Fall 2022

ANALYSIS OF ELECTROMAGNETIC INTERFERENCE PROBLEMS CAUSEDBYHIGH-SPEED MULTIPLAYER BOARDS WITH SPLIT REFERENCEPLANE

Xin Fang
Missouri University of Science and Technology

ANALYSIS OF ELECTROMAGNETIC INTERFERENCE PROBLEMS CAUSED BY

HIGH-SPEED MULTIPLAYER BOARDS WITH SPLIT REFERENCE PLANE

by

XIN FANG

A THESIS

Presented to the Graduate Faculty of the

MISSOURI UNIVERSITY OF SCIENCE AND TECHNOLOGY

In Partial Fulfillment of the Requirements for the Degree

MASTER OF SCIENCE IN ELECTRICAL ENGINEERING

2022

Approved by:

Dr. Hwang Chulsoon, Advisor
Dr. Daryl Beetner
Dr. Donghyun Kim

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. 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. 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, we have investigated and analyzed the return current path radiated emission peak below 2 GHz for 6-layer camera PCB with a signal trace go across gap and change reference planes. According to the 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. Equivalent circuit diagrams are presented built to explain the current path in a practical camera device with the separated ground. Besides, 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.

ACKNOWLEDGMENTS

I'd like to express my full-hearted gratitude to my advisor Dr. Hwang Chulsoon for his support and guidance in the past years. He has changed my life towards a better path, helping me build knowledge and skill in the electromagnetic compatibility field. I'm grateful for his advice on the Samsung and Sony project, with which I learned much and let me be qualified for a job.

I would also like to thank Dr. Daryl Beetner and Dr. Donghyun Kim for being the committee members and advising on the thesis. From Interference Control course of Dr. Daryl Beetner, I learned what is EMC. From RF measurement course of Dr. Donghyun Kim, I learned about different instruments and test methods.

The EMC lab is a great family, I would like to thank all EMC lab faculties and students for all their help. For students at EMC lab, I want to say thank everyone and it has been a great opportunity to know each other and hope to meet everyone again.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

1. INTRODUCTION

1.1. BACKGROUND

With an increasing demand for electric vehicles and autonomous driving, many components of a vehicle are being replaced as electric components. Camera sensing for autonomous driving is not only possible but preferred, allowing the car to see 360 degrees. This implicates the possibility of an increase in the malfunction of car camera.

For the camera board level design, 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.

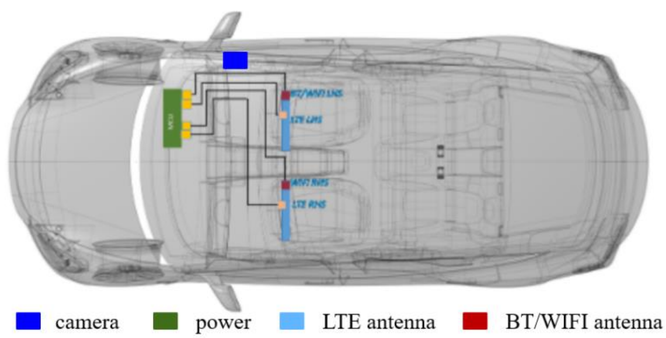
While [2], [3] include microstrip lines over a split ground plane, only 2-layer boards were considered. [4] has introduced the idea of adding the stitching capacitor across the gap to reduce radiated emission, but it would add another noise peak at certain frequency. Though [5] pointed out return current 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.

1.2. CONTENTS

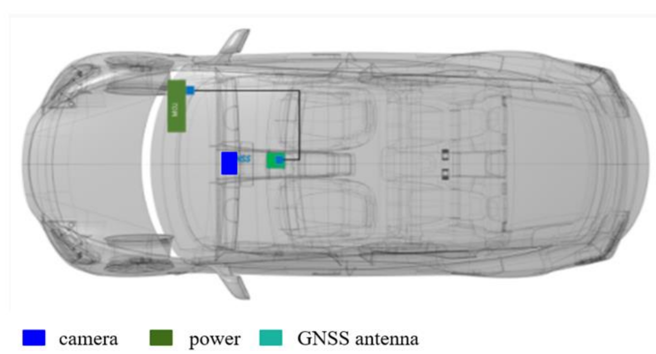
In this thesis, we have investigated and analyzed the radiated emission peak below 2 GHz for 6-layer camera PCB with a signal trace go across gap and change reference planes. According to the board structure, the power plane is between two ground planes, which can result in the high impedance current 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 three 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 S parameter 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, the simulation results shows that the stitching via degrades the noise isolation between the planes and modifies the resonance frequencies of the planes.

2. PROBLEM DESCRIPTION

There are eight cameras around car body to capture photos in 360 degree and ensure autopilot working properly. From Figure 2.1, the locations of camera are close to the antenna and harness. Therefore, it's important to control the radiated emission of camera to ensure the antenna function as intended and pass the EMC testing in vehicle level below 1GHz. In this paper, for research purpose, we'd like to extend the frequency to 2GHz.



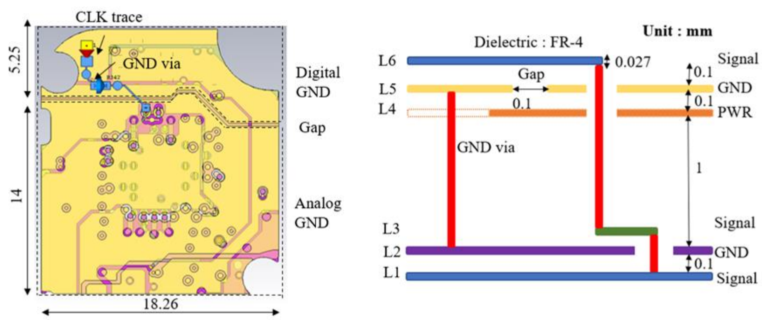
(a)



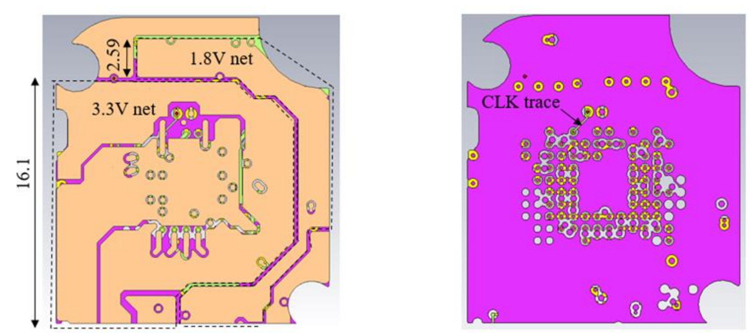
(b)

Figure 2.1 Configuration of camera harness and antenna locations. a) Antenna on the side. b) Antenna in the front.

Figure. 2.2 shows one of a 3D model of the camera board under investigation. The board is a 6 layer PCB with dimensions of 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 ground and analog ground in L5 are separated. L4 is a sliced power plane with different power net and L2 is a solid digital ground.



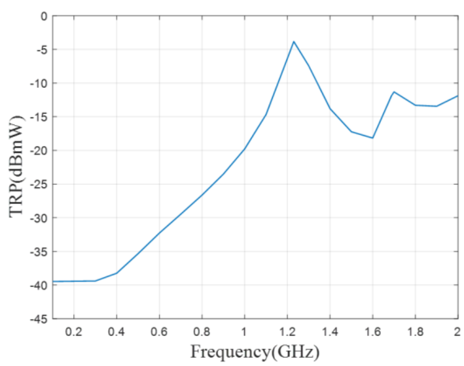
(a) (b)



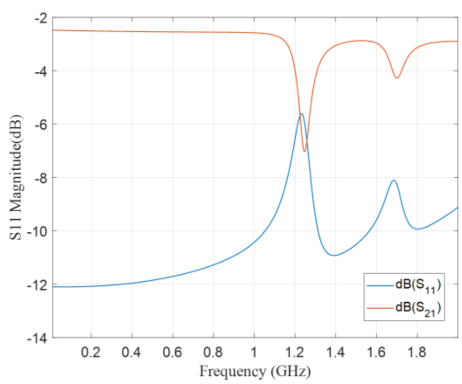
(c) (d)

Figure 2.2 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.

In this system, it was observed that there was two peaks at 1.23GHz and 1.688GHz in TRP below 2GHz when exciting the CLK signal trace, shown in Figure. 2.3(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 Figure. 2.3(b) show large reflected wave at 1.23GHz.



(a)



(b)

Figure 2.3 Simulation result of the board to be investigated. a) Total radiated power over frequency. b) S parameter result of CLK trace

Besides, in Figure. 2.4, 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.

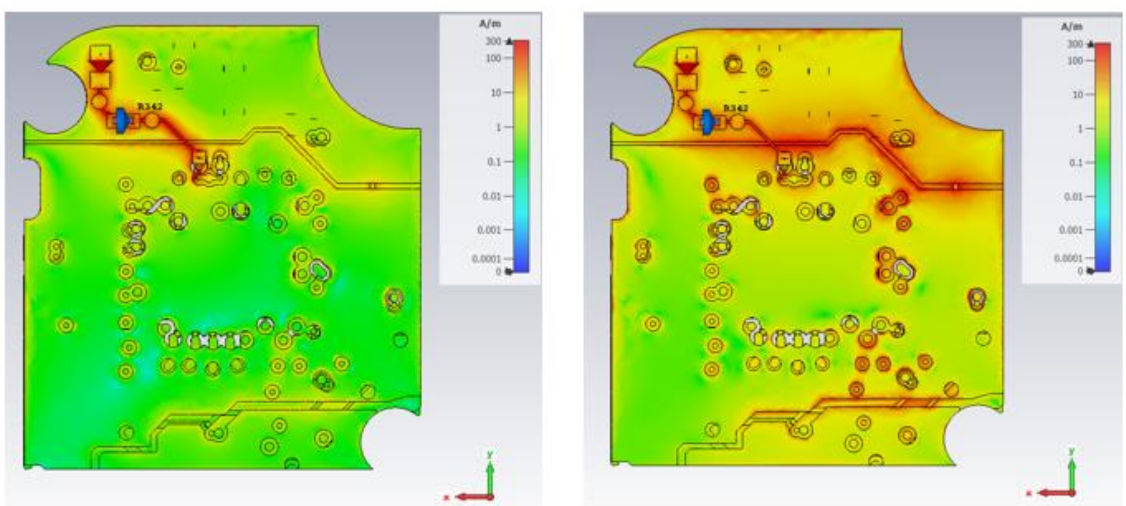
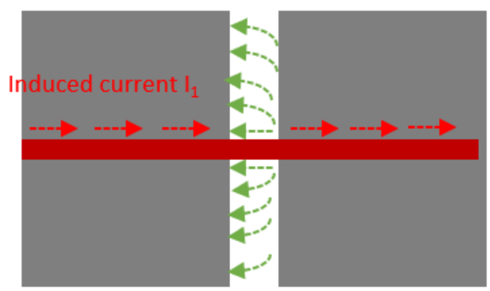


Figure 2.4 Surface current distribution of different frequency. a) At 900 MHz. b) At 1230 MHz.

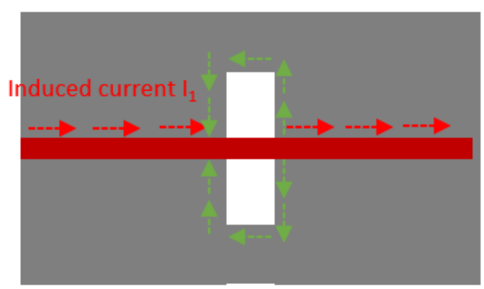
3. PROBLEM ANALYSIS

3.1. PROBLEM 1 : SPLITTED ANALOG AND DIGITAL GND

From the top view of L5 in Figure 2.2, the digital and analog GND plane in L5 is separated by a slot. When the CLK signal is transmitted from digital GND to analog GND, the splits or slots between the reference GND cause a disruption in the return current. From Figure 3.1, there is displacement current between the splits. Current have to return from longer path, incomplete GND plane can result in radiated emissions increasing.



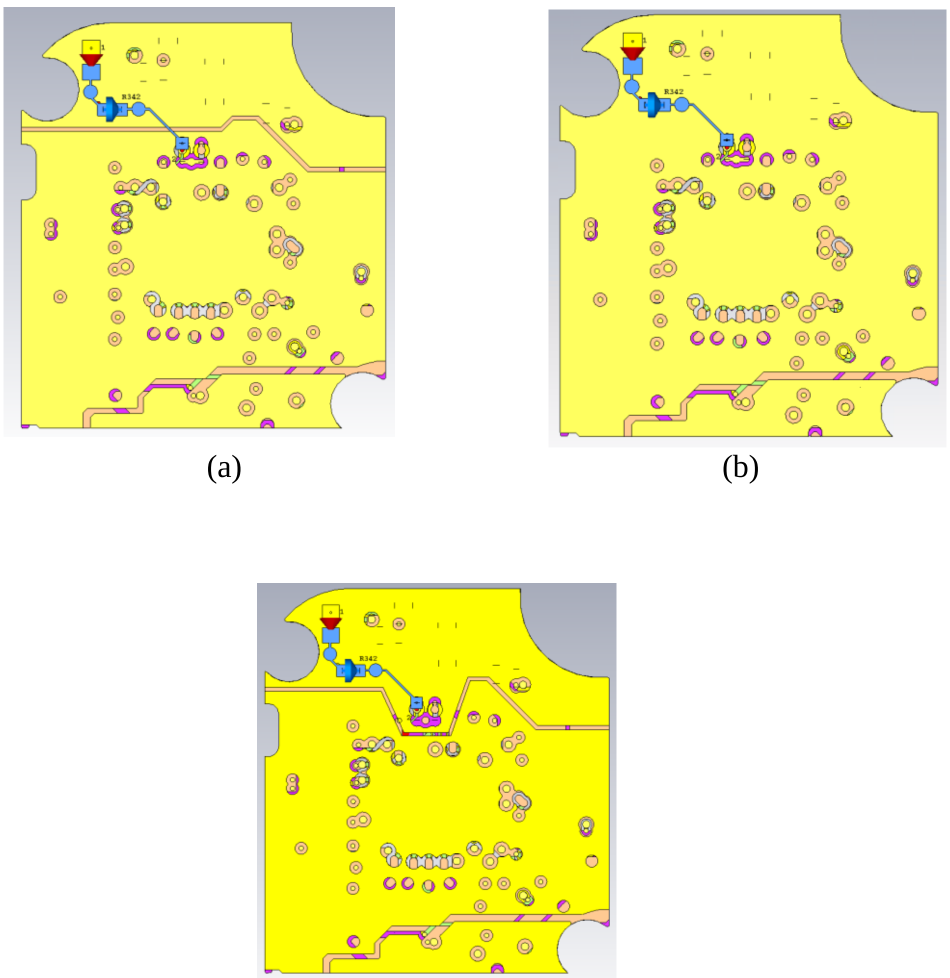
(a)



(b)

Figure 3.1 Return Current path. a) Through the splits. b) Through the slot

To figure out whether GND slot is the factor dominating the radiated emission peak, three cases are simulated in full-wave simulation shown as Figure 3.2. Figure 3.2(a) is the original board. Figure 3.2(b) shows the board without gap in L5, and Figure 3.2(c) describes the board changing the gap to avoid reference plane discontinuity.

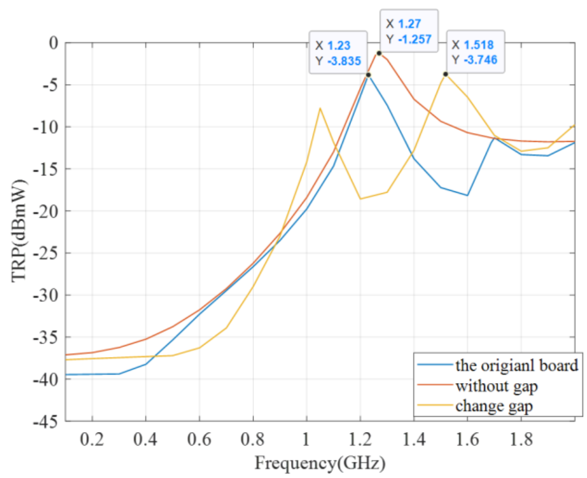


(a) (b)

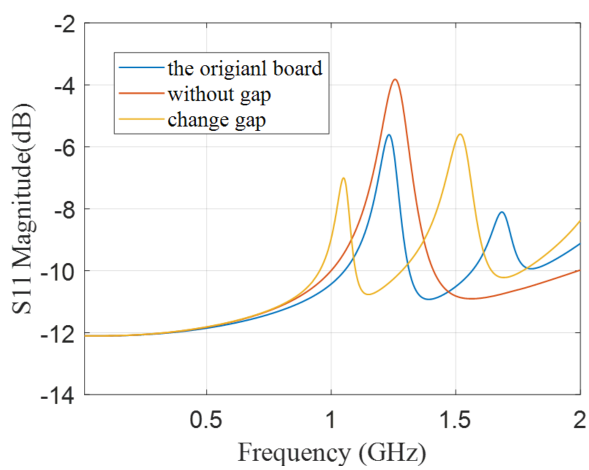
(c)

Figure 3.2 Top view of L5. a) The original board. b) Without gap in L5. c) Change gap to avoid reference plane discontinuity.

The result comparison of three cases are shown in Figure 3.3. The results from TRP curves and S parameters show that even though there is no GND slot under signal trace in L5, a peak still can be seen in TRP result and changing the slot can not damp the peak. Although the GND slot can influence the peak frequency, it is not the main factor that causing the peak in radiated emissions.



(a)



(b)

Figure 3.3 Result comparison for different GND slot. a) TRP. b) S11 magnitude

3.2. PROBLEM 2 : CLK SIGNAL CHANGING LAYERS

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 (Figure 3.4).

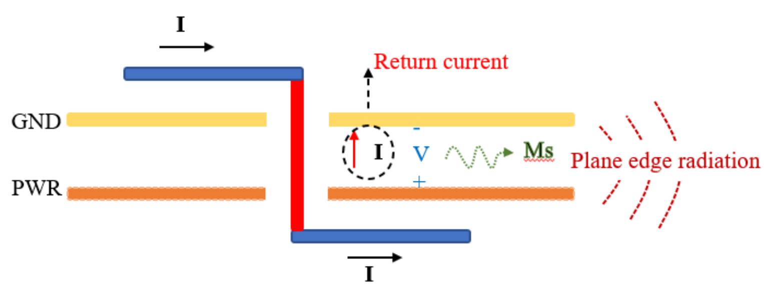
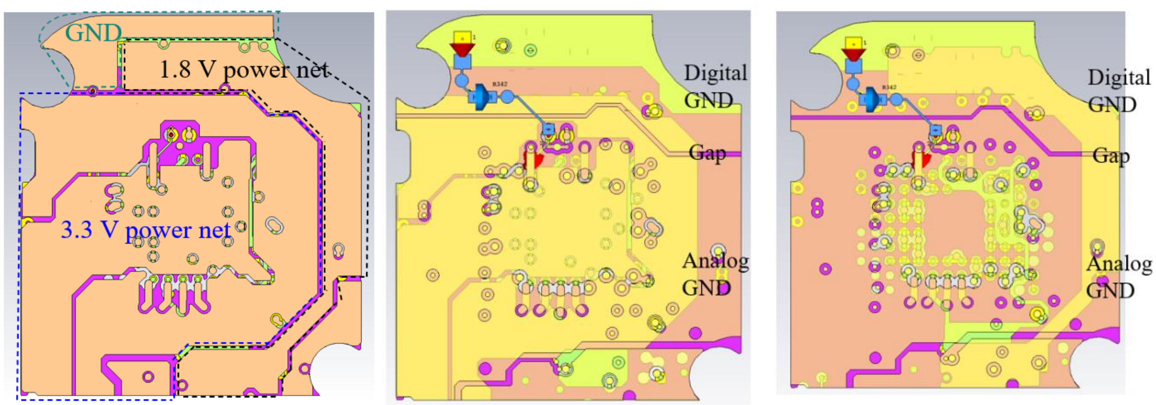


Figure 3.4 Power/ground plane edge radiation excited by clock

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 Figure 3.4 is the product of return current and power/ground plane impedance.

The camera board has a power plane L4 between L2 and L5 ground planes, as shown in Figure 3.5. There are 1.8 V and 3.3V power net in L4. As illustrated in Figure 3.4, high parallel plate 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.



(a) (b) (c)

Figure 3.5 Power net in L4. a) Top view of L4. b) 3.3V power net. c) 1.8V power net.

3.2.1. Independent Networks for 1.8V Power Net. When only with 1.8V power net in L4, the 6-layer PCB has independent plane pair networks as Figure 3.5, 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. Figure. 3.5(b) and Figure. 3.5(c) show the different independent networks as L5 is split as digital ground and analog ground. As shown in Figure.3.5(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 Figure. 3.5(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.

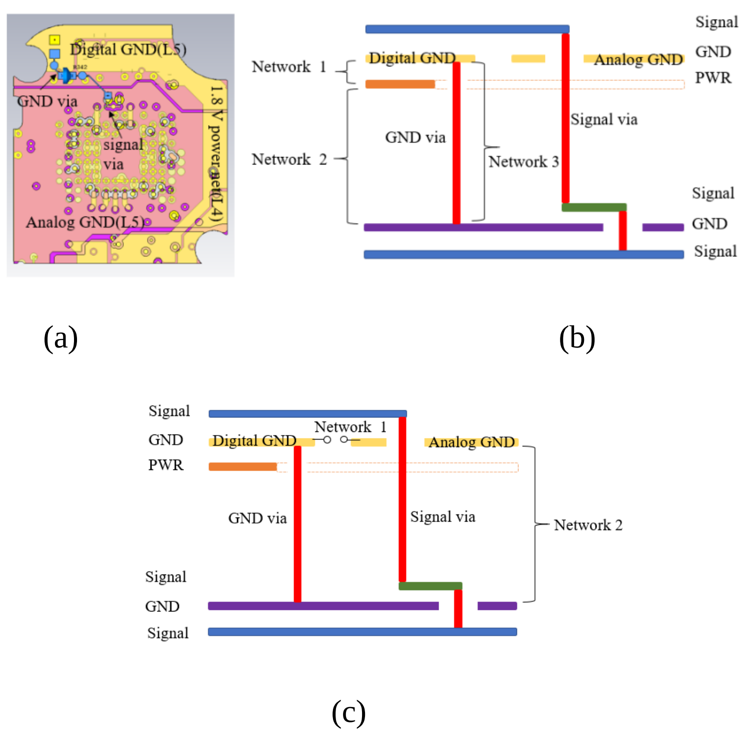


Figure 3.6 Independent networks with 1.8V power net. a) Top view of board with 1.8V power net. b) Independent networks of case 1. c) Independent networks of case 2.

Figure 3.7 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 port to replace signal via to get parallel plane impedance by simulation. 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.

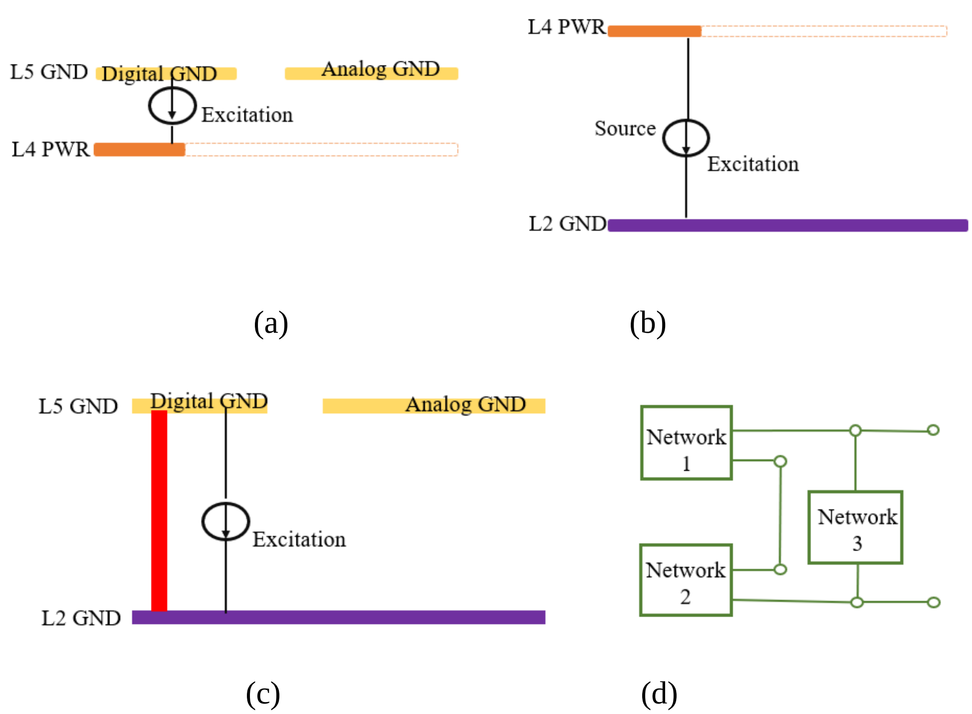


Figure 3.7 Independent networks of case 1. a) Network 1. b) Network 2. c) Network 3. d) Equivalent circuit diagram.

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 Figure.3.8. 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 Figure.3.8(c).

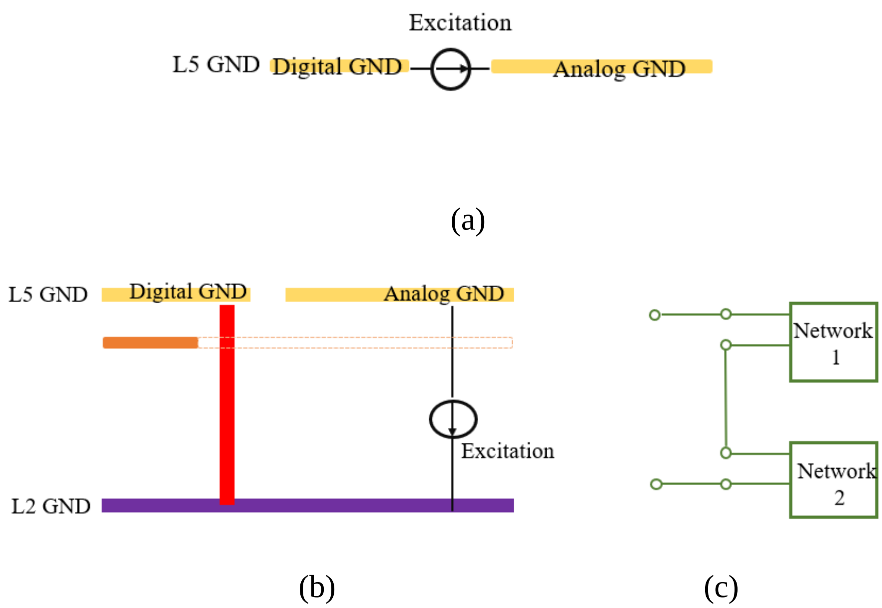
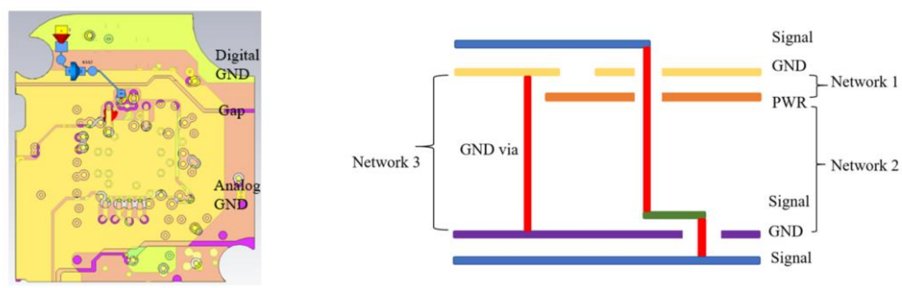


Figure 3.8 Independent networks of case 2. a) Network 1. b) Network 2. c) Equivalent circuit diagram.

3.2.2. Independent Networks for 3.3V Power Net. When only with 3.3V power net in L4, the 6-layer PCB has independent plane pair networks as Figure 3.9. The return current flowing along L5, the GND via and going back to L5 digital GND. In Figure 3.10, 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 from the equivalent circuit diagram.



(a) (b)

Figure 3.9 Overview of 3.3V power net. a) Top view of board with 3.3V power net. b) Stackup.

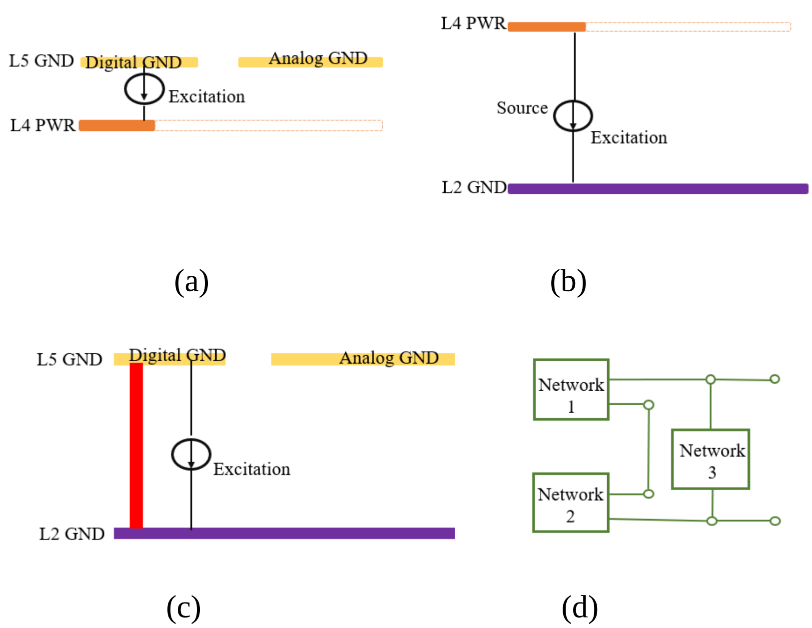
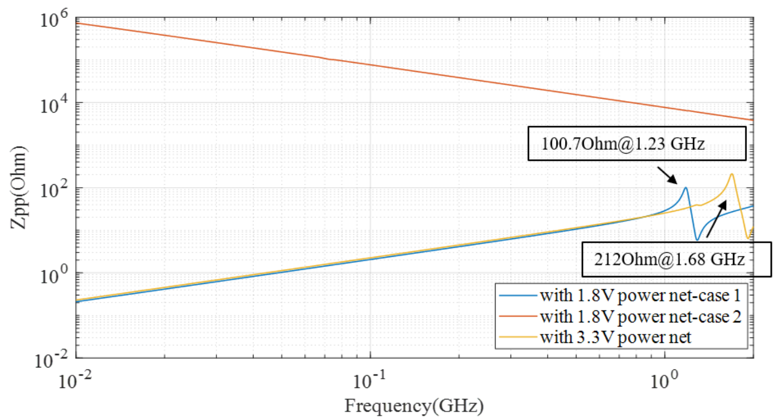
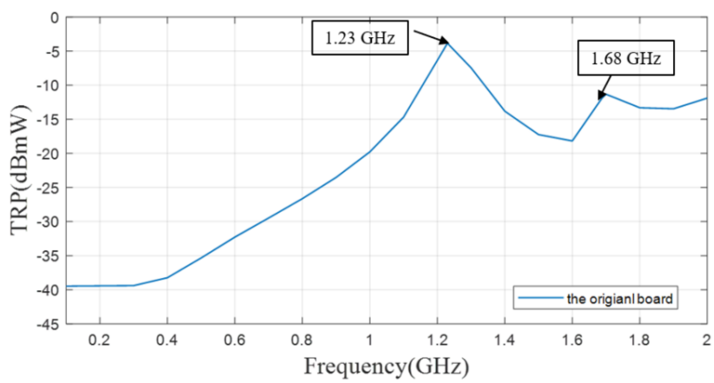


Figure 3.10 Independent networks of 3.3V power net. a) Network 1. b) Network 2. c) Network 3. d) Equivalent circuit diagram.

3.2.3. Simulation Result Comparison. In Figure 3.11, the total parallel plane impedance(Zpp) calculated from three models are compared. For 1.8V power net, parallel impedance of case 1 is much lower than the that of case 2. It can be analyzed that current prefers case 1 to return. When comparing the impedance curve along frequency, we can observe a peak in 1230MHz with 1.8V power plane and a peak in 1688MHz with 3.3 V power net , which corresponds well with the resonant frequency shown in TRP.



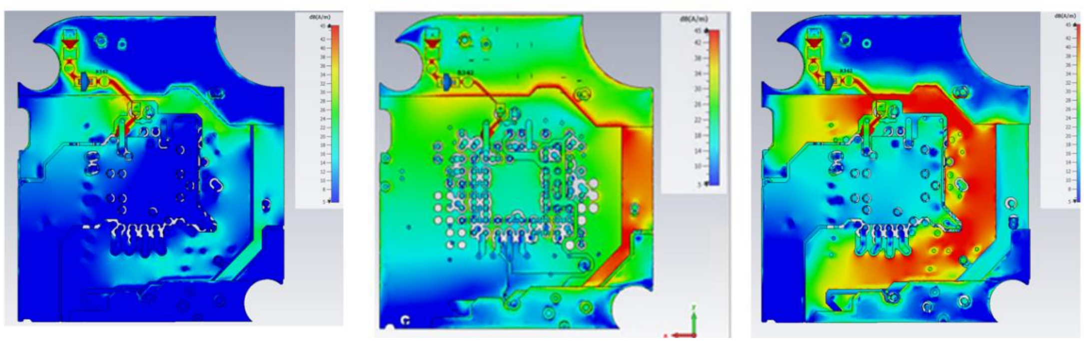
(a)



(b)

Figure 3.11 Simulation result of the board to be investigated. a) Total Zpp comparison along different current path. b) TRP of the original board.

Figure 3.12 shows the surface current comparison of the original board at different frequency. At the resonant frequency, instead of returning from ground via, current went back to source from nearby plane coupling. current flowed from digital ground(L2) to power plane(L4) coupling and power plane(L4) to digital ground(L2) coupling. Strong plane edge radiation can be observed because of the larger plane to plane voltage drop, which corresponds well with the result in Figure 2.3. Power plane is used as return path as illustrate in Figure 3.12.

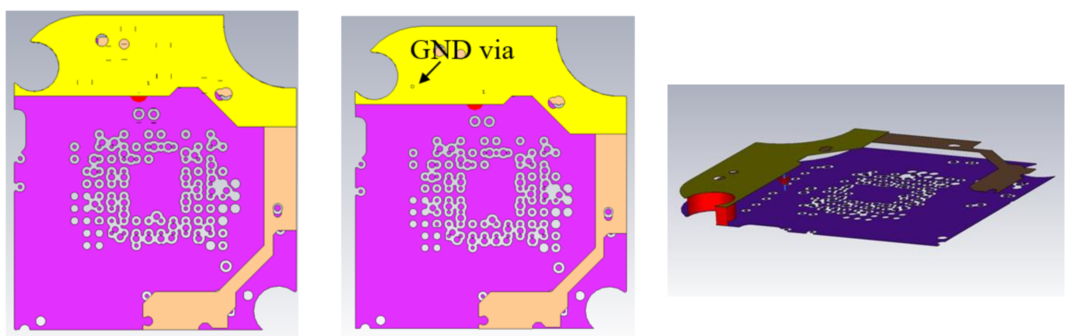


(a) (b) (c)

Figure 3.12 Surface current distribution of different frequency. a) At 900 MHz. b) At 1230 MHz. c)At 1688MHz.

3.2.4. Effect of GND Via on Parallel Plane Resonant Frequency. Three cases are built to investigate the effect of GND vias on changing the resonant frequency. To reduce the calculation difficulty, for these three cases, only L5 digital GND, L4 1.8V power plane and L2 GND are kept. Excitation is between L2 and L5. The location of port is close to the signal via. Zpp results are compared.

Figure 3.13 and Figure 3.14 shows the result comparison. By comparing the blue curve and red curve, adding GND via can shift the resonant peak to lower frequency. The peak frequency of red curve and yellow curve are almost the same, which means that adding the GND via can be regarded as adding PEC GND at the boundary. Wave travels between L5 and L4, and returns between L4 and L2 in opposite directions. Thus the dimension along this direction should be calculated twice.



(a) (b) (c)

Figure 3.13 Three cases to be simulated. a) No GND via. b) With GND via. c)With PEC sheet on the side.

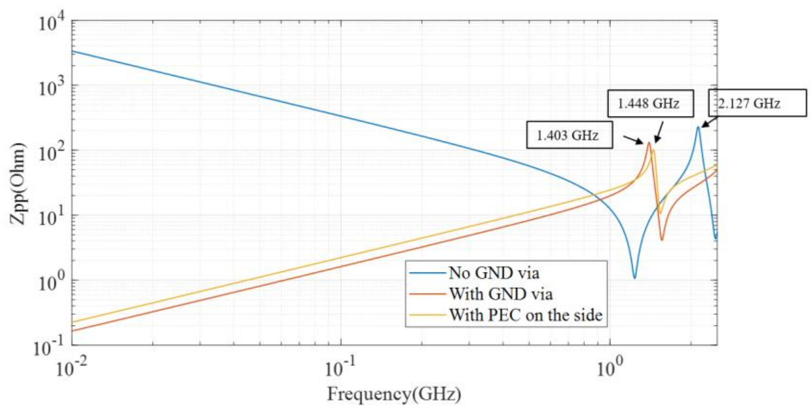


Figure 3.14 Total Zpp comparison for three cases

4. IMPROVEMENTS

When signal trace change reference planes, adding ground vias between two GND planes can suppress the radiated emission because GND vias provide return path for current. Multiple ground vias can reduce the impedance of return current path. Numbers and locations of the ground via should be taken into consideration.

Figure 4.1 shows adding different numbers of GND vias close to signal via. Deleting the Original GND via and adding GND vias as close to the signal via as possible. The distance between GND vias is 0.8mm and the radius of anti-pad is 0.32mm.

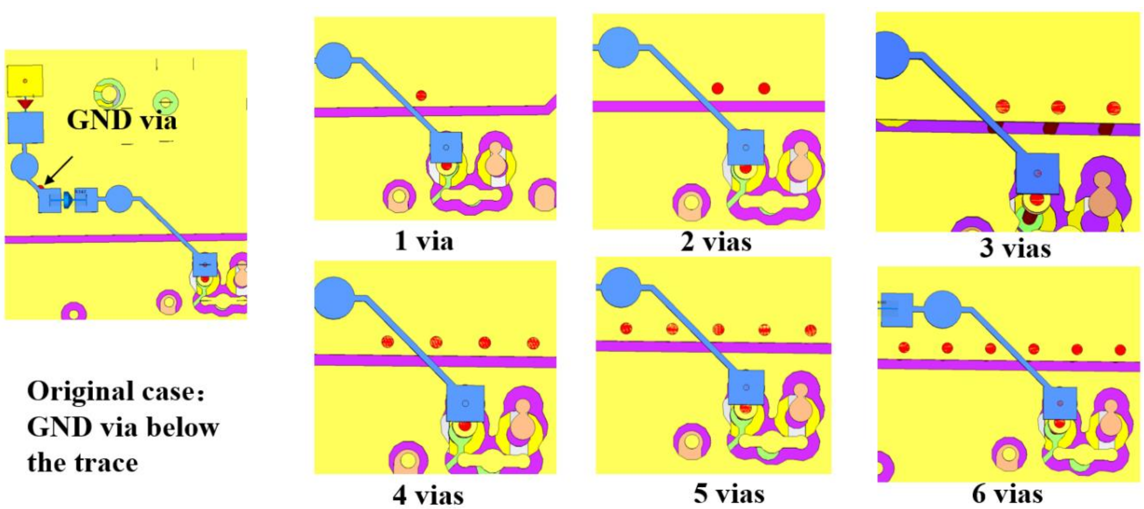
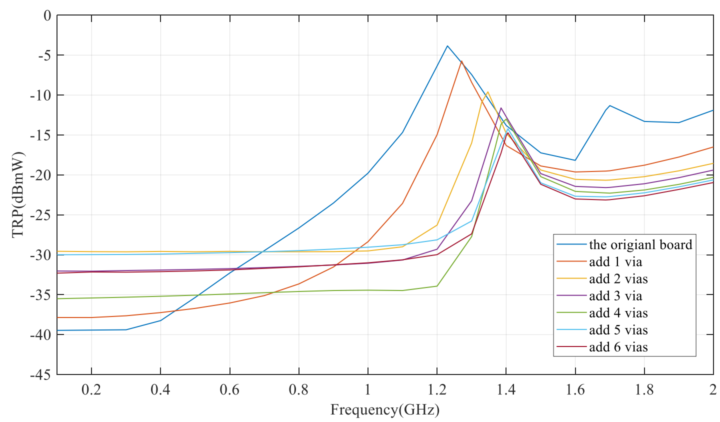


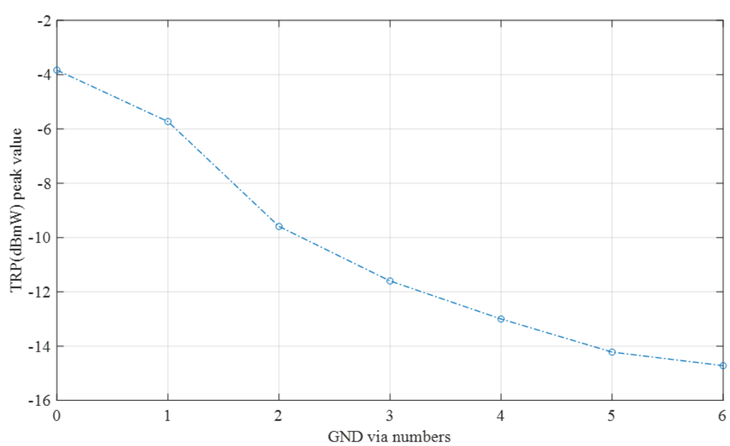
Figure 4.1 Different number of GND vias close to the signal via.

Via locations and numbers are investigated in this case. Adding GND vias as close to the signal via as possible provides the least impedance path for current to return.

Figure 4.2 illustrates that when adding one GND via, via should put as close to the signal via as possible. 3 GND vias can reduce around 8dBm noise at the resonant peak comparing to original case. When adding more than 3 vias, the reduction of field emission is similar to that of adding three GND vias. Adding vias close to the signal via can damp the resonant peak but can not eliminate the peak.



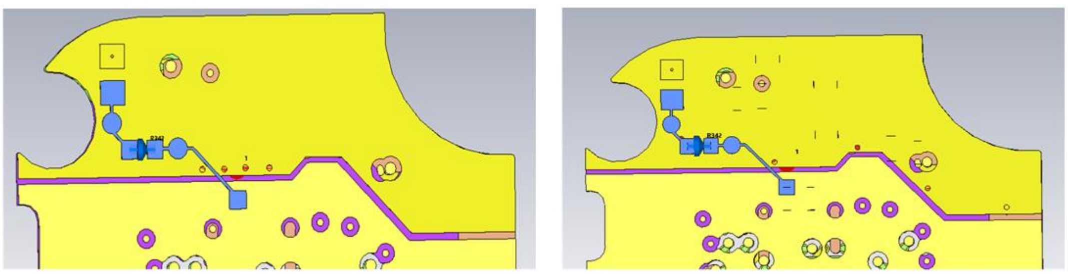
(a)



(b)

Figure 4.2 TRP results for multiple GND vias. a) TRP along frequency. b) TRP peak value.

From investigation, we found that the placement of multiple vias will also influence the radiation peak value at the resonant frequency. In Figure 4.3, case 1 is adding four ground vias as close to the signal via as possible, while case 2 spreads four ground vias along the gap. Figure 4.4 is the TRP comparison to the original board, there is no peak at the TRP curve when the three GND vias spread along the gap. The radiated power decreases 30 dBm at resonant frequency.



(a) (b)

Figure 4.3 GND via locations. a) four vias close to signal via. b) Four vias along the slot.

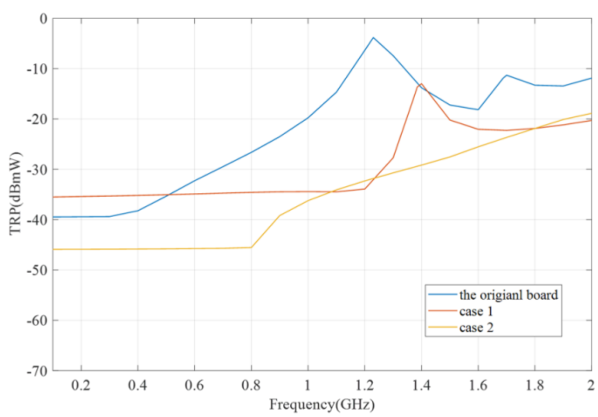
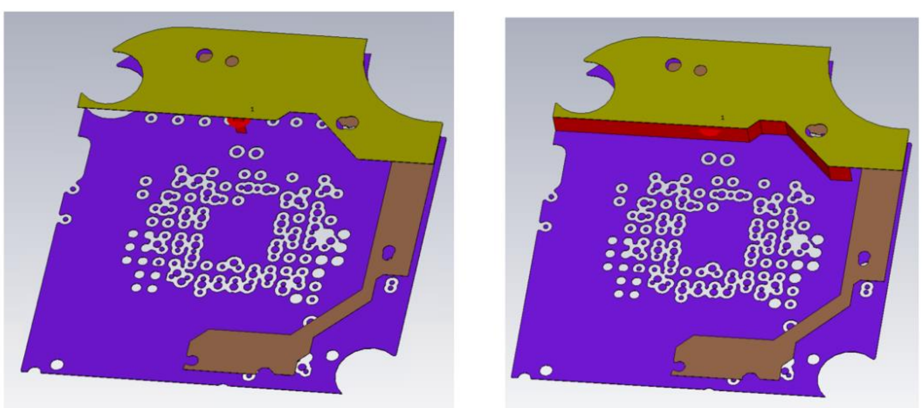


Figure 4.4 TRP comparison for via locations.

Two cases are built to investigate the reason why spread vias along the gap have no peak comparing with GND vias close to the signal via. Figure 4.5(a) is adding small PEC sheet close to excitation, while Figure 4.5(b) is adding PEC sheet along the gap. The distance between excitation and PEC sheet is 0.4 mm. Figure 4.6 shows the Zpp result. There is a peak for small PEC sheet close to excitation but no parallel resonance for PEC sheet along the gap. The result shows that adding GND vias along the slot can be seen as PEC boundary close to source, which can block input current invading into the cavity. If the GND vias are only located close to the signal via, current still can go to cavity from open area.



(a) (b)

Figure 4.5 Add PEC close to excitation. a) Small PEC close to excitation. b) PEC along the slot.

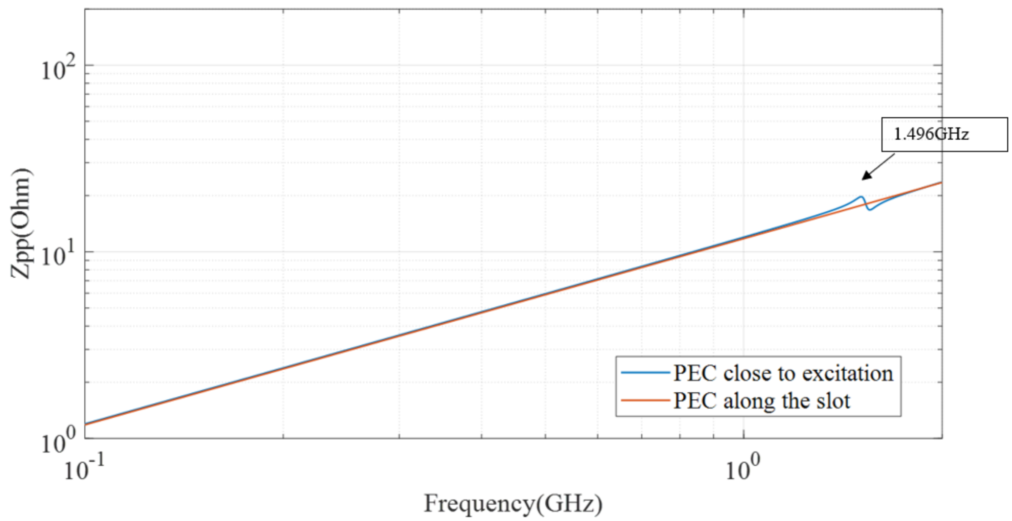


Figure 4.6 Zpp for adding the PEC sheet

5. CONCLUSION

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 got from the proposed circuit diagram. At the resonant frequency, power plane can be the current return path and the noise voltage between power and ground plane results in strong plane edge radiation. The radiated emission generated by the discontinuity of the return current path on multiplayer board with split ground can be reduced by adding stitching vias along the gap to provide short boundaries for power/ground plane cavities. For 1 GND via, the via should put as close to the signal via as possible. For multiple via placement, vias should spread along the gap. At least 3 vias are needed to remove the radiation resonance below 2GHz.

REFERENCES

[1] Pak, Jun & Kim, Jingook & Byun, Jung-Gun & Lee, Heejae & Kim, Joungho. (2003). Coupling of through-hole signal via to power/ground resonance and excitation of edge radiation in multi-layer PCB. IEEE International Symposium on Electromagnetic Compatibility. 1. 231 - 235 vol.1. 10.1109/ISEMC.2003.1236597.

[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, 2009, pp. 541-545, doi: 10.1109/EPTC.2009.5416488.

[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, pp. 576-579, doi: 10.1109/ELMAGC.1999.801393.

[4] Jingook Kim, Heeseok Lee and Joungho Kim, "Effects on signal integrity and radiated emission by split reference plane on high-speed multilayer printed circuit boards," in IEEE Transactions on Advanced Packaging, vol. 28, no. 4, pp. 724-735, Nov. 2005, doi: 10.1109/TADVP.2005.850503.

[5] Hwang-Yoon Shim, Jiseong Kim and Jong-Gwan Yook, "Modeling of ESD and EMI problems in split multi-layer power distribution network," 2003 IEEE Symposium on Electromagnetic Compatibility. Symposium Record (Cat. No.03CH37446), 2003, pp. 48-51 vol.1, doi: 10.1109/ISEMC.2003.1236562.

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

VITA

Xin Fang received her Bachelor of Science degree in Electronic Science and Technology from Zhejiang, China, in June 2019. She received her Master of Science in Electrical Engineering from the Missouri University of Science and Technology, Rolla, MO, USA, in December 2022. Her research interests included RF interference, and EMI modeling.