surface of L5 digital GND and the upper surface of L4. Network 2 is consisted of lower surface of L4 and the upper surface of L2. Network 3 is to take the effect of GND via into consideration. As the size of the board is small and the additional inductance induced by plane can be ignored, the effect of the GND via can be simulated by adding excitation between L5 and L2 while removing other layers. The equivalent circuit diagram is the serial connection of network 1 and 2 as well as the parallel connection of network 3.



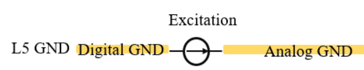
(a) (b)



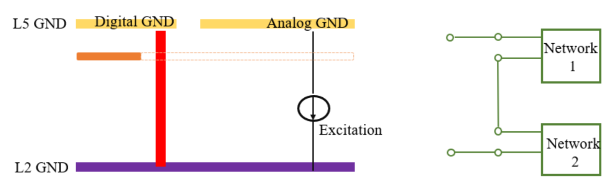
(c) (d)

Fig. 6. Independent networks of case 1. (a) Network 1. (b) Network 2. (c) network 3. (d) Equivalent circuit diagram.

If the current returns from the coupling between L2 digital ground and L5 analog ground as well as the coupling between analog ground and digital ground in L5, there will be two independent networks as shown in Fig.7. Network 1 consists of the upper surface of digital and analog ground in L5. Network 2 is formed from lower surface of L5 analog ground and upper surface of L2 digital ground. The equivalent circuit diagram is the serial connection of network 1 and 2 as illustrated in Fig.7 (c).

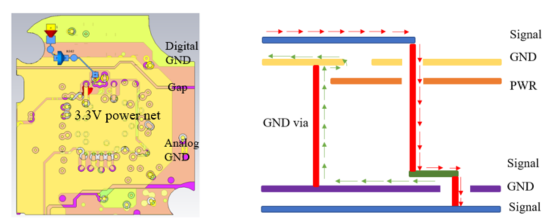


(a)

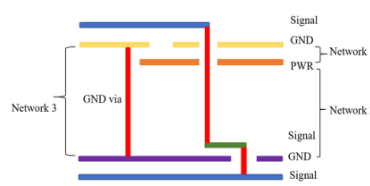


(b) (c)

Fig. 7. Independent networks of case 2. (a) Network 1. (b) network 2. (c) equivalent circuit diagram.



(a) (b)



(c)

Fig. 8. (a) Top view of board with 3.3V power net. (b) Current return path. (c) Independent networks.

For 3.3V power net in Fig.8 (a), the return current flowing along L5, the GND via and going back to L5 digital GND. There are also three independent networks as shown in Fig.8 (c). The equivalent circuit diagram is the serial connection of network 1 and 2 as well as the parallel connection of network 3. Total parallel plane impedance can be calculated when looking into the equivalent circuit diagram in Fig.9 (d).

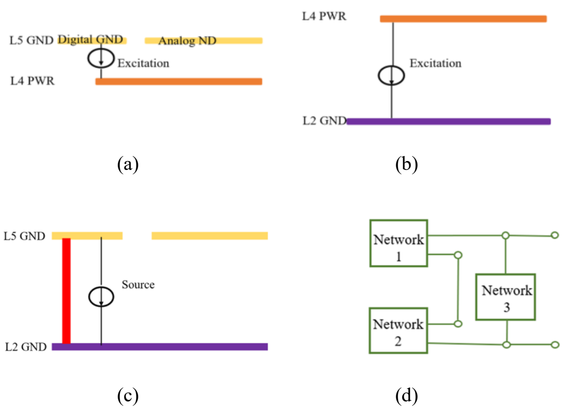


Fig. 10. Independent networks of case 1. (a) Network 1. (b) Network 2. (c) network 3. (d) Equivalent circuit diagram.

In Fig. 10, the total parallel plane impedance calculated from two cases are compared. Parallel plane impedance of case 1 is much lower than the that of case 2. As the current always flows along the least impedance current loop, it can be analyzed that current prefers current path of case 1 to return. When comparing the impedance curve of case 1 along frequency, we can observe peaks at 1230MHz and 1688MHz, which corresponds well with the resonant frequency shown in Fig. 2. Reflection at input port and signal loss are increased at resonance. This reason is low line impedance, which is supported by high impedance of plane at resonance. Strong radiation can be observed as the power/ground plane captures more power than the 50 ohm load receiver.

Fig. 11 shows that at the resonant frequency, instead of returning from ground via, current flows back to source from nearby plane coupling. Surface current increases 30dB at resonant frequency. Current flows back from digital ground(L2) to power plane(L4) coupling and power plane(L4) to digital ground(L2) coupling. Power plane(L4) is used as return path.

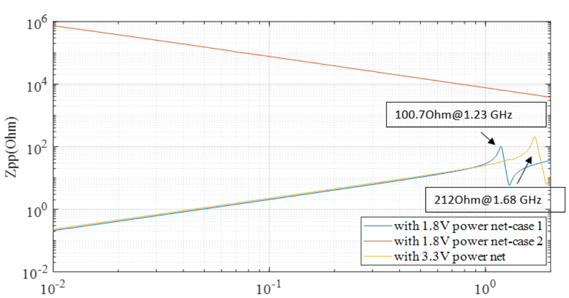


Fig. 10. Total plane impedance comparison along different current path.

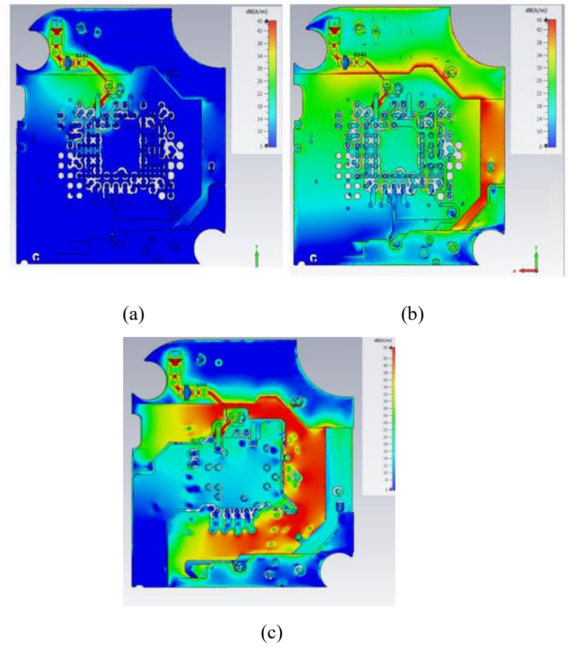
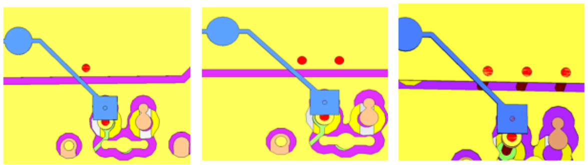


Fig. 11. Surface current distribution of L4 and L2. (a) At 900 MHz. (b) At 1230 MHz. (c) At 1688 MHz.

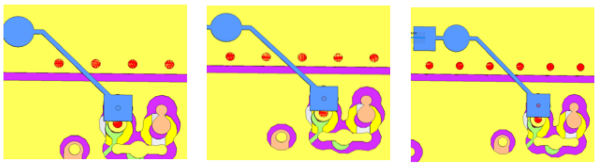
IV. IMPROVEMENTS

When signal trace change reference planes, adding GND vias between two ground planes can suppress the radiated emission because multiple GND vias can reduce the impedance of return current path. Numbers and locations of the GND via should be taken into consideration. Fig. 12shows adding 1/2/3/4/5/6 vias close to the signal via. The distance between two vias is 0.8 mm.

The above independent network of case 1 has been applied to this multiple GND vias case. Total parallel plane impedance are calculated in Fig. 13 and TRP results are simulated in Fig.14. The effect of GND via at the slot can be predicted very well by the independent network as the TRP peak curve has the same tendency as the Zpp curve. Hence, it is demonstrated that the GND vias provides the return current path through the ground plane and the reduced return loop impedance results in the reduced amount of the radiated emission. From the result comparison in Fig. 13 and Fig. 14, GND via should put as close to the signal via as possible. Adding multiple ground vias is also found to be an effective way to reduce radiated emission. 6 GND vias can reduce around 20dBm noise at the resonant peak comparing to 1 GND via. However, TRP peak curve converges when the via number increases, thus adding more GND vias will not be an feasible way to further reduce radiated emission.



(a) (b) (c)



(d) (e) (f)

Fig. 12. Add GND vias close to signal via. (a) 1 via. (b) 2 vias. (c) 3 vias. (d) 4 vias. (e) 5 vias. (f) 6 vias.

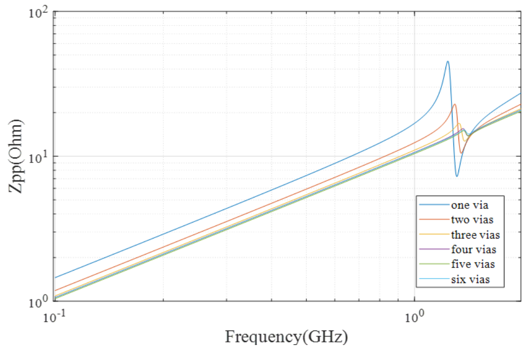
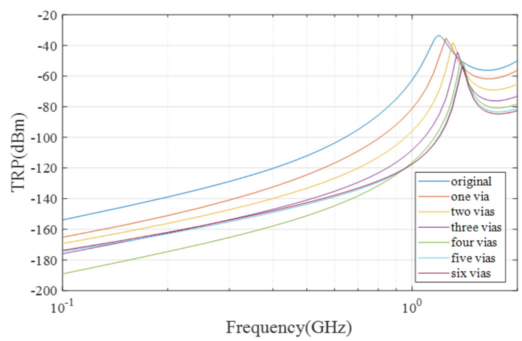
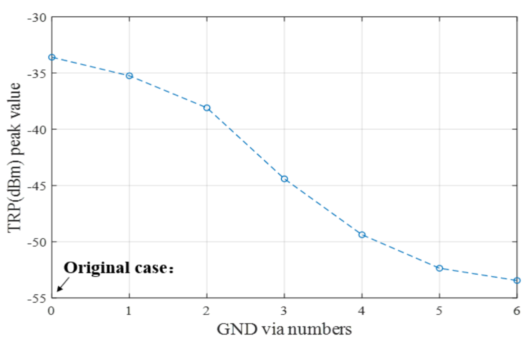


Fig. 13. Total plane impedance comparison of multiple vias



(a)

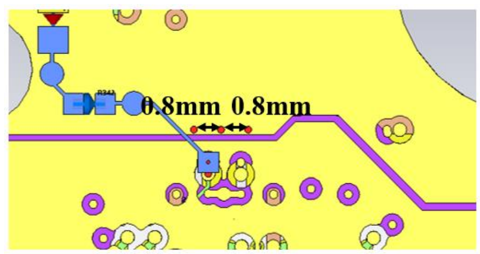


(b)

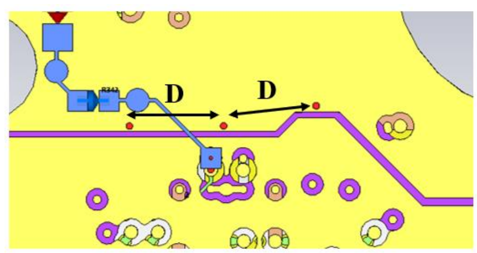
Fig. 14. TRP results of adding multiple GND vias close to signal transition. (a) TRP results from 100MHz to 2G. (b) TRP peak value with different via numbers.

Realistically, it's challenging to place so many GND vias as close to the signal transition as possible. Industrial product design usually places 3 GND vias close to signal transition via. Therefore, the location of GND vias should be designed to both satisfy the requirement of actual production and radiation suppression.

To investigate effects of GND via distance, we have placed 3 GND vias with different distance along the ground slot under the signal trace. Fig. 13(a) is the same case as Fig. 12(c). Fig. 15(b) illustrated that fixing the position of middle via and changing the distance of the other two vias at the same time. D is determined by the ratio of wavelength. For this board, to optimize the via distance up to 2 GHz, the wavelength of 2GHz is used, which is 150mm. The simulation results are shown in Fig. 16. When D increase, TRP results decrease first and then increase. The effect of radiated emission suppression is best when placing GND vias with $\lambda/37.5$ distance. Fig. 17 shows that adding 3 vias with $\lambda/37.5$ distance can have 9dB more noise reduction than 3 vias close to signal via. This is because adding via along the gap forms the short boundary of the cavities to reduce the parallel plate impedance and edge radiated emission. Besides, it works better than placing 6 vias close to signal transition, which reaches our expectation.

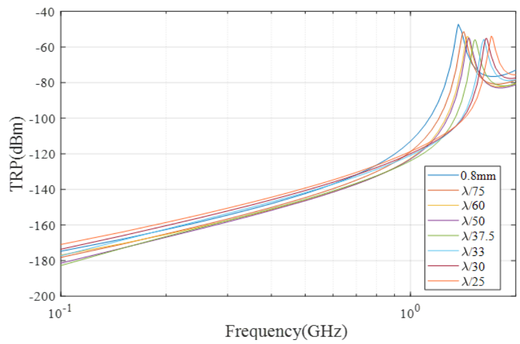


(a)

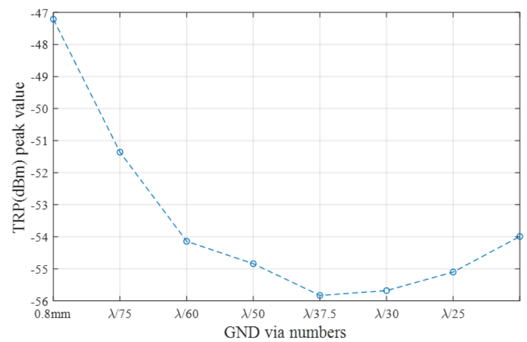


(b)

Fig. 15. Multiple GND vias placement. (a) Place 3 GND close to signal via. (b) Variable distance of 3 GND via along gap



(a)



(b)

Fig. 16. TRP results of adding multiple GND vias close to signal transition. (a) TRP results from 100MHz to 2G. (b) TRP peak value with different via numbers.

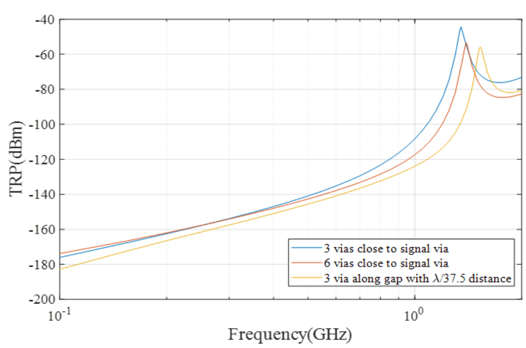


Fig. 17. TRP result of different placement of GND vias.

V. CONCLUSION

In this paper, we have investigated the effect of the ground slots on the radiated emission. Equivalent circuit diagram is built to explain the discontinuities in the return current path and thus the peak in total radiation power result. The radiated noise peak frequency can be explained and predicted from the proposed circuit diagram. At the resonant frequency, power plane(L4) can be the current return path and the noise voltage between power and ground plane results in strong plane edge radiation.

The employment of the GND vias effectively suppresses the signal distortion and the radiated emission. There should be a compromise between the design feasibility for manufacture and the guarantee of the return current path thus the number and the location of GND via should be optimized. For multiple vias placement, instead of placing GND via as close to the signal via as possible, vias should be spread along the gap with λ/37.5 distance. Three GND vias with λ/37.5 distance can suppress radiated emission by 22dB, that is 2 dB more than placing 6 GND vias close to signal transition.

REFERENCES

[1] J.S. Pak, J. Kim, H. Lee, J.-G. Byun and J. Kim, "Coupling of Through-Hole Signal Via to Power/Ground Resonance and Excitation of Edge Radiation in Multilayer PCB", Proc. IEEE 10th Int'l Symp. Electromagnetic Compatibility, vol. 1, pp. 231-235, Aug. 2003.

[2] J.-H. Lee, P.-S. Lee, T.-H. Lee, C. Kim, I.-C. Song and J.-K. Wee, "Effect of split power/ground planes using stitching capacitors on radiated emission", 2009 11th Electronics Packaging Technology Conference, pp. 541-545, 2009.

[3] Y. Ko, K. Ito, J. Kudo and T. Sudo, "Electromagnetic radiation properties of a printed circuit board with a slot in the ground plane", 1999 International Symposium on Electromagnetic Compatibility (IEEE Cat. No.99EX147), 1999.

[4] J. Kim, H. Lee and J. Kim, "Effects on signal integrity and radiated emission by split reference plane on high-speed multilayer printed circuit boards", IEEE Trans. Adv. Packag., vol. 28, no. 4, pp. 724-735, Nov. 2005.

[5] H. Y. Shim, J. Kim, J. G. Yook, "Modeling of ESD and EMI Problems in Split Multi-Layer Power Distribution Network," in Proc. IEEE Int. Symp. Electromagn. Compat., vol. 1, pp. 48-51, Aug. 2003.

[6] R. E. Collin, Field Theory of Guided Waves. New York: Wiley-IEEE Press, 1990, pp. 37–39