Evaluation and Minimization of Radiated EMI of High Frequency RF Devices

Sunil R. Gagare[1], Rekha P. Labade[2], Arun E. Kachare[3]

Assistant Professor [1] ,Associate Professor[2], Assistant Professor[3]
Department of Electronics and Telecommunication Engineering, Amrutvahini College of Engineering Sangamner, Affiliated to Savitribai Phule Pune University, Pune, India.

Abstract— *Electromagnetic Interference (EMI) is the presence of unwanted electromagnetic energy which has the potential to cause disturbances in electronic devices. The effects of electromagnetic interference is particularly troublesome for printed circuit board (PCB) designed for high frequency RF circuits. The components on a PCB may be digital or analog. Transmission lines connecting the two different sections are used to transmit signals back and forth. Unfortunately, as frequency increases and signals are enhanced, noise related to those frequencies is also enhanced, thus creating EMI. In terms of propagating path, EMI is classifies into conducted EMI and radiated EMI. The conducted EMI results from that the radiated signal stick on the power line and hard to detect and recognize. Therefore, it is necessary to build the causes and effects methodology by way of the correct measurement to maintain the electromagnetic compatibility (EMC), to target the electromagnetic interference, and to detect radio frequency interference(RFI). Electronic systems are expected to operate normally within a given environment without internally or externally radiating excessive amounts of electromagnetic energy. In this state, they are called electromagnetically compatible (EMC).*

Keywords— *Electromagnetic interference (EMI), radio frequency interference (RFI), Electromagnetic compatibility (EMC).*Introduction.

I. INTRODUCTION

Basically electronic devices/modules typically generate undesirable electromagnetic energy. This electromagnetic energy often generates an unwanted EM field or a transient within the RF band (10 kHz - 10 GHz) of the EM spectrum and are commonly referred to as EMI's (Electromagnetic interference). Such electromagnetic interference (EMI) is being known to interfere with the designated operation of the electronic circuitry of other proximate electronic devices. Radio frequency interference (RFI) is often used interchangeably with EMI, although it is restricted to the radio frequency (RF) portion (10 kHz - 10 GHz) of the EM spectrum. The printed circuit board (PCB) acts as a propagation channel for unwanted noise sources and couples this unwanted noise interference onto other peripheral circuitry, leading to the radiation of generated EMI into free space. The main causes of EMI due to PCB are following [1]:

- Common impedance coupling via power and ground traces.
- Antenna loops formed by ICs and their bypass capacitors, which include power and ground leads.
- Printed circuit board traces carrying signal currents.
- Crosstalk between adjoining signal traces.

A number of international government agencies impose guidelines on the allowable EMI that electronic modules can emit. In the United States of America (USA), the Federal Communications Commission (FCC) has divided EMI for computing electronic devices into two categories: (1) Radiated and (2) Conducted Emission. Table-I lists the maximum permissible Radiated Emission (RE) interference from the electronic devices, measured in terms of Electric Field Strength [2]-[4]. The maximum permissible limit of Conducted Emission (CE) interference is typically 260 μV over the frequency range of 450 kHz to 30 MHz, and related with the noise interference fed back to the power supply lines.

Table-I

Maximum permissible Radiated Emission (RE) FCC-Class B standards.

Freq. (MHz)	Distance (M)	Field Strength (μV/m)
30-88 MHz	3 Meter	100μV/m
88-216 MHz	3 Meter	150μV/m
216-1000 MHz	3 Meter	200μV/m

1.1Methods To Minimize Radiated Emission

A. Circuit Design Aspects

- Use slow and high noise immunity logic components.
- Operating the component in linear mode.
- Use RF suppressors like isolation transformer, optical isolators, and ferrite beads.
- Use RF chokes and inductors to confine RF energy to desired circuit.
- Minimize signal bandwidth and maximize signal levels.

- Use watchdog Timer circuit for microprocessor /microcontrollers.
- Oscillator circuits should be covered by conductive metallic bread on PCB.
- Use short lead lengths for RF components.
- Use feed through capacitors for inter stage coupling and power input connections to RF circuitry.
- Use Electrostatic screening for transformers.

B. Use of Filtering Techniques

Use of Filtering techniques in power supply lines and signal input/output is recommended. Also filtering should be used in the circuits having interference sources. [5] Use of suitable RFI filters with X and Y Capacitors and proper grounding schemes can result into reduction of conducted emission level.

C. Shielding Techniques

EMI shielding can be done effectively by using specially designed metallic Enclosures made up of Copper, Aluminum, Aluminum alloy and Magnetic materials . The Enclosures can be sealed properly by using Copper Foil and suitable Conductive rubbers /gaskets. [5], [6]

D. Selection of Shielded Cables

Twisted / shielded Teflon cable should be used for minimizing electro-magnetic interference. Terminate shield to proper type of connectors. Shield is to be grounded at one end to avoid circulating current loops. Run cables away from apertures of shielding close to conductive grounding structures. Avoid resonant length of cables and consider damping cables with ferrite suppressors. Bunching of cables carrying identical electrical signals should be avoided to minimize cross-talk. Avoid pigtails while grounding the shield. Use Copper breaded conduits for cables to increase the shielding effectiveness especially in stringent conditions of radiation fields. The standard practices for routing of cables should be followed. [7]

E. Multilayer PCB Design

From EMI point of view, multi-layered PCB is recommended. [5] Multilayer boards are formed by etching few double sided boards and gluing them together. PCB design should follow standard practices like 20-H rule for minimizing the EMI problems. [8] The recommended materials for PCB are fiber glass resin (FR-4), woven glass/ ceramic loaded, PTFE/Ceramic (Teflon) and GTEK. [8]

F. Grounding Considerations

The common ground impedance can be reduced by better grounding techniques. Single point grounding arrangement is recommended for low frequencies and multi-point grounding for high frequencies. The ground of Analog, Digital and RF circuits to be terminated to power supply ground separately. Floating ground arrangement is recommended to isolate circuits producing interference. Opto-isolator and isolation transformer can be used for this purpose. The methods discussed above can be useful in reducing Radiated Emission from Electronics equipments/systems. [18]

2 Test Setup for Measurement of Radiated EMI

For testing of radiated EMI from the circuitry, we have to use spectrum analyzer along with EMI test probes. The test set up is shown in fig-1. The radiated EMI from the circuit is measured by spectrum analyzer. According to EMI performance we have to modify the PCB of high frequency device.

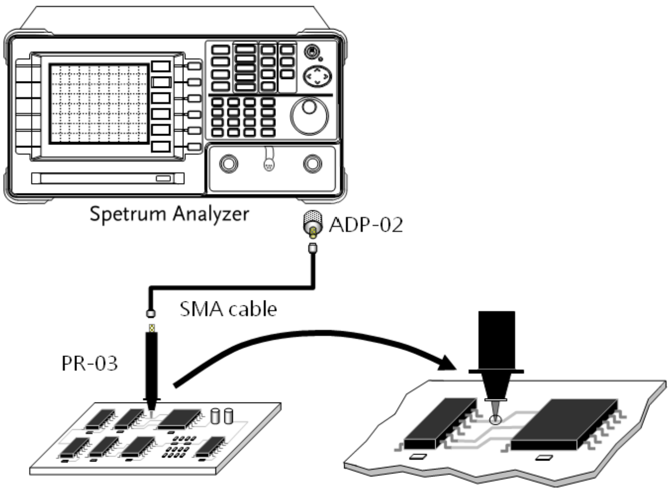

Fig-1: Test set up for radiated emission measurement

3. Radiated Emission Prediction

Noise emissions can be conducted by power lines into systems or systems can conduct noise emissions onto power lines. Noise emissions can also radiate into the internal circuit space and internal circuits can also be susceptible to radiated noise. For example, a PCB ground loop makes an antenna that effects other portions of the circuit. As illustrated in Fig.2, radiated noise can be coupled into a system and appear as conducted noise. This noise causes logic gates to trip at the wrong voltage level or sensitive analog circuits to have incorrect voltage levels. Two main coupling mechanisms from parallel conductors on the PCB board can increase noise. Parallel conductors can couple a signal onto each other if the signal on one conductor is a time-varying signal. (A time-varying signal has electromagnetic fields that can couple on the other conductor as defined in Maxwell's equations). The conductors are not physically connected but are electromagnetically connected, this is known as the transmission line effect. On the Schematic a capacitor or transformer appears to be connected between the two conductors [5].

For intentional transmitters (e.g. broadcast towers), the electromagnetic field next to a transmitting antenna is very complex. This field is called the **near field**. However, the field becomes an uniform plane wave some distance from the antenna. This field is called the **far field**. The near to far field transition (equation below) occurs at about one-sixth of a wavelength from the transmitting antenna.

Near to far field transition $\approx \frac{\lambda}{2\pi}$ [meters]

This next equation, Equation 1, shows how to calculate the far-field radiated emissions from any RF transmitter:

$$E= \frac{\sqrt{30\ X\ Pt}}{r} \text{ [Volts/meter]} \qquad (1)$$

For example, the far field for a FM transmitter at 100 MHz occurs at about one-half meter. The transmitter electric field strength 100 meters (***r***) away from the transmitter (Pt=250 kW) equals 27.4 volts per meter. For unintentional noise sources, E/E designers should consider circuit loops.
For unintentional noise sources, the designers should consider circuit loops.

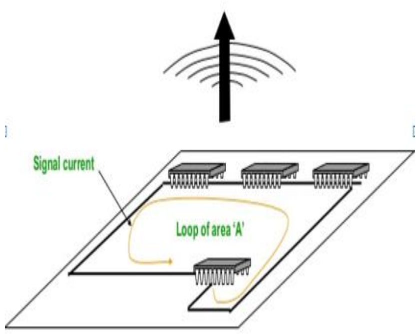

Fig-2: Differential-mode EMI radiation from a PCB

Equation 2, gives the maximum radiated emission in dBµV per meter from a small loop. Differential mode (DM) current, *In*, is the normal signal or power current that flows in a loop.

$$E_{DM} = 1.316 \left(\frac{f^2 . A . I}{r}\right) \mu V/m \qquad (2)$$

Where,
A is the area of a small loop
F is the spectral signal frequency
I is the spectral signal current
r is the distance from the small loop to the measurement antenna or the distance between a radiating generator loop and a receptor circuit.

Using equation 2, the EMI field strength can be calculated. For example,

where: f = 50 MHz
Vp-p = VOH - VOL = 5V - 0.2V = 4.8Vp-p
Zc= 100 Ohm, r = 2 m , L = 5 cm, W = 5 cm
A = L • W = 5 cm • 5 cm = 25 cm^2

$$I= \frac{Vp-p}{Zc} = \frac{4.8}{100} = 0.048A$$

$$E= (1.316 \text{ X } 50^2 \text{ x } 25 \text{ x } 0.048)/ 2$$

E=1974 $\mu V/m$=65.90 dB $\mu V/m$
The FCC Class A limit at 3 m is 300 µV/m while the Class B limit is 100 µV/m.

Reducing the Loop Area with ground plane:
So what is solution? The solution is to reduce the loop area of circuit. The higher the frequency, the better the performance and the higher the emissions potential. Loop current, to a large extent, depends on the device technology, the loads, and the operating frequency. Faster devices with higher peak currents produce higher emissions. PCB layout is the best control over loop area, and smaller loops are better [7]. Attention to detail in the layout will help minimize emissions radiating from the PCB [16].
The PCB layout must be designed to keep high-speed loop areas small [5]-[7].

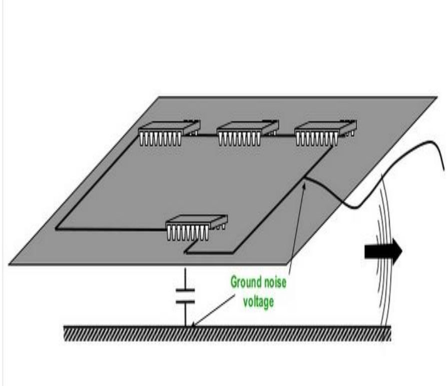

Fig-3: Ground plane added to PCB

For the circuit shown in Fig. 3, the loop width now is 0.15 cm compared to 5.0 cm in Figure 2. Using the same length dimension of 5.0 cm, there is a 32 times improvement in the EMI performance for this PCB.
E=(1.316x 50^2 x 0.75 x 0.048)/ 2 = 59.22 $\mu V/m$
=35.45 dB $\mu V/m$

In Figure 3, the actual path of the current flow is from IC 1 decoupling capacitor through the chip to the IC 1 output pin. The current then flows to the input pin of IC 2, through the chip to the ground pin, and via the ground plane to the input pin. From here the return current flows to the ground terminal of the decoupling capacitor IC1.
So by proper designing of PCB, the radiated EMI becomes reduces according to FCC class-B regulations. The EMI response from spectrum analyzer is shown in fig-4.

Fig-4: EMI minimization by proper design of PCB

4. CONCLUSIONS

The main aim of this paper is to measure and analyze the radiated EMI of high frequency devices. The radiated EMI may affect the performance of other devices. So it is required to minimize the EMI and to meet the electromagnetic compatibility (EMC).There are different methods to control the radiated EMI such as shielding, grounding, proper PCB layout design etc. Here differential mode radiated EMI is controlled by keeping small loop area in PCB layout. The measurement is done by using spectrum analyzer along with EMI testing probes.

REFERENCES

[1] B. Slattery and J. Wynne, "Design and layout of a video graphics system for reduced EMI", Application Note, AN-333, Analog Device.

[2] T. A. Jerse, "The effect of open-loop gain on the radiated emissions from the power supply lead of an oscillator", *1993 FCS*, pp. 62-66.

[3] I-Chang Wu, C. W. Lo, and K. L. Fong," Method and apparatus for a crystal oscillator to achieve fast start-up time, low power and frequency calibration", US Patent No.: 7, 348, 861 B1, arch 25, 2008.

[4] Rafel Fried and Reuven Holzer, " Reduced Power Consumption and EMI", IEEE 1995 Custom Integrated circuits conference, pp. 301-304.

[5] Roger Swan berg, "Estimating Emissions From Your Printed Circuit Board" March 2007.

[6] Swan berg, R., PCB Emissions Prediction Workbook, V3.2, used in D.L.S. Controlling Emissions by Your Design seminars, D.L.S. Electronic Systems, 2005 - 2007.

[7] Mardiguian, M., *Controlling Radiated Emissions by Design*, Kluwer Academic Publishers, 2001.

[8] Michael Tao Zhang , "Electrical, Thermal, And Emi Designs Of High-Density, Low-Profile Power Supplies" PhD thesis Virginia Polytechnic Institute and State University, February 17, 1997

[9] A.Majid, H.B.Kotte, J.Saleem, R.Ambatipudi, S.Haller, K.Bertilsson, "High Frequency Half-Bridge Converter using Multilayered Coreless Printed Circuit Board Step-Down Power Transformer", 8th Internal Conference on Power Electronics- ICPE 2011 at Jeju South Korea (May 28- June 03 2011)

[10] Tim William, "EMC for Product Designers" 4th edition ISBN–13: 978-0-75-068170-4 11. Ales, A.; Belkacem, F.T.; Moussaoui, D.; , "Laboratory Line Impedance Stabilization Network: Experimental studies," *Environment and Electrical Engineering (EEEIC), 2011 10th International Conference on* , vol., no., pp.1-4, 8-11 May 2011

[12] Chris Likely, "Achieving EMC for DC-DC Converters" http://www.complianceclub.com/archive/old_archive/021132.htm (last accessed February28, 2011)

[13] Krishna Mainali, , Ramesh Oruganti, Kanakasabai Viswanathan, , and Swee Peng Ng "A Metric for Evaluating the EMI Spectra of Power Converters", *IEEE Transactions on Power Electronics*, Vol. 23, NO. 4, JULY 2008 2075

[14] Mark J. Nave, Power line filter design for switched-mode power supply, Van Nostrand Reinhold, New York, 1991.

[15] Shuo Wang; Lee, F.C.; Chen, D.Y.; Odendaal, W.G. "Effects of parasitic parameters on EMI filter performance,", *IEEE Transactions on Power Electronics*, vol.19, no.3, pp. 869- 877,May 2004.

[16] P.V.Y. Jayasree, J.C. Priya, G.R.Poojita and G. Kameshwari, EMI filter design for reducing common mode and differential mode noise in conducted interference. EMI Filter Design for Reducing Common-Mode and Differential-Mode Noise in Conducted Interference

[17] A.Majid,J. Saleem , H.B. Kotte, R.Ambatipudi and K. Bertilsson,Design and Implementation of EMI filter for high frequency power filter.

[18] Shreenivas R Jog, M. S. Sutaone, and V. V.Badawe: "Investigation of Radiated Emissions from GPS Based Vehicle Tracking System Board and it's comparison with different EMI /EMC Standards and Remedies to reduce the
Radiations" International Journal of Computer and Electrical Engineering, Vol.4, No.2, April 2012.