EMI-EMC ANALYSIS OF PRINTED CIRCUIT BOARD TRACES

[1]SATISH KUMAR DAS, [2]VISHAL H. SHAH

[1,2]Dept. of E.C.E, B.I.T, Mesra, Ranchi, India
E-mail: satish.das296@gmail.com

Abstract- This paper presents the Electromagnetic interference (EMI) and Electromagnetic compatibility (EMC) analysis of a Printed Circuit Board (PCB). The width of the differential pair trace of the PCB was varied and the emission values were measured and compared, demonstrating an improvement in radiated emissions when the width of the trace was decreased. The trace width of the PCB was increased from 0.1143 mm to 0.8 mm, which resulted in improvement of emissions up to 40 dB.

Keywords- EMI,EMC,PCB,FCC.

I. INTRODUCTION

Electromagnetic interference (EMI) is everywhere and unavoidable. It exists in nature and is also manmade. EMI is the presence of unwanted electromagnetic energy which has the potential to cause disturbances in electronic devices. Sources in nature can come from lightning and electrostatic discharges, while man-made EMI can originate from motors, power lines, fluorescent bulbs, and many other places. Any object which has a time-varying electric or magnetic field can be a potential source of EMI[1][3]. All electronic devices have the potential to be sources of EMI by generating conducted or radiating emissions, along with being victims by accepting EMI from other sources. The amount of EMI a device contributes is referred to as conducted or radiated EMI, and the amount of EMI the device is able to withstand is referred to as EMI susceptibility. In an ideal world the perfect device would not conduct or radiate any EMI and would be perfectly immune to EMI susceptibility[4], however due to the way electronics work this is an impossible task to achieve. All devices will conduct or radiate some EMI; the amount must be regulated to below a threshold set by the FCC inSection15 of its regulations. Any electronic device which is to be bought or sold must be certified that it meets electromagnetic compatibility (EMC) requirements.

II. EMI-EMC ANALYSIS OF PCB

A. Types of EMI

There are two types of EMI emission types. These types are conducted and radiated. Conducted EMI is energy which has coupled to a power or signal bus, and is propagating through the system. Radiated EMI is interference which is no longer confined to a wire, but is an electromagnetic wave propagating away from the source. The source of EMI is wherever the interference is generated. A receptor for EMI is the device being affected by the interference. The amount the device is disturbed by unwanted EMI is referred to as susceptibility. Reducing the device susceptibility to EMI is also referred to as EMI hardening. There are two different domains which can be used to analyze EMI[9]. These domains are commonly referred to as the time and frequency domain. In the time domain, signals are analyzed as they change with time. In the frequency domain, signals are analyzed with respect to a specified band of frequencies.

B. PCB and Its Type

A printed circuit board (PCB) [6] mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate. PCBs can be single sided (one copper layer), double minded (two copper layers) or multi-layer. Conductors on different layers are connected with plated-through holes called vias. Advanced PCBs may contain components [7] - capacitors, resistors or active devices - embedded in the substrate. Printed circuit boards are used in all but the simplest electronic products. Alternatives to PCBs include wire wrap and point-to-point construction. PCBs require the additional design effort to lay out the circuit but manufacturing and assembly can be automated. Manufacturing circuits with PCBs is cheaper and faster than with other wiring methods as components are mounted and wired with one single part. The various types of PCB are

Single-Sided PCB:
Single-sided PCB [8] means that wiring is available only on one side of the insulating substrate. The side which contains the circuit pattern is called the shoulder side whereas the other side is called the opposite side. Single sided printed circuit boards are widely used in various electronics applications such as simple circuitry and where the manufacturing costs to be kept minimum.

Double Sided PCB:
Double-sided printed [8] circuit boards have wiring patterns on both sides of the insulating material; i.e. the circuit pattern is available both on the components

side and the solder side. The component density and the conductor lines than the single-sided boards.

Multi Layered PCB:
Multi-layer boards use plated-through holes, called vias [8], to connect traces on different layers of the PCB. Multi-layer PCB's become attractive for designs that require high functional density in a small space. The additive process is commonly used for multi-layer boards as it facilitates the plating-through of the holes to produce conductive vias in the circuit board.It is used in situations where the density of connections needed is too high to be handled by two layers.

III. EMI SIMULATION AND ANALYSIS

The space surrounding an antenna is usually divided into three regions: (a) reactive near-field; (b) radiating near-field; and (c) far-field regions[11][10]. In the immediate vicinity of antenna, there is the reactive near-field and the fields are predominantly reactive fields. The radiating near-field is the region between near and far field, in which reactive fields are not dominate; the radiating fields begin to emerge. However the shape of radiation may vary appreciably with distance. The far-field region is a region far from the antenna where the angular field distribution is independent of the distance from antenna. This region is dominated by radiated fields. Thus near-field and far-field are defined as the field of electromagnetic radiation that is emitted from the antenna along with transition zone.

PowerSI is a new generation of power and signal integrity tool designed for the electrical analysis of integrated-circuit packages and PCB's. It provides fast and accurate full-wave results and allows designers to overcome the challenge of high speed design issues related to power, ground, and signal integrity. PowerSI simulates the electromagnetic field phenomenon directly in the frequency domain. PowerSI simulation tool provides an actual simulation flow for EMI with near-field and far-field analysis. It also imitates the microwave anechoic chamber environment. As shown in Fig.1, the device under test (DUT) is placed at the height of 1m on the experiment table and the antenna is put at the height of 4m. The horizontal distance between them is 3m [1].

The analysis of a pair of differential trace of a known multi-layer PCB layout is done using PowerSI. The differential pair trace isexcited and then it is simulated at various frequency ranges. The radiation valuesat various frequency ranges are analyzed and compared with the allowable limits set by FCC by varying the trace width.In attempting to break down complicated EMI problems with different electronic devices, multiple circuit designs were analyzed separately. By analyzing each problem independently observations could be made about how each contributes to the overall EMI problem.

All radiated EMI measurements are done using Power SI tool over a range of frequencies from 100 MHz to 500MHz.The EMI measurement of the differential trace pair is carried out keeping the length of the trace fixed and unchanged. The measurement is carried out in accordance with varying width of the traces.Fig.2, and Fig.4 are the result of near field simulation and Fig. 3 and Fig. 5 are fixed point observation resultsat various trace widths where [E]v and [E]h indicate the vertical and horizontal electric field intensity.

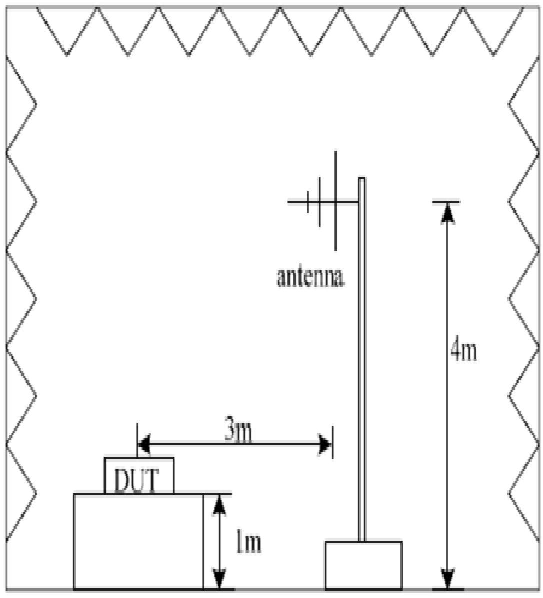

Fig.1 Microwave anechoic chamber environment

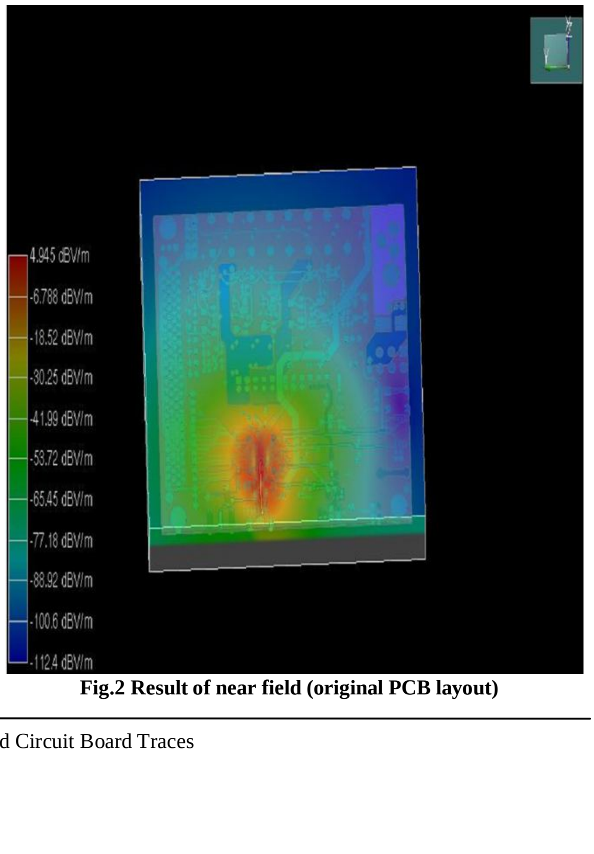

Fig.2 Result of near field (original PCB layout)

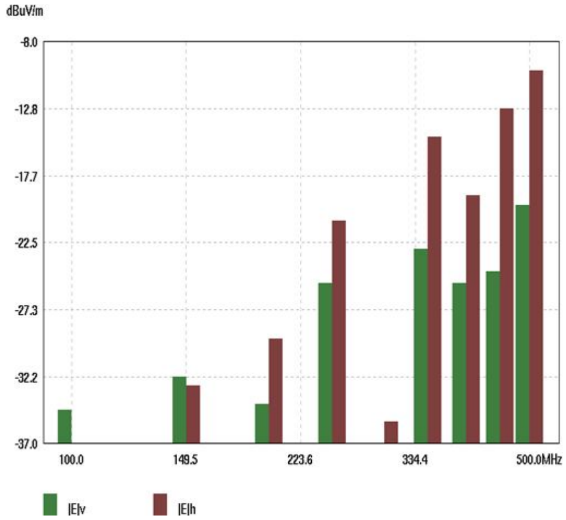

Fig.3 Fixed Point Observation Plot

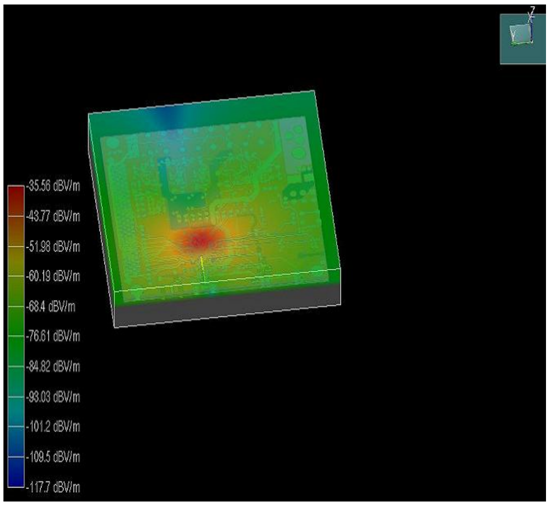

Fig.4 Result of near field (trace width decreased)

CONCLUSION

As per the results obtained the emission form the layout of the board was found very high above the layout of modified board where the trace width was increased .The point of this analysis was not to design the perfect ultra-quiet PCB but rather to show the merits of following proper design strategies to reduce emissions[9][12]. Using this low emission design strategy it is found that the radiated emissions were reduced very effectively when the width of the trace was increased keeping the length of the trace constant. The trace width of the PCB was increased from 0.1143 mm to 0.8 mm, which resulted in improvement of emissions up to 40 dB.

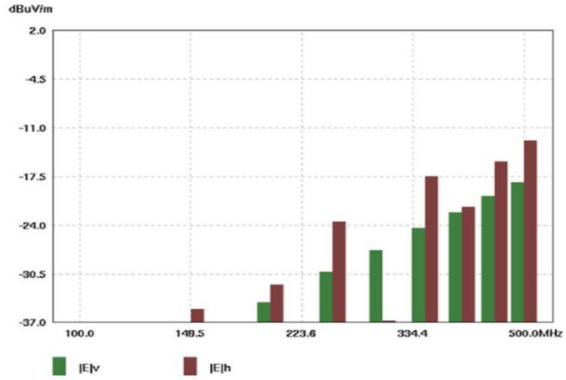

Fig.5 Fixed Point Observation Plot (trace width increased)

REFERENCES

[1] WANG Shenglong, SUN ling,SUN Haiyan and YANG Lingling, "Relationship between SSN and EMC in Sip," International Conference on Electronic Packaging Technology, pp. 6222-625, 2013.

[2] Arthur T. Bradley, Jennine Fowler, Brian Yavoich and Stephen Jennings," Reducing Printed Circuit Board Emissions with Low-Noise Design Practices", IEEE, pp.613-616, 2012.

[3] Bruce R. Archambeault, PCB Design for Real-World EMI Control, Kluwer Academic Publishers, Boston, 2002.

[4] C.R. Paul, Introduction to Electromagnetic Compatibility, 2nd ed., Wiley Interscience, New Jersey, 2006.

[5] S. Radu and D. Hockanson, "An investigation of PCB radiated emissions from simultaneous switching noise", Proc. IEEE Int. Symp. Electromagnetic Compatibility, vol. 2, pp.893 -898 1999

[6] T. H. Hubing, "Printed circuit board EMI source mechanisms", Proc. IEEE Int. Symp. Electromagnetic Compatibility, vol. 1, pp.1 -3 2003 .

[7] N. Kashyap , T. Hubing , J. Drewniak and T. Van Doren "An Expert System for Predicting Radiated EMI from PCBs", Î¡roc. IEEE Int. Symp. On EMC, pp.444 -449 1997

[8] C. T. Robertson, Printed Circuit Board Designer's Reference (electronic reproduction), Prentice-Hall, Upper Saddle River, NJ 2004

[9] W. B. Halaberda and J. H. Rivers "Measurement Comparisons of Radiated Test Facilities", Proc. IEEE Int. Symp. on EMC, pp.401 -406 1992

[10] T. Hubing , J. Drewniak , T. Van Doren and N. Kashyap "An Expert System Approach to EMC Modeling", Proc. IEEE Int. Symp. on EMC, pp.200 -203 1996

[11] M. Coenen "On-Chip Measures to Achieve EMC", Proc. of 12th International Zurich Symposium on Electromagnetic Compatibility, pp.31 -36 1997

[12] W. E. Halaberda and J. H. Rivers "EMC Site Comparisons", Proceedings of the 9th International Zurich EMC Symposium, 1991

★★★