ISSN: 2321-8169
2180 – 2183

Emission Reduction Techniques for Printed Circuit Board

Satish Kumar Das
Dept. of E.C.E
B.I.T , Mesra
Ranchi, India
email: satish.das296@gmail.com

Mr. Vishal H. Shah
Dept. of E.C.E
B.I.T, Mesra
Ranchi, India

Abstract- In this paper two prototypes of PCB have been used to measure the emission pattern from the board. Both boards have the same dimensions, material properties and connectors. In prototype PCB1(Good Board) have been created to improve the emission results and analyze the difference in results by following proper layout rules and incorporating various emission reduction techniques. However PCB2(Bad Board) proper layout techniques are not followed to analyze the radiation pattern and it has been created to violate the layout rules. The emissions from the two boards were then measured and compared demonstra -ting an improvement in radiated emissions and passing FCC Class B limits.
Key words: PCB,FCC,EMI

*****

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 of its regulations. Any electronic device which is to be bought or sold must be certified that it meets electromagnetic compatibility (EMC) requirements.

II. 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][11], 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.

ISSN: 2321-8169
2180 – 2183

III. PRINTED CIRCUIT BOARD DESIGN

Two PCB are designed using identical electronic circuitary.PCB1 is called the "Good Board" is designed to have several low noise layout practices discussed in this paper while "PCB2" is made to neglect this practices.

A brief summary of the specific differences is outlined in Table I. A more detailed discussion of each difference follows the table.

TOPIC	PCB1(Good Board)	PCB2(Bad Board)
Ground Planes	Separate digital and analog ground planes	Single ground plane
Decoupling Capacitors	Complex decoupling network	Single decoupling capacitor value
Cut outs	No cut out or voids present	Presence of voids and cut outs
Via connection	Connection of all ground via to ground plane and power via to power plane	Absence of connection
Power plane	Power plane is shorter than ground plane	Power plane is larger than ground plane
Component placement	Components of one clock trace are placed closely	Placed far away increasing the emission

Table 1 Difference between printed circuit boards

Board Differences

1) Ground Planes

Many designs benefit from multiple ground planes whose interconnections are carefully managed. This type of ground isolation can reduce return current spreading effects-such as a particularly noisy clock return from modulating the reference of an analog input signal. For this reason, the "Good" board was designed with separate digital and analog ground planes. [13][14]The "Bad" board also used two ground planes, but they were stapled together using multiple vias across the surface of the board-essentially acting as a multi-layer ground plane. Conceptually, this should allow more undesired current spreading between the components.

2) Decoupling Capacitors

Decoupling capacitors are used to reduce the noise resulting from dynamic current switching . These capacitors can be viewed as both dynamic current sources and power supply filters. Decoupling capacitors serve two purposes: First, they are sources of charge to devices that are sinking or sourcing high frequency currents. The capacitors act like charge buckets, quickly supplying or accepting current, as required by the devices located in the immediate vicinity. Decoupling capacitors reduce the voltage sags and ground shifts.

Secondly, the capacitors provide a path for the high frequency return currents on the power plane to reach ground. If the capacitors are not available, these currents return to ground through I/O signals or power connectors, creating large loops and increasing radiation. For the PCB2, 0.1 uF decoupling capacitors were on every integrated circuit.. For the PCB1, a network of decoupling capacitors, ranging from 0.01uF to 45uF, was used to help broaden the power-to-ground impedance null, thereby minimizing the power-supply switching noise.

3) Cutouts

Voids or cutouts were made in PCB2 while these were avoided in the good board or PCB1.Voids and cutouts adds to emission from the board.

4) Via Connection

In PCB1 or good board all ground vias are connected to every ground plane, and similarly, every power via to all power planes are connected at equal potential. While this technique is not followed in PCB2 which gives rise to conducted emissions from the board.

5) Power plane

The power plane in PCB1 is kept shorter than the ground plane by at least 5X the spacing between the power and ground planes. This allows any AC difference in potential to be absorbed by the ground plane.

6) Component placement

It is usually good design paractise to reduce emission from the board by placing all components associated with one clock trace closely together. This reduces the trace length and reduces radiation in PCB1.

ISSN: 2321-8169
2180 – 2183

IV. FCC Limits

The Federal Code Of Regulation (CFR) FCC Part 15 is a common testing standard for most electronic equipment. FCC Part 15 covers the regulations under which an intentional, unintentional, or incidental radiator that can be operated without an individual license. FCC Part 15 covers as well the technical specifications, administrative requirements and other conditions relating to the marketing of FCC Part 15 devices.

Frequency in MHz	Class A radiation (dBuV/m)	Class B radiation (dBuv/m)
30-88	49.5	40
88-216	54	43.5
216-960	56.5	46
Above 960	60	54

Table 2 FCC Limits

IV. RESULT AND ANALYSIS

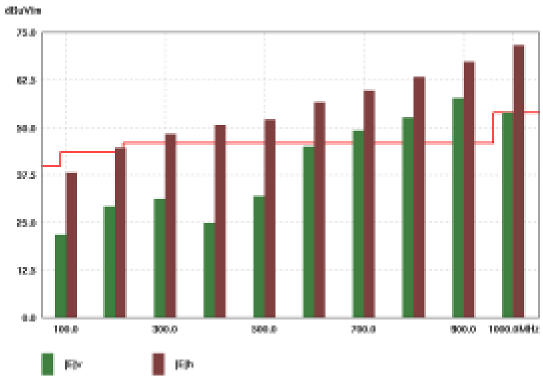

Fig.1 Emissions from bad board (failing FCC class A limits)

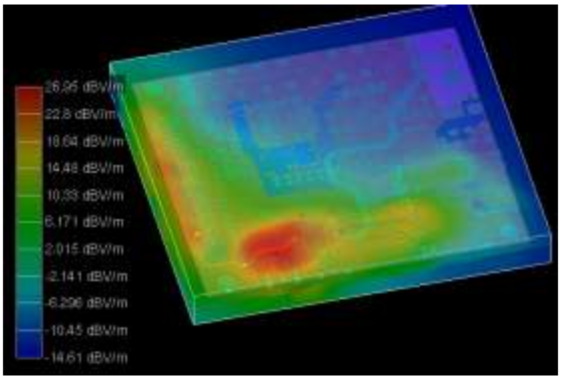

Fig.2 E field radiation from bad board at 1GHz

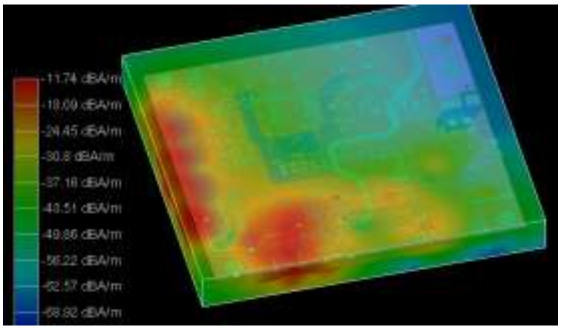

Fig.3 H field radiation from bad board at 1GHz

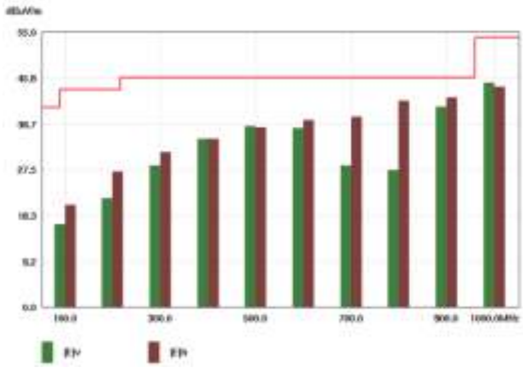

Fig.4 Reduced emission from good board(passing FCC class B limits)

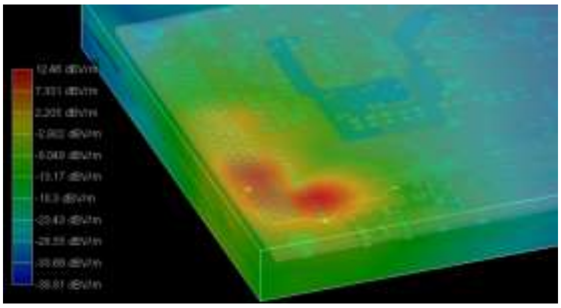

Fig5. Reduced E field radiation from good board

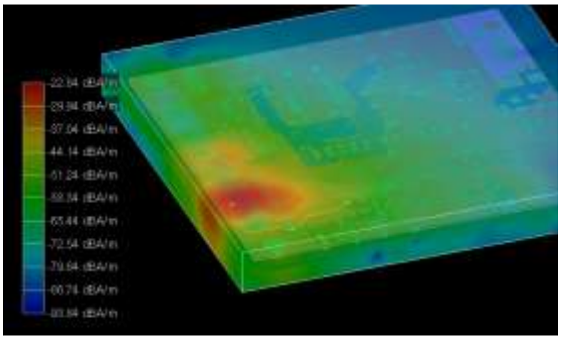

Fig.6Reduced H field radiation from good board

V. CONCLUSIONS

As shown in the data, the emissions from the "Bad" board were as much as 20 dB above those of FCC class B than the "Good" board. Following the proper low noise design practises the (E field and H field) radiations were decreased by 14dB.

ISSN: 2321-8169
2180 – 2183

REFERENCE

[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] Tim Williams, EMC for Product Designers. 41h ed., Elsevier Ltd., Massachusetts, 2007.

[12] Mark I. Montrose, EMC and the Printed Circuit Board, Wiley Interscience, New York, 1999.

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

[14] H.W. Ott, Noise Reduction Techniques in Electronic Systems, 2nd ed.,

[15] Wiley Interscience, New York, 1988.