Analysis of Electromagnetic Interference Reduction from Printed Circuit Boards

Qixiong Liang

Physics, University of California Santa Barbara, 93106, United States

qixiong@ucsb.edu

Abstract. Nowadays, electricity is the most important and necessary energy in human society and daily life, because electricity is convenient and safe to transfer between different locations. However, electromagnetic interference (EMI) always accompanies electricity and has brought a lot of problems. Especially in printed circuit boards (PCB), the EMI can break the circuit or cause a short circuit, which is dangerous in creating a PCB. To effectively reduce EMI from PCB, three different methods of reducing EMI from PCB were chosen to compare with each other. By comparing their price, practicability, effectiveness, data, and technology for reducing EMI from PCB, PCB designers can choose the methods that are suitable for their situation.

Keywords: electromagnetic interference (EMI), printed circuit boards (PCB).

1. Introduction

Electricity is the most important and necessary energy in human society and daily life. Without the energy form of electricity, it is difficult for human beings to live in nowadays society. Although electricity is convenient for human daily life and helps to create a comfortable environment to live, electrothermal effect and electromagnetic interference is inevitable when electricity was used. Especially for electromagnetic interference, if any method to reduce this effect cannot be found, most electric appliances will finally be broken by electromagnetic interference. Their performance will degrade because of electromagnetic interference and at the same time, they will interfere with other tools driven by electricity. Therefore, it is important to find a way that can help designers to reduce electromagnetic interference, especially in printed circuit boards, because electromagnetic interference can generate short circuit in printed circuit boards which will destroy the whole printed circuit board. Also, all electronic products cannot be without a printed circuit board [1]. To reduce the EMI in a different electronic device, it is necessary to consider how to reduce the EMI in the PCB of each electronic device and it is the only way to reduce the EMI for the whole electronic device and make it to be electromagnetically compatible [2]. Digital circuits with a high range of frequency, analog circuits, and clock circuits are contained in PCB boards [3]. When the clock circuit increase, it will increase the circuit's potential to radiate, which finally increase the EMI and damage the security of PCB [4,5].

Although PCB designers do have different methods to reduce electromagnetic interference, the most effective methods were not found to attain their goals. Hence, in this essay, three different methods of reducing electromagnetic interference in printed circuit boards were be found and will be comparedwith each other to find advantages, such as the advantage of their price, practicability, effectiveness, data, and technology of each method. Three different methods will be described and compared. As a result, in a different range of frequencies, this paper gives suggestions for people to choose from three different methods, which will help people who want to build their printed circuit board with low electromagnetic interference or help them to find the methods that most satisfy their requirements.

2. Electromagnetic interference radiations from the PCB traces

This method effectively reduces the PCB in EMI about the switched mode power supplies (SMPS), which have less weight and great efficiency in designing the PCB. However, using SMPS also will bring a lot of electromagnetic interference even greater than another design method. Therefore, it is important to use this method to decide on the great design of lengths of the trace of printed circuit boards to be electromagnetically compatible. This result of this method can help them to choose the most suitable length of trace to decrease electromagnetic interference. SMPS makes PCB produce drive signals, which makes PCB have a lot of current flowing with high frequencies and voltages. Because every electronic device is composed of PCB, it is important to think about the strength of electromagnetic interference in each PCB design. To reduce the EMI in PCB design, it is available to consider the electromagnetic compatibility for PCB design. However, by measuring the radiations from PCBs, which are different in different devices, it is more efficient to reduce EMI from PCBs. When a digital design's speed becomes faster, an emission of PCB will at the same time be increased. This method thinks of SMPS as the dipole. Apart from that, FR4 represents its dielectric substrate. During the range of frequency of about 30 MHz to 500 MHz, the PCB's emissions are analyzed. Different lengths of PCB traces in an electric field intensity have been analyzed in dielectric substance of FR4 according to a mathematical model in Table 1 and using the CST studio software tool in Figure 1.

Table 1. PCB Trace Analysis [6].

	Frequencies in MHz									
PCB Trace(m)	**50**	**100**	**150**	**200**	**250**	**300**	**350**	**400**	**450**	**500**
0.05	51	57	61	63	65	67	68	70	71	71
0.13	60	66	69	72	74	75	77	78	79	80
0.21	64	70	73	76	78	79	81	82	83	84
0.3	67	73	77	79	81	83	84	85	86	87

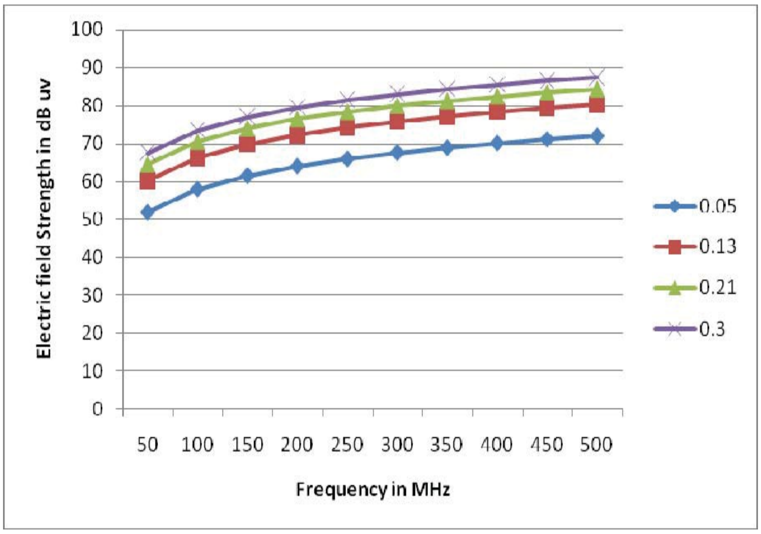

Figure 1. Calculated results of radiation from the PCBs 50 to 500 MHz [6].

This method considers the width of PCB from 5 cm to 30 cm. When the frequency ranges from 30 MHz to 500 MHz, by shortening the trace length of PCB, EMI will be reduced. Apart from that, the lower the frequency, the less EMI will emit. However, the calculation also presents that when the frequency is above 500 MHz radiation emission, the strength of EMI emission will only depend on the resonance mechanism, which means changing the trace length of PCB cannot influence the strength of EMI emission [6].

3. EMI reduction from PCB using electromagnetic bandgap structures (EBG)

This method is effective for suppressing PCB radiation from their power bus over an ultrawide range of frequencies by using metallo-dielectric electromagnetic band-gap structures, such as simultaneous switching noise, in other words, Delta-I noise or ground bounce. This noise is composed of excess voltage fluctuations on the PCB power bus about the parallel plate's resonance waveguide library generated by the bus of the power plane. Most importantly, this method has been testified by laboratory PCB prototypes, which support the efficiency of this method. Although in the practice, one of the methods that are usually be used is via stitching and its intensity has been supported by rigorous finite-difference time-domain (FDTD) simulation as same as the report in [7], employing discrete capacitor component has a base restriction because of the inherent inductance emerge from capacitors' leads. Apart from that, the reliability considerations and the high cost of embedded capacitance reduce its practical use. By using EBG or high-impedance surfaces (HIS), we can decrease power buses' radiation of PCB. HIS component is a periodic component, which can prevent the diffuse of electromagnetic surface waves in arrange of frequency. A new method of using the HIS component to EMI was described in this method to reduce the strength of the EMI in PCBs. This method describes that for PCBs with more than one layer, the surfaces with strong impedance in different power planes can effectively protect the board's edges and hence decline such edges of radiation. Also, the HIS construction not only can suppress the radiation of EMI from power busses but also can be used in signal layers to suppress radiation as well. The HIS construction also allows the designer to reduce the EMI in the range of frequencies that are higher than 0.5 GHz. This is one of the advantages of this method because it is seldom to effectively reduce the EMI in the range of frequencies that are bigger than 0.5 GHz. However, both the simulation results and experimental results show that this method can effectively reduce the EMI in a range higher than 0.5 GHz. The paper that illustrates this method also gives different examples to show the effective range of frequency in different radiation suppression methods, highlighting that even not complicated HIS structure can solve the problem in the 0.5-10 GHz range, which is beyond the limits of other methods. Apart from that, it is useful to apply the HIS construction to the range of frequency that is lower than 0.5 GHz, which is the range of frequency that other methods applied. However, there is a defect in applying HIS construction in the relatively low-frequency range. The defect is that the PCB designers have to consider the complex structure of HIS construction to build an effective PCB to suppress the noise and EMI in the range of frequency that is lower than 0.5 GHz [8].

4. Enhancement of PCB Design for Radiated Immunity Compliance in Analog Signal Measurement

It is the PCB that allows designers to minimize both the price and the size to build electronic devices with high efficiency and low price and at the same time reduce electromagnetic interference. It is easy for electromagnetic interference to influence the electronic device with signals of low power. When the operational amplifier is influenced by the noise of electromagnetic interference, it will be broken in high probability. This method pays more attention to improving the design of PCB in the region of the operational amplifier to attain the goal is immune to the radiation of electromagnetic interference. This method was invented from an accident of the operational amplifier with little signal amplifier in the test of Radiated Susceptibility. To solve this problem, the method was proposed to improve the design of PCB. Finally, this method successfully realized the interference immunity of the operational amplifier segment. This method was made from three different ideas that are shielding, filtering, and grounding, which can reduce the power of EMI equally. Considering the construction of the electronic device which is composed of polycarbonate, it is impossible to use the idea of shielding to reduce the radiation of EMI because of the particularity of this material. It cannot be wrapped without any gap [9]. Therefore, this method pays attention to solving the problem by using the idea of grounding to reduce the impedance of the ground. Designing the PCB in the order of multilayer, allows PCB to gain the advantage of board capacitance, which is composed of the planes of substrate, power and ground that act to be a dielectric. Apart from that, this method also points out that changing the operational amplifier output to shorten than before between the ADC lines and the operational amplifier output helps to reduce the EMI because it can be ensured that they do not in the role of the antenna to radiated waves. Finally, going through all of this improvement, the PCB displays a clean and clear road throughout the range of frequency from 80 MHz to 2.7GHz. This method focuses on the achievement of compliance with radiation immunity. Also, this method points out the significance of realizing the ground planes and the length of the trace, which both influences the design of the PCB. Apart from that, this method did not put any change into the circuit, which is difficult to generate a general theory of reducing the EMI in PCB. Under this circumstance, this method successfully improved the PCB design to fit the standard of Radiated Immunity test. The device of PCB that has problems in the range of frequency of 100MHz to 230 MHz (continues error in current was measured) and 250 MHz to 560 MHz at different frequencies in bands, where greater than 15% to 18% in the current reading, had improved by using this method. As a result, this method fixed all of the problems that this device had before [10].

5. Comparison of different methods that reduce electromagnetic interference on PCB

By comparing three different methods of reducing electromagnetic interference on PCB, the suitable method will be emersed for the different situations of reducing the electromagnetic interference on PCB. The first method shortens the length of the PCB trace to reduce EMI on PCB. However, this method can only apply to the range of frequency that is lower or equal to 500 MHz. The method will lose its efficacy in the range of frequency that is higher than 500 MHz because it will emerge a resonance mechanism in this range of frequency. Although there is a limitation to the first method, it is the first method that allows PCB designers to reduce the EMI with low complexity and price. In the same range of frequency, it is relatively difficult for PCB designers to reduce the EMI by using the second method because they have to design a complex HIS construction to effectively reduce the EMI in a range of frequencies that is lower than 500 MHz. The third method is effective in the range of frequency from 80 MHz to 2.7GHz, which not only shortens the circuit of CB trace, but using the multilayer board to reduce the EMI in PCB. This method is also convenient for PCB designers to use. However, the third method did not show any data in the range from 0 MHz to 80 MHz. Therefore, it cannot guarantee that this method is effective in this range. Hence, in the frequency range of 0 MHz to 80 MHz, it is effective to use the first method. In the frequency range of 80 MHz to 500 MHz, it is reliable to use both the first method and the third method to reduce the EMI, if designers want to reduce the EMI to the limit. In the range of frequency that is higher than 500 MHz, both the second method and the third method are effective. The second method is innovative to use HIS construction to reduce EMI on PCB and it is easy for designers to use this construction on PCB in the range higher than 500 MHz. The third method used the traditional idea of grounding to reduce the EMI, which is safe to reduce the EMI on PCB. If designers want to pursue safety and stability, the third method is suitable for them. Although the second method is innovative and seldom used, this method has been supported by experimental data, which means the second method is also effective. Apart from that, if the material designers used cannot be wrapped completely, which means the material cannot use shielding to reduce EMI, designers must consider the third method to reduce the EMI.

6. Conclusion

Every method has its irreplaceable characteristic, which means PCB designers have to choose the most suitable method by considering their special situation, such as the range of frequency, price, and material. This paper has compared three different methods to help designers choose. In the range between 0 MHz to 80 MHz, it is effective for designers to choose the first method with great efficiency and low price and time. In the range between 80 MHz to 500 MHz, it is important to both use the first method and the third method to reduce EMI to the limit, which means designers should both shorten the PCB trace but also apply the multilayer board in PCB design. In the range higher than 500 MHz, both the second and third methods are effective. It is safe for designers to choose the third method, but the second method can be one of the features of PCB design. Therefore, this paper suggests PCB designers search for the most suitable method to reduce the EMI for their PCB design. It will help them to increase efficiency in the PCB design. However, because only three methods were compared, this paper cannot guarantee that the methods listed above are the only chosen for PCB designers.

References

[1] Paul C and Nasar S 1987 Introduction to electromagnetic fields, 2nd ed. McGraw-Hill, New York.

[2] Montrose M 1996 Printed circuit board design techniques for EMC compliance

[3] Brist G, Krieger J and Willis D 2012 Pcb trace impedance: impact of localized pcb copper density 1–9, Intel Corp.

[4] Senthinatan R and Price J 1994 Simultaneous switching noise of cmos devices and systems Norwell, MA: Kluwer.

[5] Radu S and Hockanson D 1999 An investigation of pcb radiated emissions from simultaneous switching noise in Proc. IEEE Int. Symp. Electromagnetic Compatibility, vol. 2, Seattle, WA, pp. 893–8.

[6] Shantala and Sudheer 2018 Analysis of electromagnetic interference radiations from the pcb traces International Conference on Recent Trends in Electrical, Control and Communication (RTECC), pp. 74-8.

[7] Ye X, Hockanson D, Li M, Ren Y, Cui W, Drewniak J and DuBroff R 2001 Emi mitigation with multilayer power bus stacks and via stitching of reference planes IEEE Trans. Electromagn. Compat., vol. 43, pp. 538–48.

[8] Shahparnia S and Ramahi O 2004 Electromagnetic interference (emi) reduction from printed circuit boards (pcb) using electromagnetic bandgap structure in ieee transactions on electromagnetic compatibility, vol. 46, no. 4, pp. 580-587, Nov. 2004.

[9] Montrose M 2000 Printed circuit board design techniques for emc compliance, IEEE press series.

[10] Virdi A, Salunkhe P, Choudhary D and Mahadik T 2018 Enhancement of pcb design for radiated immunity compliance in analog signal measurement international conference on electromagnetic interference & compatibility (inceimc) pp.1-4.