DOKUZ EYLÜL UNIVERSITY

GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

PCB DESIGN AND INTERNAL WIRING WITH REDUCED EMI (ELECTROMAGNETIC INTERFERENCE) FOR FLAT TV APPLICATIONS

by

Atınç ÖĞÜT

June, 2009

İZMİR

PCB DESIGN AND INTERNAL WIRING WITH REDUCED EMI (ELECTROMAGNETIC INTERFERENCE) FOR FLAT TV APPLICATIONS

A Thesis Submitted to the

Graduate School of Natural and Applied Sciences of Dokuz Eylül University

In Partial Fulfillment of the Requirements for the Degree of Master of Science

in Electrical&Electronics Engineering Program

by

Atınç ÖĞÜT

June, 2009

İZMİR

M.Sc. THESIS EXAMINATION RESULT FORM

We have read the thesis entitled **"PCB DESIGN AND INTERNAL WIRING WITH REDUCED EMI (ELECTROMAGNETIC INTERFERENCE) FOR FLAT TV APPLICATIONS"** completed by **ATINÇ ÖĞÜT** under supervision of **ASST. PROF. DR. YEŞİM ZORAL** and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Asst. Prof. Dr. Yeşim ZORAL

Supervisor

(Jury Member) (Jury Member)

Prof. Dr. Cahit HELVACI

Director

Graduate School of Natural and Applied Sciences

ACKNOWLEDGEMENTS

First and foremost, I would like to express my sincere gratitude to my advisor Asst. Prof. Dr. Yeşim ZORAL for her constant support in the construction of the original idea of this dissertation and for her efforts to improve my theoretical background.

I would like to thank to my company VESTEL Electronics for opportunity to improve original idea, chance to realize theoratical knowledges and put assertions into real life and my colleagues for their help and fruitful cooperation. I have both privileged and fortunate to have had such colleagues.

I am so obliged to Mr. Omer KOCABAS (Vestel R&D EMC Manager) and Mr. Murat SARPEL (Vestel R&D General Manager) about their useful comments and motivations.

I would like to thank to my family for their great support.

I am thankful to my dear friend Damla ATESALP for her love and great patience. She has always been with me to help and to support.

Atınç ÖĞÜT

PCB DESIGN AND INTERNAL WIRING WITH REDUCED EMI (ELECTROMAGNETIC INTERFERENCE) FOR FLAT TV APPLICATIONS

ABSTRACT

In this thesis, a design guide constructed with theoratical info for tv applications with a low electromagnetic interference which is almost mandatory in recent times for both technical benefit and commercial trading. This report recommendates fruitful ideas for design and troubleshooting.

Electronic products may often unexpectidely produce radio–frequency (RF) energy. Therfore, these kind of devices have the potential of causing unwanted interference with this energy, which is called Electromagnetic Interference (EMI). EMI is actually the process by which disruptive electromagnetic energy is transmitted from one electronic device to another via raidated or conducted paths. TV is one of the most used electronic product in daily life. And it’s used with other electrical and electronic products in the areas of concern when unintentional sources of RF energy may be observed. As digital and high-frequency systems are increasingly developoed in TV applications and other consumer products, EMI becomes a wider concern.

In this thesis, a design approach for TV applications is constructed in order to save time and money. According to regulations in Europe, it is necessary to pass some tests related to EMI to take a place in Eurepean market. This feature of regulations is related electromagnetic compatibility (EMC). For those companies that send TV blindly to test-laboratory hoping the best and get their unit with failing results, it becomes the responsibility of someone back at the factory to tackle this problem. Therefore a guidance is needed at this point .

The focus of this thesis is to assist and advise in the design and testing of high-technology television at minimal cost. Implementing suppression techniques saves money, enhances performance, increases reliability, and achieves first-time compliance with emissions requirements. TV technology is altering rapidly so new projects have to take place in market in a short period of time. The importance of efficient product development will escalate in this competitive marketplace. Regulatory compliance will always be mandatory. On the other hand, those who integrate EMC into their design process will optimize their productivity and their financial return. This thesis is including the design procedure for low EMI for this aim.

Keywords: electromagnetic interference, electromagnetic compatibility, emc design and testing, maxwell equations, suppresion of EMI, TV applications

FLAT TV UYGULAMALARINDA DÜŞÜK SEVİYELİ EMI (ELEKTROMANYETİK KİRLİLİK) YAYILIMINA SAHİP PCB DİZAYNI VE KABLOLAMA YÖNTEMLERİ

ÖZ

Bu tezde, tv uygulamaları için düşük seviyeli elektromanyetik interferansa sahip dizayn kılavuzu oluşturulmuştur. Son zamanlarda, düşük seviyeli elektromanyetik yayılımlı tasarımların önemi gerek teknik gerekse ticari anlamda artmıştır. Bu tez, kurallar ve uygulanabilir çözümleri sunar.

Elektronik ürünler akım taşıdıkları için Radyo Frekans (RF) enerjisi üretirler. Bu yüzden, bu tip ürünler elektromanyetik kirlilik (EMI) adı verilen istenmeyen interferans oluşturabilirler. EMI, elektromanyetik enerjinin bir elektronik devreden diğerine yayılım ya da iletim yoluyla geçmesi işlemidir. EMI ile ilişkili elektronik ürünlerin bir sınıfı da televizyonlardır (TV). TV günlük yaşamda oldukça sık kullanılan bir elektronik üründür. Aynı zamanda çevresinde RF enerji üreten ve istenmeyen RF enerjiden etkilenecek bir çok elektronik ürün ile birlikte kullanılması muhtemeldir. Teknolojinin gelişmesi ile birlikte, tüm elektronik uygulamalarda olduğu gibi TV uygulamalarında da dijital ve yüksek frekanslı sistemler kullanılmaya başlandı. Bu gelişmeler ile birlikte, EMI konusu her geçen gün daha da önem kazanıyor.

Bu tezin amacı, TV uygulamaları için bir dizayn prosedürü oluşturup EMI konusu için dizayn sonrasında harcanacak zaman ve maliyeti minimuma indirecektir. Avrupa Birliği standartlarına göre, bir televizyonun Avrupa pazarında yer alabilmesi için Elektromanyetik Uyumluluk (EMC) testlerinden geçmesi gerekiyor. Testlerde çıkacak problemler maliyet ve projelerin devreye alınmasında ek süreye neden olur. Bu nedenle, süre ve maliyeti azaltmak için bir rehber gereklidir.

Bu tezin öncelikli amacı, EMC ile ilgili problemleri minimum maliyet ile çözebilmek adına TV tasarım sürecinin bir parçasını oluşturmak. Tasarım aşamasında bu rehberi oluşturacak kurallar dizisi göz önüne alındığı takdirde, maliyet minimuma indirilecek, EMC performansı başarılı olacak ve sonuçların kararlılığı artacaktır. TV teknolojisi çok hızlı olarak gelişmektedir. Bu yüzden projeler en kısa zaman sürecinde devreye alınmalıdır.. Standartlara uyum her zaman için kalıcı ve vazgeçilemez bir kuraldır. Eğer ki tasarım aşamasında EMC kriterleri göz önüne alınırsa üretilebilirlik ve maliyet anlamında bir adım öne geçilecektir. Bu tez, bu çalışmayı sağlayacak dizayn prosedürünü içerir.

Anahtar kelimeler: elektromanyetik kirlilik, elektromanyetik uyumluluk, emc tasarımı ve testi, maxwell denklemleri, elektromanyetik kirliliğin bastırılması, TV uygulamaları

CONTENTS

CHAPTER ONE
INTRODUCTION

In recent years, the concern of EMI takes a big role in product development and in electronic environment. It's getting important in product development, because of the signal integrity. Working of a product as it's in expected functions can be realized unless there are crosstalks, interferences, etc.. between signal traces, cabling mechanism and other intrasystem requirements. It's getting important in environmental, because of the disturbance to other electronic devices. An electronic device shouldn't effect the performance of the other electronic devices those are working in cooperation or without cooperation but just placing by the side of it.

The history of the effects of EMI leans to World War II (Montrose & Nakauchi, 2004). It was also problem in the war while at those times, it was called as Radio-Frequency Interference (RFI). Spectrum of communication transmitters and receivers were developed along with radar systems. Because of the size and expense of the equipment that military owned, the high-technology electronic systems started to have unstable operations.

Following the Korean War, most EMI work was classified when it dealt with the specifics of a particular tactical or strategic system such as ballistic misssile, bombers, and similar military equipment. Conferences on EMI began in the mid-1950s where unclassified information was presented. During this time, US Army created strong programs dealing with EMI, RFI and related areas (Montrose & Nakauchi, 2004).

In 1960s , NASA stepped-up EMI control programs for its launch vehicled and space system projects. Governmental agencies and private corporations became involved in EMI emission problems in equipments such as security systems,
hi-fidelity amplifiers and etc.. Although all of the devices were analog based systems. And work around EMI area increases rapidly when US Air Force declares

the cpncerns due to EMI with Distant Early Warning (DEW) line radars (Montrose & Nakauchi, 2004, chap. 1).

As digital logic devices are increasingly developed fro consumer systems, EMI became a wider concern. In the late 1970s, problems associated with EMI became an issue for additional products including home entertainment systems (TVs, VCRs, PCs, etc..). During this period public became aware of EMI and problems associated with it.

After the public became involved with EMI associated with digital equipment used within public areas, the FCC (Federal Communications Commission in US) in the mid-to-late 1970s began to publicate an emission standard for personal computers and similar product families. In Europe, concerns regarding to EMI developed during World War II, especially in Germany in the forum of VDE (Verband Deutscher Electrotekniker) That's why in Europe, government regulations are more rigorous (Montrose & Nakauchi, 2004, chap. 1).

The issue of compliance became a concern when the European Union (EU) through EMC Directive 89/336/EEC imposed emission requirements. With these regulations, electromagnetic compatibility (EMC) compliance has an increasing role played by electronics in all electronic system. One of these systems is televisions.

Televisions are increasingly developed in recent years under the concern of EMI and EMC. Therefore EU has stated the regulations and strict market audits for one of the most used consumer electronic product; televisions. For broadcast associated audio video equipments family, standard, EN 55013, and for ITE equipments family, EN 55022, have been announced by EU. Televisions have to be sure of compliance with emission requirements according to these standards. Association which is authorized by government has the right to take products from market and test emission levels according to standards stated above. If compliance is not obtained as result of tests, the association complains about the products to the government. And televisions that have the same model with uncompliant, are collected from the market and not allowed to be sold in the market anymore. Government should also give penalty to the manufacturer if there is no corrective action for those models. Performing a corrective actions means “rework”. Amount of those uncompliant televisions have to be collected again from market and modify all sets to ensure compliance. This could cause a big trouble and money loss for the manufacturer.

According to facts given above, the design of televsions with reduced EMI is getting more important day by day with the increase of market inspections. If EMI or EMC is considered at the design stage of projects, there is less chance to fail market inspections, to have penalty from government. Moreover the most important thing is related to quality of products and prestige of manufacturer along the other competitors.

It is the intent of this thesis to present applied engineering concepts and principles along hardware techniques to get a design of a television with reduced EMI. The information is presented in practical measurements and experiments under the theory of electromagnetics. Therefore Maxwell equations, highly technical aspects of circuit theory, field propagation and numerical modelling are mostly guided to this thesis. The aim of this thesis is not to provide discussions on management or legal issues, such as how much testing, how much cost for EMI shielding components is necessary to budget.

General definitions about magnetic field theory and other therotical information are presented in this chapter. EMC regulations, EMC standards and testing procedures follows this chapter to present the idea of how the compliance can be realized and to give information about the regulations. General TV design concepts and EMC related parts of televisions which have high frequency signals or wide range of spectrum are presented next. This should be mentioned because there are other restirictions for the design such as the performance, the artwork. High frequency circuits should be investigated in details while the most problems occur in these parts. As the main subject general EMC design procedures are given in the subsequent chapter. Combining the testing procedure and tv design concepts, design rules for reduced EMI are constructed and tested for the verification of theoratical knowledge in that chapter. Finally, conclusion about the thesis is going to be given. Theoratical knowledge about field propagation given in this chapter should assist to understand and observe the source of emission with probes or other EMI equipments stated in troubleshooting chapter.

1.1 Nature of Product Testing

The clearance of the concept of product testing makes feeling comfortable during design. Understanding how testing system works and which measured datas are valid and accurate will guide the design for defencing TV from compliance tests.

Electromagnetic compatibility test are generally performed in nonideal conditions. Therefore preliminary radiated emission testing should be performed in anechoic rooms that simulate real world exception of high ambient noise. This may assist for predicting whether a product will pass the test or not at open-area test site. But in open-area test sites, before relying totally on data taken, the results should be investigated detailly whether it comes from test setup or from the design. Detailed information of EMC testing environment will be given in the following chapters.

Compatibility refers to radiated electric or magnetic fields that propogate between functional section in a television or digital components or cable harness. Determination must be made according to test results in advance of the candidates of emissions. Depending on the placement of components, susceptible circuits or I/O connectors, potential coupling of internal radiated RF energy must be expected efore finalizing the design of a PCB or routing cables in the manner of high bandwidth switching components. But this condition should be taken into consideration after the validation of measured data has been definite and accurate. This makes the television testing critical.

There are some automatically testing methods of radiated emission. It generally prevents the concern above hand made faults or humanbeing effect. But even computers that perform automatically testing have been developed too much, it is the best way to ascertain validity of data is having a test engineer or technician questioning the results from automated testing. For instance, a problem that may occur during testing can result a spike in the measured data. It should be in a certain time and not continous but it will effect the result of TV testing. At that time, it is more certain to manually observe a large spectral bandwidth of the frequency of emission over an extenden period to determine whether it is a random spike or not. Or in the reverse side, an emission that is not taken into account at final measurement means skipping a real emission comes from television.

Product testing and testing environment is very important. Televisions should be tested under the supervision of related standards. Wrong testing results should provide gross mistakes for manufacturers. Especially a customer or Original Equipment Manufacturer (OEM), who has to verify the others' test results should have conformity concern. Therefore the worst thing of a company selling large quantity of televisions is to be informed by its customer that the design is defective or failing. A greater concern exists for the manufacturers located in Europe that buy their competitor's products and test them for conformity. Regardless of how extensive the original testing may have been, notification of failing data will not be given to the manufacturer but to government authorities. The purpose of reporting nonconforming products to authorities forces the authority to investigate if illegal designs are being placed into markets. This is done to make competitor company in order to result a punisment, penalty, fine or bruise brand name and quality. Therefore primary item to consider when testing a television for certifying for Europe is to perform valid and accurate testing. There shouldn't be any leak of harmful emission during testing.

1.1.1 Compliance and Precompliance Testing

Many country and customers demand compliance according to European standards before goods are imported. EMC Directive in Europe requires manufacturers to issue a Declaration of Conformity (DOC) listing the test standards they have provided when using the standards route to conformity. Customs officers should require this document for importing televisions in Europe. But EMC Directive forces customs officers to request EMC full compliance test reports form the manufacturer.

There are lots of benefits of performing precompliance tests to discover whether there are EMC concerns before mass production of that design starts. Precompliance testing has the advantage of interrupting tests any time and perform a modification or cure for the design for reduced EMI. Full compliance testing is getting more expensive per day and there is no ability for disruption in the test sequence. But although precompliance testing is more sufficient in time and money, compliance form thirdy part accredited laboratories should be taken in order to not face with a penalty. It is always best to never certify a television based on a just precompliance test.

In Europe, European Parliament has enacted legislation that electrical equipments such as televisions to comply with emission levels of protection. When full compliance according to test standards are realized, television is marked via CE logo (Conformity of European).

1.2 Basics of Interference

Electromagnetic interference has to be prevented on a PCB for two basic reasons. One of the reason is to provide signal integrity. Low interference is the result of signal degradation along a transmission path. Parasitic coupling between circuits including crosstalks and field coupling between other internal assemblies in the product have to be reduced for signal integrity. Other reason is the external interactions which are related to emission testing of a product. Emissions of electromagnetic interference derive from harmonics of clocks or other periodic signals. Studies should concentrate on containing the periodic and high frequency signals to as small area as possible, and blocking parasitic sources and coupling paths from the outside world.

RF currents produce risky EM fields when they left enclosures,which are called closed loop circuit. This leaks are critical when equals to significant fractions or times of a wavelength of a rise time distance. Whenever an EMI problem occurs, it is helpful to review and determine the list above according to design specifications. Understanding these five items will clearly define much mystery of how EMI occurs. Applying these five considerations is the process of basics of interference.

But there are specific solutions for each design. For instance, single-point grounding is excellent when it is applied to low-frequency applications such as audio circuits. But is it completely inappropriate for RF signals. And then EMI problem exists. Every general design rule shouldn't be applied blindly for all product designs without realizing more detailed in specific circuits and design.

The aim of this thesis is including television based EMC design according to circuits or cables used in a TV. The solutions and design procedures is specificly intended to television design as it comes from the title of thesis. This necessity comes from the basic realization of EMI described above.

When designing a PCB of television, RF current flow should be concerned instead of voltage. It has a simple reason. Current always travels around a closed-loop circuit following one or more paths. It is the advantage to manage or direct this current within its paths for proper system operation. To control path in which RF current flows requires low impedance. It is also necessary for also return path of current to the source of energy. And interference current should be diverted away from victim with a high impedance. The easiness of controlling RF current comes from this solution. Defining RF current paths is critical to eliminate crosstalks and emission levels. But there are some applications that require a high impedance path from the source to the load. At this point, all possible paths through which the return current may travel should be considered.

There are five major considerations during an EMC analysis on a product or design. The following questions related to definitions below have to be clearified during the development in design process.

- Frequency: Where is the emission in the frequency spectrum ?
- Amplitude: How strong is the energy level of emission source and how great is its potential to cause harmful interference?
- Time: Is the problem continuos in time domain or is it only existing in certain cycles of operation.
- Impedance: What is the impedance of both source and victim unit and the transfer impedance of transfer path between these. Are there any impedance mismatchs?
- Dimensions: What are the physical dimesions of the emitting parts or circuits on the design that cause emissions?

1.2.1 Time Domain and Frequency Domain Analysis

It is more common in digital design to think in terms of time domain but electromagnetic interference is generally viewed in frequency domain. It is difficult to understand EMI problem in time domain alone. All digital transitions in time domain produce a spectral distribution in frequency domain.

Baron Jean Baptiste Joseph Fourier (1768–1830), a French mathematician and physicist, formulated a method for analyzing periodic functions. He proved that any periodic waveform could be defined as a infinite serious of sine waves at harmonics of fundamental frequency. The composition of harmonics is determined via mathematical application called Fourier Transform. Fourier series can easily be calculated for simple waveforms and displayed with modern instrumentation (Montrose & Nakauchi, 2004, chap. 1).

As it is seen, there are two ways of thinking during design stage. One is time domain thinking which mostly cares about signal integrity and functionally importance. And the other one is frequency domain thinking which mostly cares about passing legally defined limits to get compliance. In reality, two domains are closely related to eachother, while a transmitted EM field can be viewed by oscilloscope in time and spectrum analyzer in frequency.

In physical universe, the only natural waveform is an analog sine wave. Digital components are truly analog devices using an infinitely fast AC slew rate signal. For instance an op-amp is high gain amplifier in which fast feedback is added to control its overall response characteristics. These kind of digital components are used to perform a wide variety of linear functions. There fore the meaning of word “digital” can be technically replaced with infinitely fast slew rate signal. This is illustrated in Figure 1.1.

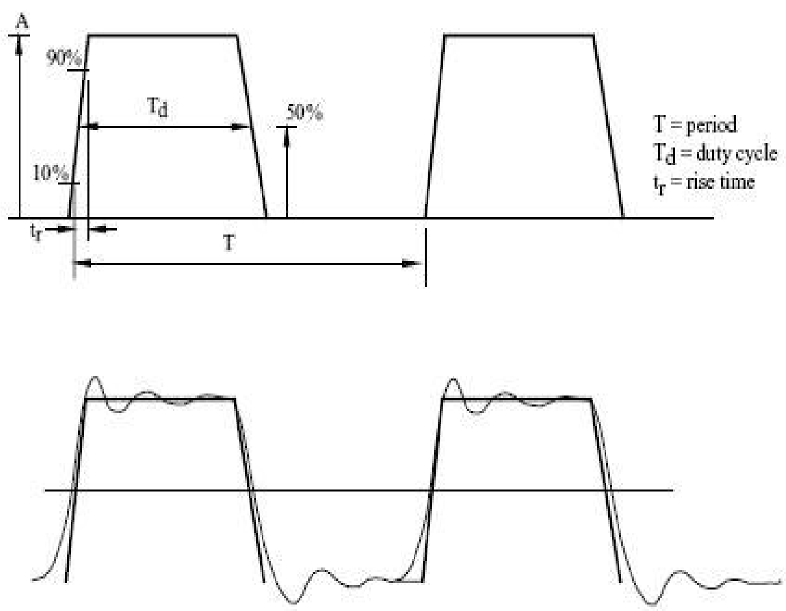


Figure 1.1 Analog sine wave appears as a digital transition

CHAPTER TWO
ELECTRIC AND MAGNETIC FIELD THEORY

In this section, instead of touching on detailed electromagnetic concepts, basic types of fields that exist are mentioned, the manner how they propagate and the types of coupling that affect overall system operation. Electromagnetic interference through free space radiation or by propagating EM fields. And coupling is a significant aspect of EMC design.

A primary concern of EMC is depending on the two modes of current flow which are differential and common mode. Definition of these two modes and how they relate eachother is mentioned in this section. Because electromagnetic problems are mostly occured by one of these current flow mode. For almost every type of designs desires differential mode current flow but common mode has to be distinguished from the overall system in order to have reduced EMI in the design. Therefore these two modes have to be seperated. In this section the theory of these two modes and relationship between them will be introduced for these reasons.

2.1 Relationship Between Electric and Magnetic Fields

Understanding field theory is a must before testing for EMC. In this part, basics of electromagnetics are introduced but in terms of EMC design. There are two basic types of field, electric and magnetic. The word electromagnetic comes from two word; electric and magnetic. Therefore, these two types of fields have to be investigated simultaneously. Both fields have their own characteristics and diffenrences from the other one that can be easily understood. According to Maxwell's equations, a time variant current produces a time variant magnetic field, which also rises up the electric field whereas these two fields are related to eachother mathematically. The structure of these two fields differs them also. A current in a transmission line creats magnetic field and a static charge distribution in the same function of capacitor produces alectric field. To understand these fields, source of field and how it effects a radiated signal have to be examined. Additionally the strength of emission or field decreases while the distance between source and observation point increases. The close part is called near field and in general EMC literature, it is determined as λ/6. Any distance greater than this value is called far field.

Current flow exists via two kind of sources:

- Magnetic sources which assumes current flow in closed loop configurations (Simulated by a loop antenna)
- Electric sources (Simulated by a dipole antenna)

The effects of near field and far field is illustrated in Figure 2.1. In the figure, the change in RF energy can be observed while it comes from near field to far field.

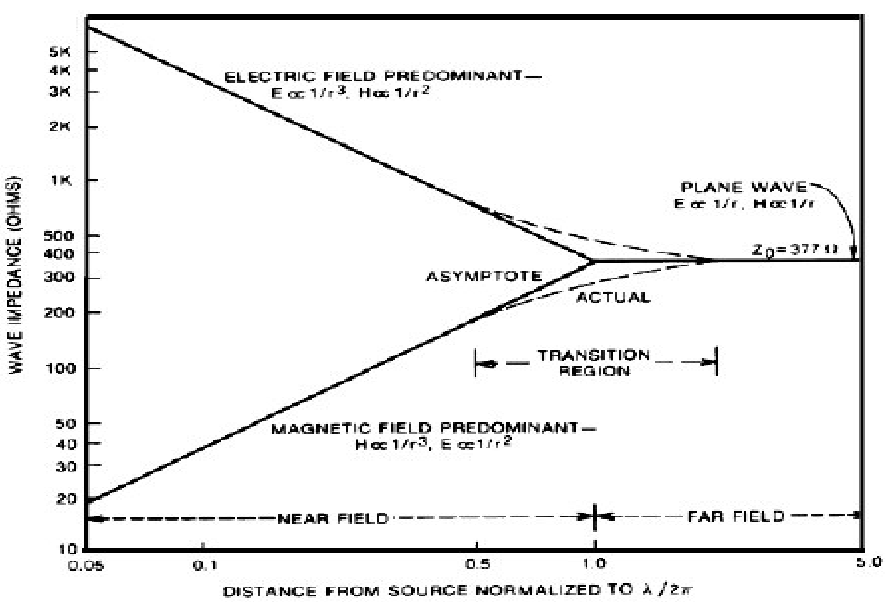


Figure 2.1 Wave impedance vs distance from electric or magnetic sources

All electromagnetic waves are combination of electric and magnetic components This is represented by plane wave or Poynting vector. Actually Poynting vector is a method of expressing the direction and power of EM wave in units of W/m^2 . In far field, electric field and magnetic field are right angles to eachother and perpendicular to direction of propagation. Electric and magnetic fields cannot be experssed by just only themselves at any field. The reason that it is called as planar wave comes from the appearenc of physical profile from wavefront. Both fields propagate radially from the source with the velocity of light (c ~ 3x 10^8 m/s where μ_o=4π x 10^{-7} H/m and ε_o= 8.85 x 10^{-12} F/m). Electric field component is in terms of volts/meter and magnetic field component is in terms of ampers/meter. The ration of these two components is identified as characteristic impedance of EM wave and in terms of ohms. Wave impedance is constant and does not depend on the source type. As it is known, for a plane wave in free space;

$$Z_o = \frac{E}{H} = \sqrt{\frac{\mu_o}{\varepsilon_o}} = \sqrt{\frac{4\pi x 10^{-7} H/m}{(1/36\pi)(10^{-9})F/m}} = 120\pi$$

or approximately 377 Ω.

2.1.1 Magnetic Sources

The circuit given in Figure 2.2 is a simple magnetic source type.

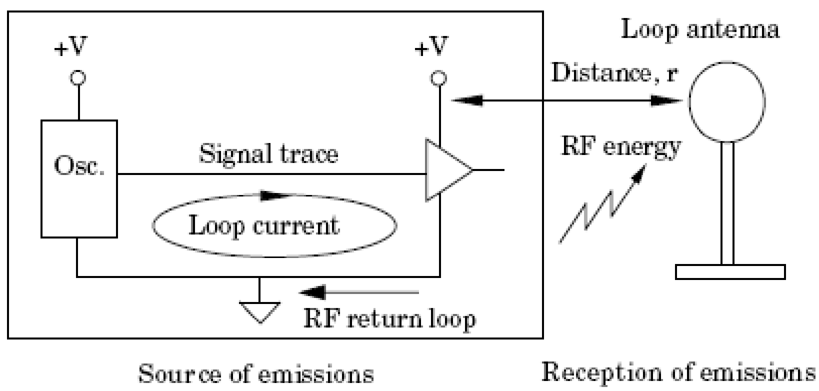


Figure 2.2 Magnetic source circuit

Current in this circuit is flowing in a closed loop (signal transmission and return current). Emission or generated field of the magnetic source can be easily examined. The following four parameters gives opinion on what to do if reduced EMI is desired.

- Current Amplitude: Radiated field is directly proportional to energy of the current in the signal line.
- Orientation of Source Relative to Measuring Device: For a radiated field to

be measured, orientation of the source loop should match with the measuring antenna orientation. In EMC testing procedure, measuring antenna can be oriented in horizontal and vertical polarization.

- Size of Loop: If loop size is much less than the wavelength at the generated signal frequency, field strength is propotional to the area of the loop. Larger loops generate more EMI.
- Distance: The rate of EMI is related to the distance between the source and measuring antenna. The distance also determines whether magnetic or electric component is dominant. When measuring antenna is too close to the source, magnetic field falls with the cube of distance ($1/R^3$). When the distance is in the far region, EM plane wave is observed. In this region, it field strength falls inversely with increasing distance (1/R). Electric and magnetic fields intersect at one sixth of the wavelength. (or $\lambda/2\pi$). As it is known wavelength is the ratio of speed of light to frequency. (Montrose & Nakauchi, 2004, chap. 2)

2.1.2 Electric Sources

Electric sources can be represented as a dipole antenna carrying a time varying change in electric charge. The change in electric charge provides a current flow through dipole length. EMI created by electric sources is also a function of four variables.

- Current Amplitude: Generated field is directly propotional to the amount of current flowing.
- Orientation of Source Relative to Measuring Device: This idea is also same as defined for magnetic source.
- Size of Dipole: Field created is directly propotional to the length of current element. For a specific physical dimension, it should be in resonant frequency.
- Distance: Electric and magnetic fields behave in the same form according to distance. Both field strength falls of with increasing distance.

Propagation of RF energy can be represented in the form given in Figure 2.3 to simulate how electric or magnetic field influence measuring in theory. A time varying electric field between two conductors can be represented as a capacitor configuration. A time varying magnetic field between these conductors can be represented as mutual inductance. Figure 2.3 illustrates these coupling configurations.

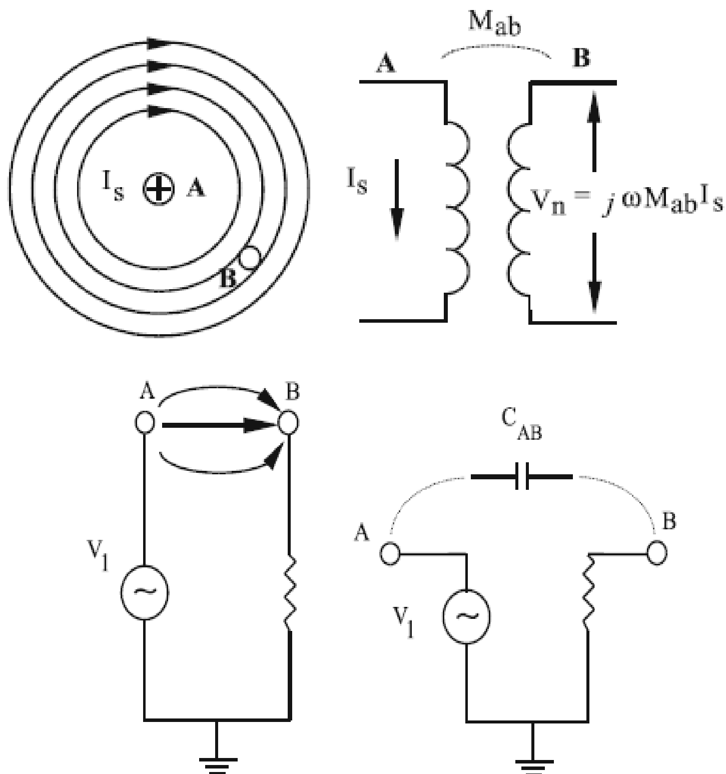


Figure 2.3 Electric and magnetic noise coupling analysis

Generally in EMC study, these noise coupling models are not valid. There are two basic reasons in order to say invalidity. First one is the hardness of solving Maxwell equations for most real world situatioins beacuse of the complicated boundary conditions. Second one is that numerical modelling usually doesn't show all RF energy paths. EMI is generally generated by common mode current flow. Common mode crrent exist if there is an imbalance in differemtial mode transmission which is a primary aspect of EMI. Other parts that take role in EMI is transmission line, reflections and etc.. If they can be modelled in the circuit implementation, it is hard to get accuracy due to the first reason.

For this reasons, it is mostly hard to simulate EMI in this numerical modelling. This kind of simulation would only provide a sight about how RF energy may exist. But it is necessary to have knowledge about how fields are created and propagated in order to decrease the level of EMI.

2.2 Noise Coupling Methods

There are two basic ways to reduce EMI at a design. One is to decrease RF energy emmited from the source. And the other one is to prevent emission by removing coupling paths.

In an EMC situation, there are always a source and a victim. The connection between them is a coupling path. If both source and victim are within the same electrical part, the system is named as 'intrasystem' and if they are in seperate function groups, it is called as 'intersystem'. Whether the coupling path is in intrasystem or intersystem, product design should be analyzed regarding to EMC requirements or reduced EMI.

Almost every electronic design contains elements or components intend to behave like an antenna. These elements include internal cables, PCB traces or connectors, mechanical stuructures and etc.. If there is a source of emission, these components unpurposely transfer RF energy through electric or magnetic field. The supression of this EMI can be provided by components or metal partions that minimize radiated emission by absorbing energy and directing it to back to source or ground. Coupling may occur in either capacitive or inductive method. In this section it is intended to understand the manner of coupling methods to minimize transferring RF energy.

Propagation of RF energy should occur not only in one direct path from source to victim but in different path configurations such as given in Figure 2.4

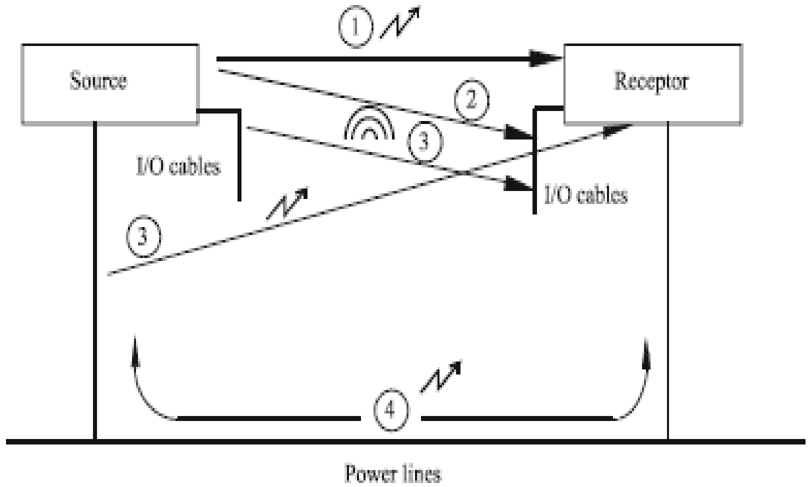


Figure 2.4 Different coupling path configurations

- Direct radiation from source to victim or receptor
- Coupling from source to receptor's cable mechanism
- Coupling from source's cables to receptor's cable or itself.
- Coupling from the same power line used.

The process of coupling can be prevented if the knowledge of field propagation and the manner of coupling types is used logically. For this purpose, the following modules are introduced to reduce EMI by coupling.

2.2.1 Magnetic Field Coupling

Magnetic coupling occurs if a magnetic flux created by a current loop transfers through magnetic flux pattern of another current loop. It is related with the mutual inductance between two loops. Induced RF noise voltage should be defined as;

$$V_2 = \frac{M_{12} dI_1}{dt_2}$$

where M_{12} is mutual inductance. Induced voltage doesnot depend on victim's circuit impedance. It depends on the seperatin of conductors and the length of it which forms also mutual inductance. These information is necessary in a TV design when magnetic coupling occurs on a PCB to distinct it.

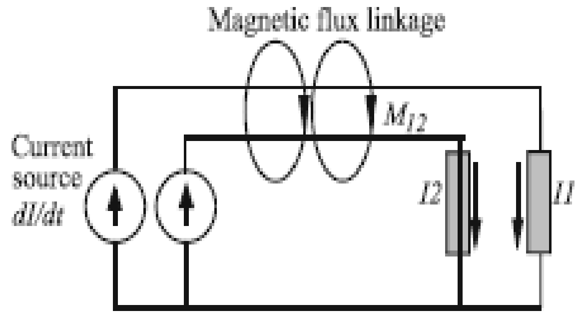


Figure 2.5 Magnetic field coupling

A special condition occurs in magnetic coupling when multiple bonds of cable such as a cable harness is made in TV design. It is useful to partion cable harness according to function areas such as DC voltage, signal, control to reduce magnetic coupling between them.

For magnetic flux coupling, if a RF return current path is defined closely to source tranmission line, fluz cancellation may occur and EMI should reduce in that situation. When forward and return current paths are close enough, The magnetic flux created by the source wire is in the opposite direction of return RF energy. So they should cancel eachother. The process of flux cancelling is a widely used in TV designto achieve EMC and reduced EMI.

Mutual inductance between cable pairs depends on the distance between them. It is inverselt propotional to the log of the distance which is shown in Figure 2.6. Between two transmission line, it is possible to reduce EMI if the distance between them is increased.

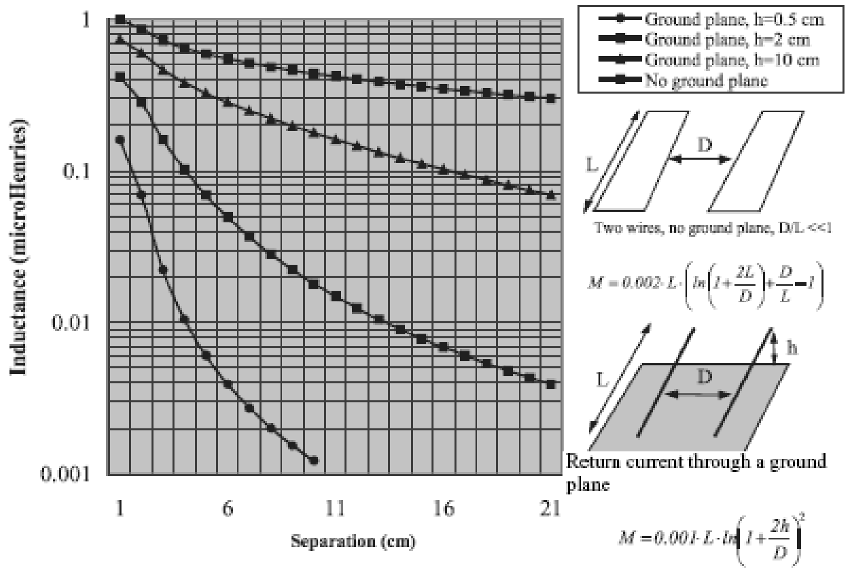


Figure 2.6 Mutual inductance between two transmission lines

2.2.2 Electric Field Coupling

Electric field coupling occurs when a potential difference between two transmission lines or wires is provided. Because if there is a potential difference between two conductors, electric field is developed. Unlike magnetic field coupling which forms a potential in victim circuit, electric field coupling significantly affect the current in victim circuit which may produce EMI.

Induced voltage in victim's circuit is given as;

$$V_{in} = C_{in} Z_{in} \frac{dV_s}{dt}$$

where C is mutual capacitance between conductors. As it is also seen from the equation, the impedance Z of the receptor's circuit is also a variable in electric field coupling. High impedance circuits is more susceptible to capacitive coupling. Electric flux coupling is represented by mutual capacitance. And the noise current injected to victims circuit is given as I= C (dV/dT). This noise current should produce extra EMI in the circuit.

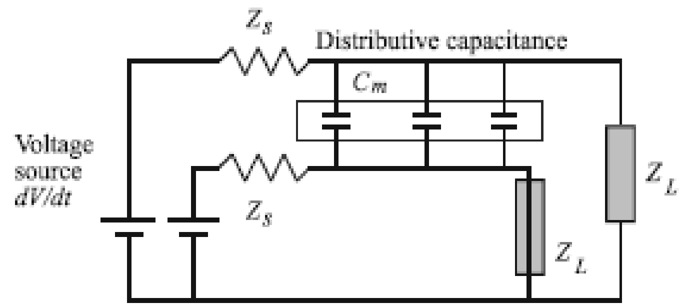


Figure 2.7 Electric field coupling

Mutual capacitance is affected by seperation of distance, especially with the aea of overlap on two wires. In addition, the dielectric material between two lines should also affect the magnitude of capacitive coupling. For situations of high dV/dT should take attention such as switch mode power supply converters. In Figure 2.8, the concern of electric field is presented.

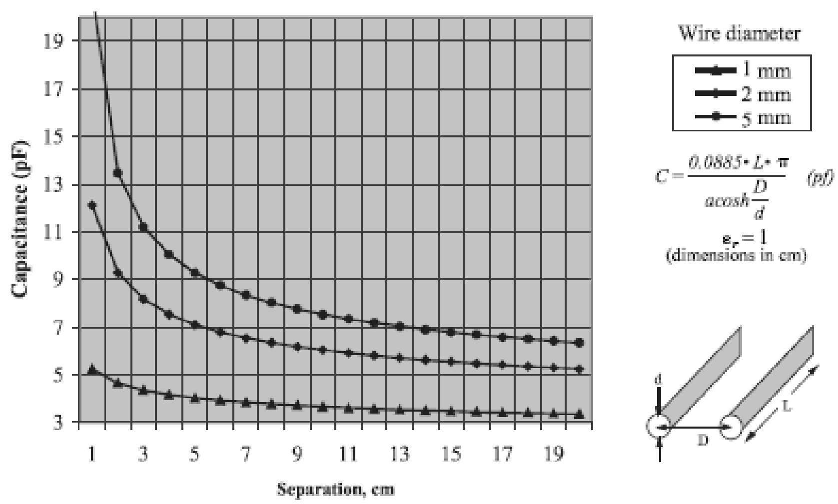


Figure 2.8 Mutual capacitance between two transmission lines

2.3 Common mode and Differential Current EMI Sources

In all circuits, both differential and common mode currents exist. Both types of currents provide RF energy propagated between circuits or transmission lines. But there is a significant difference between two current types. Differential mode DM carries signal data or information on the transmission line which is a desired situation. But common mode (CM) is an undesired side effect of DM transmission and a trouble for EMC.

Common mode is a primary concern of EMI. Simulation software analyses only DM transmission. Therefore with only calculating DM transmission, it shoud predict about only EMI without common modes. When in real life, parasitics, noise coupling or any other reason of common mode show up and unexpected EMI graph is obtained which differs from software simulation.

2.3.1 Differential Mode Current

Differential mode cuurent is the component of RF energy when both signal and return path have the opposite direction of RF energy transmission. If absolute 180o angle between signal and return path, differential mode effect on EMI should be cancelled. But even again, common mode effects may occur because of ground impedance or power plane fluctation etc..

DM signals carries desired information and also have the opportunity of low or none EMI with its return current path. In DM signals generated current is received by a load and same amount of return current should be transferred back to source. The difference between forward and return current due to crosstalks, noise coupling etc. should provide a common mode EMI. In this manner, EMI of DM signalling can be reduced if ground impedance of return path is adjusted as low as possible, short return current pathis provided or etc.. All these methods have the same manner of controlling the excess energy fields through source and back to source.

2.3.2 Common Mode Current

Common mode (CM) current is the component of RF energy when both signal and return path have the same direction of RF energy transmission. The measured EMI is the sum of generated rF fields by both forward and return path current. CM current is generated if there is an imbalance in the circuit. Radiated emission is the result of these imbalances. Flux cancellation is poor when DM signal is not exactly opposite and in phase with its return path. The portion of RF current that is not cancelled exists as common mode current. Common mode (CM) signals are the major concern of EMI and does not carry useful information. The most important thing to prevent CM energy and EMI is to understand and manage RF return current paths.

Return RF current path would like to attempt RF current in the source path and couple with source path to cancel flux. But when return current path is not symetrical to source path with a least impedance, common mode signal occurs. However there is always common mode signal due to residual CM current, not perfect flux cancellation and etc..

Common mode signal can be simulated as a pair of parallel wires carrying DM signal. Along these wires, DM signals flow in the opposite direction. These parallel wires act as a balanced transmission line whcih delivers a differential signal to load. But when CM voltage is placed on this wire no useful information is carried to the load. This wire pair behaves as a dipole antenna with respect to the ground. This antenna radiates unwanted CM energy which is also same as EMI. Common mode currents are generally observed in I/O cables in a television. This is why I/O cables radiate RF energy.

To illustrate this phenomena, consider the figure given below ;

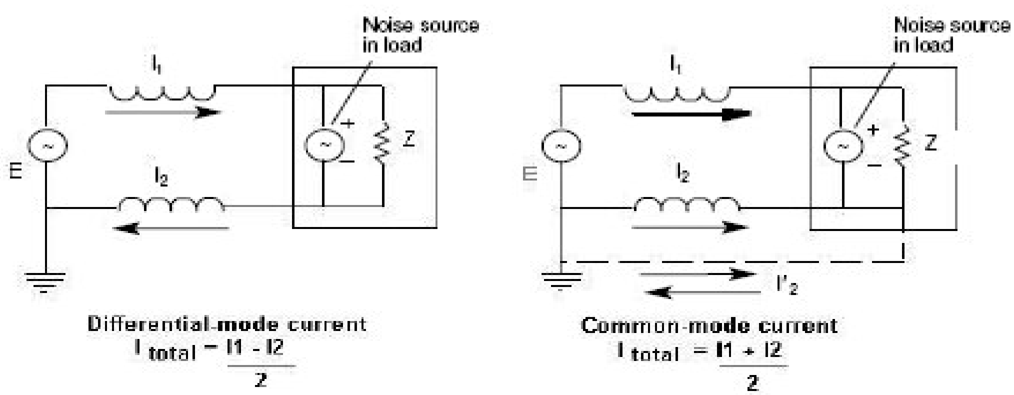


Figure 2.9 Equivalent circuit of DM and CM currents

This example is not mathematically or precisely correct but it gives a simple and good idea to understand a complex topic. The flow current from source E to load Z is represented as I_1. Return current flows back to source is I_2 and this current provides I^1_2 returns in a different path defined as dotted line. This secondary path is called a reference or ground path referred to PCB. And I^1_2 produces CM energy.

For a simple analogy, assume that 1A current is transmitted from source to load. For a correct DM transmission, 1A of source transmitted current must return to the source represented by I_2. If there is no loss on the transmission system and amount of transmitted current is as same as the return current, there is a perfect balanced system. Magnetic flux between these transmission lineswould cancel eachother and there should be no radiated EMI that cause harmful interference to other circuits, cable mechanisms or to air that will cause imcompliance on EMC testing.

Regarding to CM configuration above, assume half of transmitted current is consumed within the load. That means there is a loss of 50%. Under this situation, when Kirchoff’s law is applied to the circuit, there is a missing 50%. Half of the current is transmitted to the load while half of it propagates through an alternate reutrn path represented by I^1_2. For this example, the dotted line above configuration more than one return path exists to satisfy Krichoff’s law. The summation of source and return current equals 100% at a fixed point in time.

Applying Kirchoff’s law to the circuit CM configuration above, it gives beloq explanantions;

$$I_{total(dm)} = \frac{I_1 - I_2}{2} = \frac{1A - 1A}{2} = 0A$$

$$I_{total(cm)} = \frac{I_1 + I_2}{2} = \frac{1A + 0.5A}{2} = 0.75A$$

For DM transmission, electric field component is created by the difference of I_1 and I_2. If they are equal to eachother that means perfect balanced system, there will be no RF field radiation. If there is an imbalanced system that is caused by RF loss on a system, CM energy is produced that causes EMI. This CM current is the majority of EMI problems. (Montrose & Nakauchi, 2004, chap. 2)

2.3.3 Radiation Due to Differential and Common Mode Current

Differential mode radiation is generally caused by the flow of RF current loops in a system. Radiated RF energy due to DM current is approximately given as below;

$$E = 263x10^{-16}(f^2 A I_s)(\frac{1}{r}) V/m$$

RF energy is created from the curretn flowing between assemblies and 0 V reference plane. Radiated emission can be modeled as a small lopp antenna carrying RF currents. The area of the loop is critical for RF radiation. The maximum loop area that will not exceed a specific area given as below;

$$A = \frac{380rE}{f^2 I_s}$$

Conversely, radiated electric field can be calculated from the expression above. As it's seen above, radiated energy falls off inversely proportional distance between the source and receiving antenna. Therefore, special attention should be taken while routing traces, locating source and load components close to eachother, or providing additional shielding.

Common mode radiation is generally caused by unintentional voltage drops at 0 V reference plane. Cable mechanisims that is connected to dirty reference plane will act as a dipole antenna. The far field electric field component is approximately described as;

$$E \approx \frac{1.26(f I_{CM} L)}{r} V/m$$

At a specified antenna length and constant current source, EMI is related direct proportionally with frequency. It is more hard to solve CM radiation. The best and almost only way to reduce EMI from CM current is to decrease the impedance of return current path which is a result of good grounding of PCB. It means in real world a clean ground that gives the stability of 0 V reference. This can be achieved with a proper design of PCB. Otherwise it should be necessary to use CM chokes or filters to reduce EMI produced by CM current which will result additaional cost for the design. (Montrose & Nakauchi, 2004, chap. 2)

This phenomena has to be clearly understood to design a PCB with reduced EMI. However sometimes, there are other restrictions in a design of television such as customer demands, other design parameters, mechanical structure and etc.. Sometimes design rules cannot be clearly applied due to these reasons. At these points, electric field should be calculated with the given expressions in this chapter to control whether emission level is above the limits specified by the offical EU standarts or not. Otherwise it will be an EMC design that gives no opinion of compliance at the design stage and makes the designer send design blindly to EMC testing.

CHAPTER THREE
EMC TESTING

In this section, the questions about the idea of EMC testing, how EMC testing is performed, what the limits are, where the tests are realized have to be answered in order to get precautions for compliance. EMC standart related to TV applications is EN 55013. The scope of standarts that are published in European Union is to give test procedure, specifications and limits. Its aim is to provide the same conditions and testing results in all test laboratories.

According to testing methods, the aim of this thesis is to understand the manner for low EMI in order to get compliance in EMC. The manner in propagation of radiated field occurs is through a dielectric medium which is free space in EMC testing. Free space is capable of supporting field propagation. Transmission lines provide a path for current to flow directing from source to load. It is a think just like a road or high-way. Road is the transmission line and automobile is the current that carries electrical information from one place to another. Transmission lines also provide mechanism same as antenna either in dipole or loop configuration. Electromagnetic radiation is a category of field propagation.

For each field , there are two primary modes of signal transfer; common mode or differential mode. It is impossible to distinguish the mode when the field is measured by antenna but common mode is the more dominant when an EMI is observed. Dependless of two modes of signal transfer, RF radiated energy consists of both electric and magnetic fields simulteanously. When testing a product for complinace purpose or troubleshooting, it is helpful to know which field is dominant. The purpose of this chapter is to provide information about radiated tests and fields. Regulatory standarts contain specification limits to ensure that the magnitude of undesired RF radiated energy is low enough not to cause harmful interference to other devices.

3.1 Performing Radiated Emission Test

Non compliance from this test can occur at any time. The stages of testing and analysis during development are as follows.

3.1.1 Radiation Investigation in Office

This part of analysis includes measurements with near field probes,simulation programs and applying design rules.Various subassemblies are tested for self investigation before final compliance test. Simulated analysis and near field investigation gives an idea about field propagation but may not reflect actual field and far field measurements with all interconnection cables connected, especially if a metal construction is provided in device. This is due to parasitcis, field reflections and capacitive effects between PCB and metal chasis. Time domain analysis defined in previous chapters have to be performed for signal integrity and frequency domain analysis for radiation fields.

This is the most important part of EMC design to perform careful level of analysis during design stage. Analysis should imclude performance, manufacturabilityand compliance to regulatory standarts. During this period, when hardware does not exist, considering and applying design rules, use of simulation tools are the only possible ways of analysis. The element of concern is whether RF energy can be detected at which parts of design. Theoratical and previous experiences take place in this part. These will effect the emission level of the product and cost of EMC troubleshooting techniques. Because if the design rules are not considered at this stage, additional EMI shielding components suh as ferrites, gasker and etc.. should be used in order to get compliance. It will both effect cost, manufacturability and quality of product.

3.1.2 Pre-compliance Testing

This refers to in house testing using a full equipment system and same test setup in fully compliance testing. This work is mostly generated in preproduction samples. The aim of pre-compliance testing is to see real world results for radiated fields and take precautions on PCB if necessary.

Precompliance analysis involves testing relevant to EMC standarts and test procedures. Most EMC standarts describe testing methods themselves or refer toother documents. Radiated Emission test for compliance according to Europe's EMC Directive involves testing methods is EN 55022 and EN 55013 for televisions. These standarts requires an Open Area Test Sites (OATs). But it is difficults to find a location without high ambient noise level because of broadcasting, therefore the use of shielded test facilities, chambers are being more popular. But shielded test facilities should satisfy some properties such as shielding effectiveness, Normalised Site Attenuation (NSA) measurements which is alos defined in EMC standarts. The important item about precompliance testing ,s yo get an idea of all test procedures and what errors may produce while formal testing. Whe performing precompliance testing, one may think reasonable atennas, high cost test sites shoud require but the view point of precompliance testing is to observe sources of radiation fields with considerable cost. There are several ways to perform pre-compliance testing. An example set-up of using current probes or clamps given in Figure 3.1. (Montrose & Nakauchi, 2004).

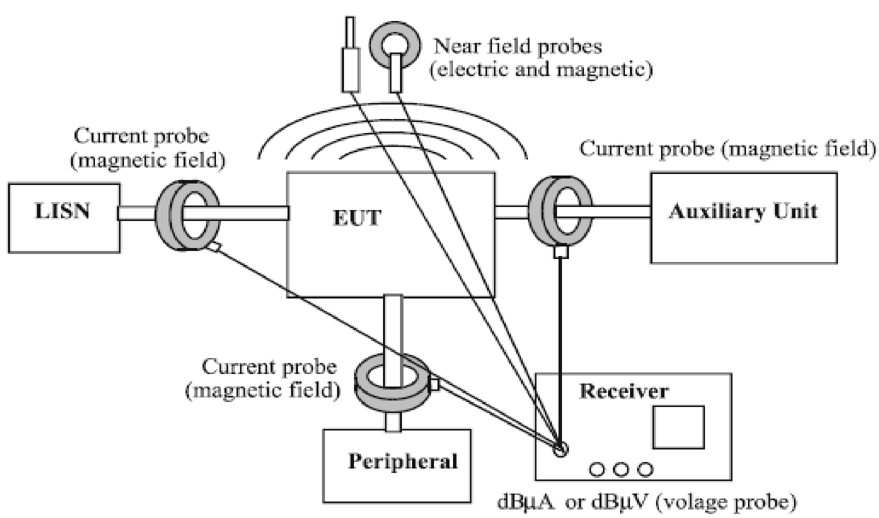


Figure 3.1 An alternate radiated emission precompliance testing using current and near field probes

As it is stated above, there should be instrumentation errors or field reflections when precompliance testing is compared with formal EMC testing. And receivers that is used should not use quasi peak detectors which is used in formal EMC testing. But the results of precompliance testing should clearly identify sources of emissions even they are not going to be uncompliance in EMC. But it saves time and money to decrease the level of radiated field strength on a PCB with the results of preompliance testing.

3.1.3 Formal EMC Qualification Testing

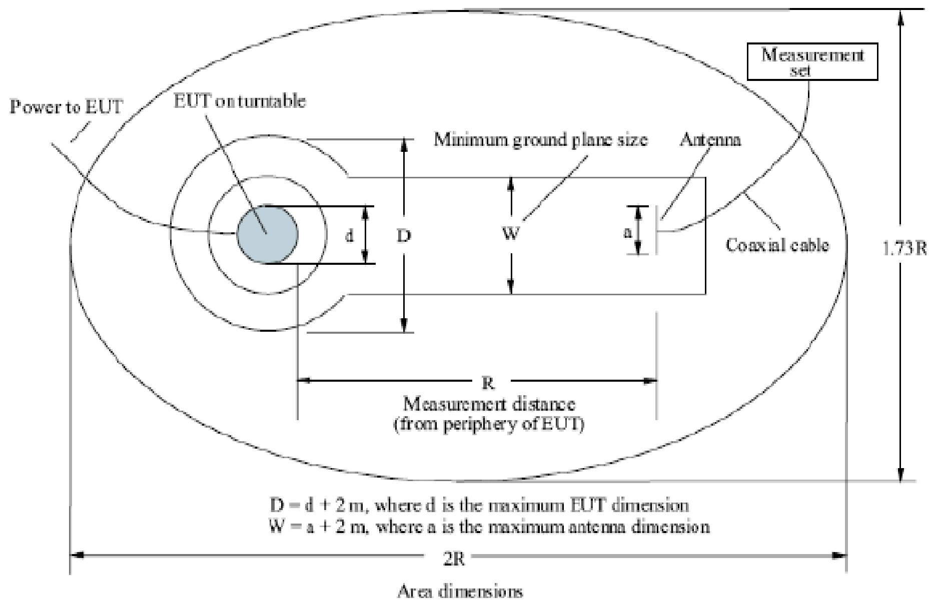
This level of investigation is for official certification and testing of televisions according to regulatory requirements. Tests are performed in accordance with published standarts for televisions which are EN55013 (Sound and Television broadcast receivers and associated equipments- Radio disturbance characteristics- Limits and methods of measurement) and EN55022 (Information technology equipment- Radio disturbance characteristics- Limits and methods of measurement). As it is stated above, test specifications are developed to get system EMC in almost every anticipated location. Formal radiated emission test can be performed in any test site which satisfy requirements stated in the standards, for instance Normalised Site Attenuation (NSA) measurement. If it gives compliance to requirements even in a livingroom, tests can be performed. But due to EM wave reflections and EM sources in daily life, some special rooms are needed to satisfy NSA measurements.

Open-Area Test Sites (OATs), chambers and cells are generally used for formal EMC testing. OATs is the most common one where it is the easiest way to supply.

The requirements of an OATs are;

- System power and cable interconnects for test configuration
- Measurements precautions, flat and free of overhead wires on the ground, away from reflecting structures for NSA measurement
- EUT turntable and antenna positioner for test procedure.

Dimension requirements of an OATs is given below.



R = 3 m	R = 10 m	R = 30 m
6 m x 5.2 m	20 m x 17.3 m	60 m x 52 m

Figure 3.2 Site configuration of OATs

Measuring distance is typically 3,10 or 30 meters. But for a reduction from 10 to 3m, an increase of 10.5 dB is added to specified limit. For instance for televisions at 30 MHz, the limit for 10 meters is 30dBμV/m while 40dBμV/m in 3 meters.

Normalised site attenuation is defined in the standards CISPR 16, CISPR 22 and CISPR 13. According to standards site attenuation is performed just like in the figure below.

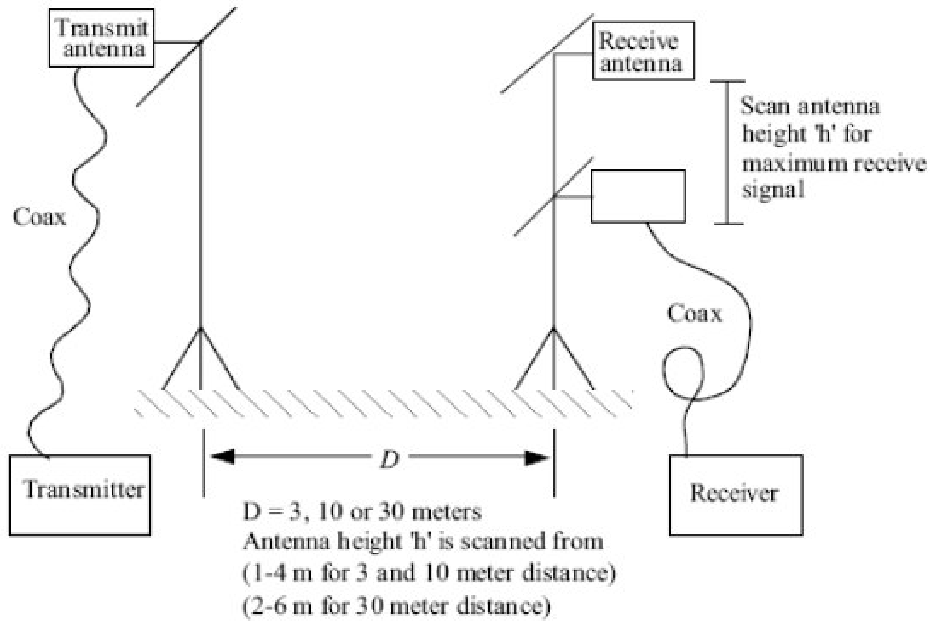


Figure 3.3 Schematic of site attenuation measurement

It is the process of calibrating test site. The measurements should be +/- 4 dB of therotaical NSA curve which is given below (Armstrong & Williams, 2007, part 1).

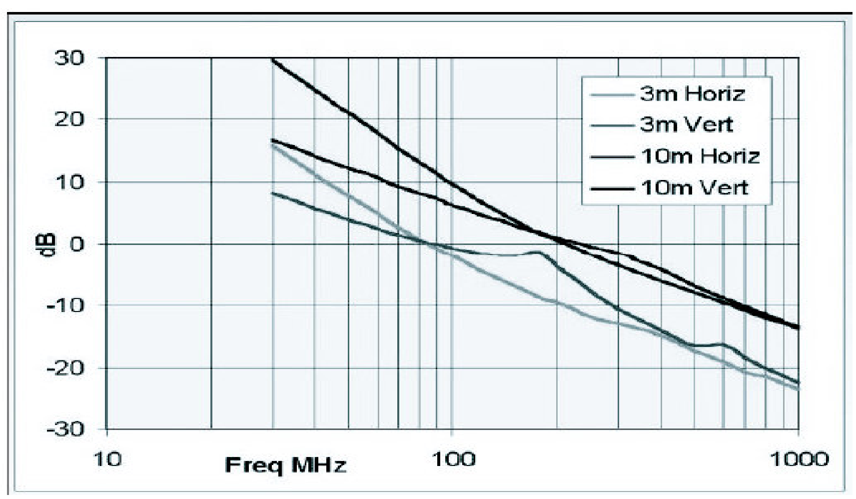


Figure 3.4 Theoratical NSA curves

Electromagnetic scattering should be considered during an OATs construction. Scattering includes RF reflections from biildings, or other metal structures, power lines, trees and bushes, underground water abd cables buried close to the surface. To avoid underground scattering, metallic ground planes are used which is bonding to earth surface. And test place should be far away from large and metallic objects. One example of an OATs is given in the below figure.

Figure 3.5 Example of an OATs

Special rooms where OATs is not sufficient can be used ofr formal EMC testing. They are generally called anechoic chamber in EMC language. There are two types of anechoic chambers which are fully and semi anechoic. Ferrite tiles, absorber cones are used to prevent reflections of EM waves. The difference between fully ans semi anechoic chambers is the ground plane. Fully anechoic chamber includes ferrite tile at all sides where semi anechoic has a metal ground plane and no ferrite at the ground.

Anechoic rooms have more to satisfy the requirements which are NSA as at OATs, field uniformity and shielding effectiveness. General view of a chamber is given in below configuration. (Schaffner, 2000)

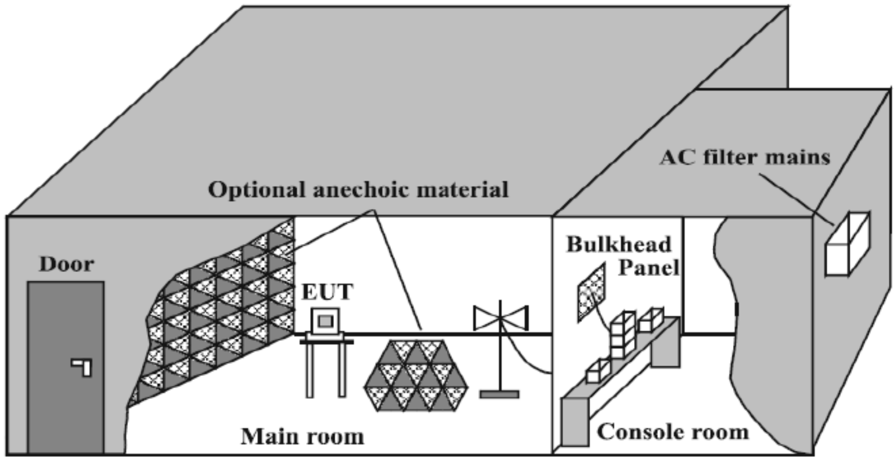


Figure 3.6 Shielded room with anechoic material

Formal EMC testing is generally perfromed in such test sites. And here is how the test is perfromed. Practical test configurations are given in Figure 3.7

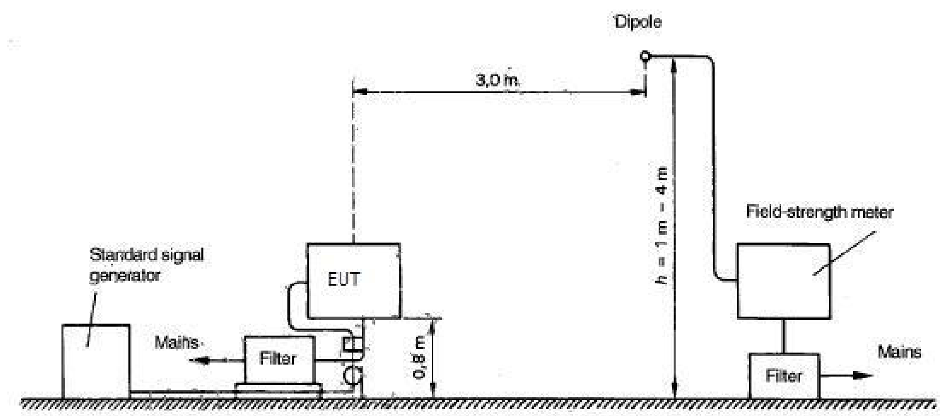


Figure 3.7 Formal radiated emission test measurement setup

Test is performed according to below instructions to find maximum radiation from television according to EN55013;

- Television is installed in the middle of turntable
- Colour bar signal is supplied to TV from pattern generator from control room (all the required channels and TV standards, increasing level until getting a noise-free picture)

- TV is placed on a wooden table on the non metallic turntable of 0.8 m height at a distance of 3 m from the receive antenna.
- Prescan measurements of TV are taken at 0, 90, 180 and 270 degrees of turntable at 1.00 m and 1.55 m horizontal, 2.00 m and 2.50 m vertical polarization.
- At the end of the prescan, final measurement for suspicious frequencies are examined.
- At each suspicious frequency, table is turned from 0 to 360 degree and antenna height is moved from 1 to 4m for horizontal and 2 to 4 m for vertical polarization.
- Highest Quasi-Peak value for each frequency is obtained and noted.

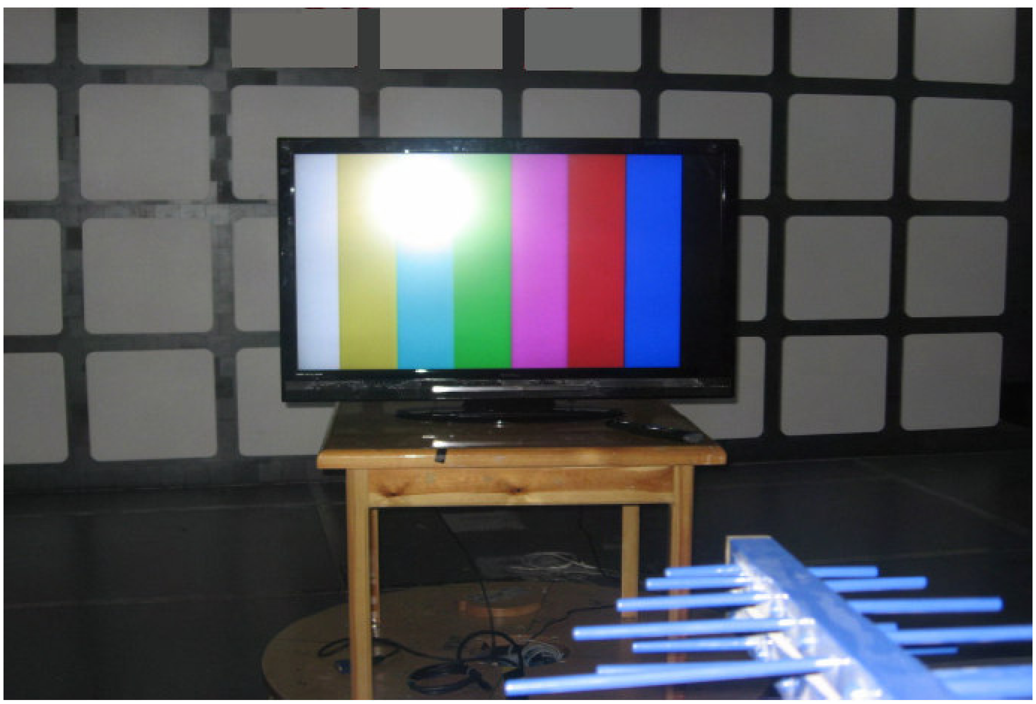


Figure 3.8 Radiated emission test measurement setup according to EN 55013

Procedure is a little bit different according to EN 55022. The procedure is given below for EN55022 ;

- TV is installed with its associated equipment with at least 10 cm distance each.

- TV is turned on and Burn-In-Test software is started to to exercise the EUT with scrolling Hs
- All ports are terminated with suitable cables. Excess lengths of cables shall be bundled at the approximate centre of the cable with the bundles 30 cm to 40 cm in length. Where there are multiple interface ports all of the same type, connecting a cable to just one of that type of port is sufficient, provided it can be shown that the additional cables would not significantly affect the results.
- TV is placed on a 10 cm high wooden pallet or on a wooden table on the non metallic turntable of 0.8 m height at a distance of 3 m from the receive antenna.
- Software is started to take prescan measurements of EUT at 0, 90, 180 and 270 degrees of turntable at 1.00 m and 1.55 m horizontal , 1.55 m and 2.00 m vertical polarization.
- At the end of the prescan, final measurement is performed for suspicious frequencies.
- At each suspicious frequency, the tableis turned, antenna height from 1 to 4m is moved for both horizontal and vertical polarization.
- Highest Quasi-Peak value is obtained for each frequency and it is noted.

The limits for 3m distance for both EN 55013 and EN 55022 are given below:

Table 3.1 Radiated emission limits

Equipment Type	**Source**	**Frequency (MHz)**	**Limit values (dBµV/m) Quasi Peak**
Television receivers	Local oscillator	< 1000	Fundamental 57
		30 to 300	Harmonics 52
		300 to 1000	Harmonics 56
	Other	30 to 230	40
		230 to 1000	47

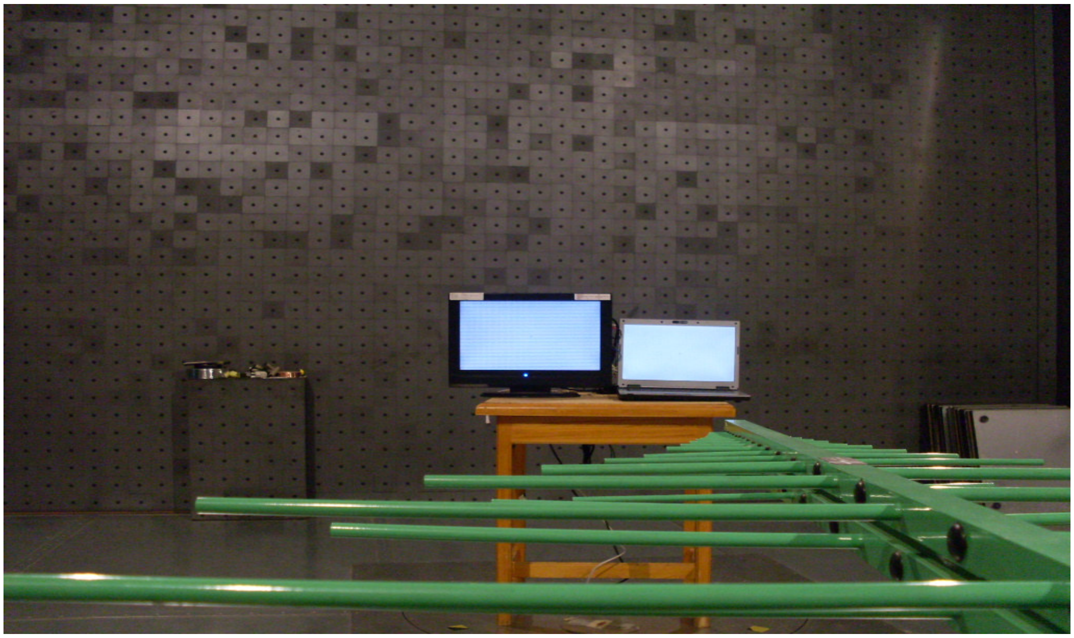


Figure 3.9 Radiated emission test measurement setup according to EN 55022

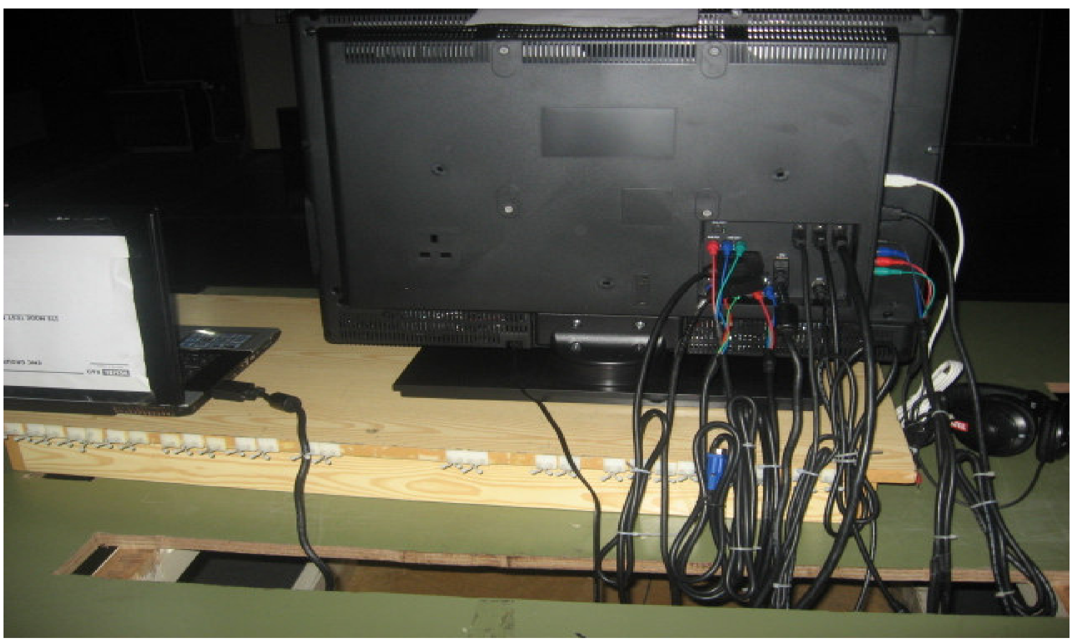


Figure 3.10 All ports are terminated with suitable cables in EN 55022

Compliance is decided according to flow chart given below (Armstrong & Williams, 2007, part 1) ;

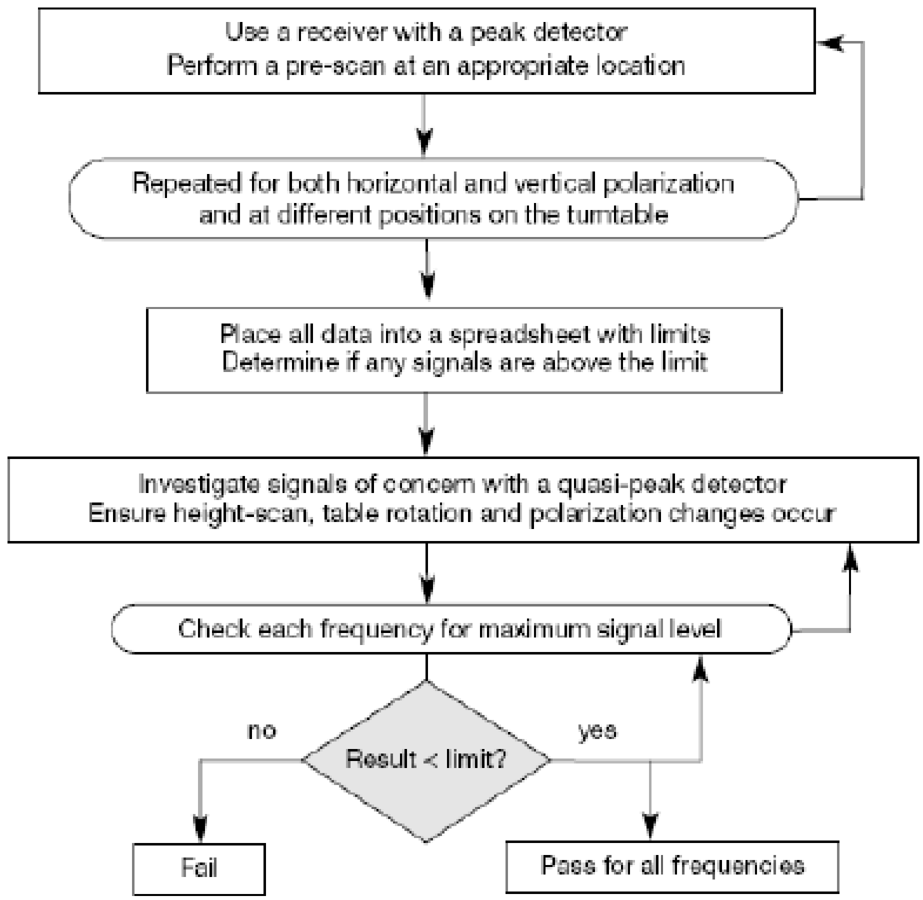


Figure 3.11 Compliance measurement procedure

The main idea of this chapter is to understand the phenomena of EMC testing for the design procedure. It is better to know how the test is performed, which precautions should be considered during EMC testing. More than this, the way to save money with precompliance testing is also mentioned in this chapter. In thge next chapter, design considerations of a television will be considered. And as a result of mixture of these two topics, design rules for low EMI will be constructed.

CHAPTER FOUR

GENERAL TV DESIGN CONCEPTS

This chapter is related to just general television design concepts which are critical for electromagnetic interference. The circuits which have high frequency operation modes, which needs low impedance ground planes in order to avoid common mode current, which needs to be oriented on PCB very carefully should be defined in this chapter. This would inform about signals, circuit demands and performance criterions of the circuits. A circuit with reduced EMI is a perfect design if only the circuit works properly. Therefore it is necessary to know the blocks and interconnections between these blocks for the perfect design.

On a television PCB, a main IC which is generally called concept is accomplished. This is the most critical part of television where all signal processing is performed and useful data to LCD panel is sent. All other parts are controlled and communicated with main IC.

Supported peripherals of a television is listed below. This list is in general form. While technology in TVs is improved day by day, more peripherals should be added in the list after the date of publication of this thesis.

- RF input VHF1, VHF3, UHF @ 75Ohm
- Side AV (SVHS, CVBS, HP, R/L_Audio)
- SCART sockets
- YPbPr
- PC input
- HDMI
- Stereo audio input for PC
- Line out
- Subwoofer out
- S/PDIF output

• Side S-Video
• Headphone
• Common interface
• USB
• RS232
• Smart card connector

In stead of all interfaces, EMC critical peripherals are going to be observed detailly. Also high frequency parts of televisions except peripherals just like DDR Rams are going to be examined. Because mostly design rules are created according to EMC critical parts. Design procedure includes much more about high frquency circuits rather than low frequency and analog parts. That doesn't mean there shouldn't be a problem due to analog signals. All pcb signal routing rules should be implemented for all circuits. For instance, many problems due to audio signal had been seen in practical.

Firstly a general block diagram of a television is going to be illsutrated. Block diagram does not include all circuits but emc critical circuits are going to be shown. After that parts like LVDS, HDMI, USB and etc.. are going to be investigated one by one. Internal block diagram and signals with high frequency are going to be mentioned.

Below a diagram of a general television concepts is given. Then details of EMI critical parts will be investigated. With the mixture of this information and EMC testing expressed in previous chapter, design considerations will be created.

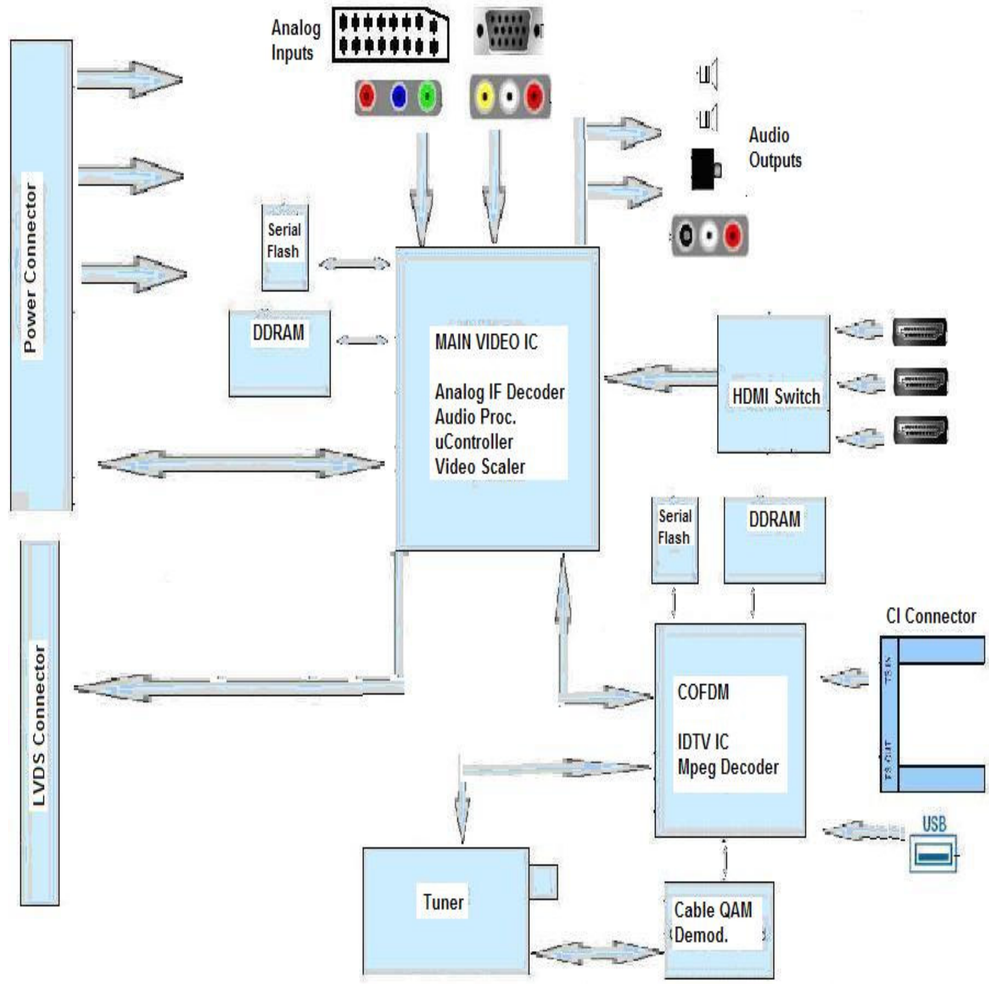


Figure 4.1 General block diagram

There are some specific circuits which have a high probability of radiating EMI. These circuits are going to be investigated detailly in the manner of EMC. The sub blocks are listed below:

- Power stage
- Tuner
- IDTV (Integrated Digital TV)
- Digital Common Interface (CI)
- LVDS (Low Voltage Differential Signal)
- HDMI (High Definition Multimedia Interface)
- USB (Universal Serial Bus)

- VGA (PC input)
- DRAMs, Serial Flash and EEPROMs

4.1 Power Stage

This is one of the critical parts of radiating EMI. Because as it is seen in Figure 4.2, different supply voltage are produced from the voltage levels come from power supply unit. These supply voltages is distributed to all circuits on PCB. Therefore if there is an EMI or dirty supply voltage, it should effect all over PCB.

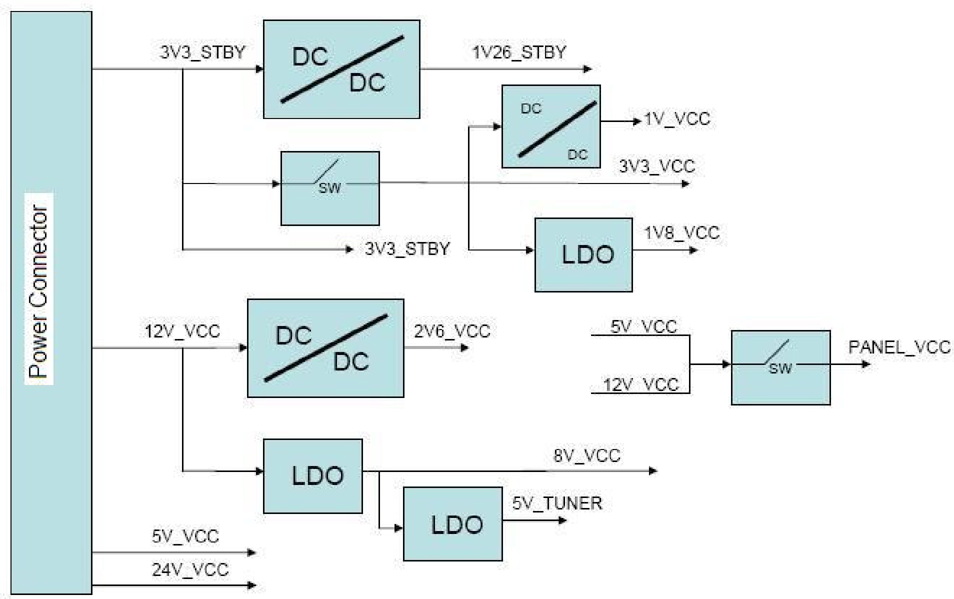


Figure 4.2 Power distribution on a television

This is not the only solution of providing power stage in a television mainboard. It depends on the project itsself. Because if there are other ICs that use different voltage levels expect stated ones in Figure 4.2, there should be a design to produce those voltage levels.

In this design, 3V3,12V,5V and 24V comes from power connector. This supply voltage levels should be low impedance while they are generally processed in power board. Different voltage levels are obtained from these supply voltages by using step down converters and regulators. Step down converters are critical EMC components, while they are inductors in basics. Inductor structure radiates EMI and if there is a cable nearby step downs, it has the possibility of couple EMI to cable which would behave like an antenna.

These supply voltages are distributed to all over PCB. Therefore a security region between differnt supply voltages should be provided in order to the possibility of coupling. Further design rules about supply voltages are given in Chapter five.

4.2 Tuner Section

Tuner is the part that receives broadcasting and gives the system IF signal which is a modulated signal containing video and audio information. As it comes from its name 'tuner', it gives an accurate total signal level for the required broadcasting vy tuning at wanted frequency. The reception of broadcasting signal and providing Analog IF signal which contains CVBS (video) and QSS (audio) information is performed inside tuner. The remaining parts are given in Figure 4.3

Modulated signal generally goes to a filter called SAW (Surface Acoustic Wave) filter before it is demodulated. Filtering and impedance matching of received signal is performed in SAW filter and then the process contains IF demodulation and decoding part.

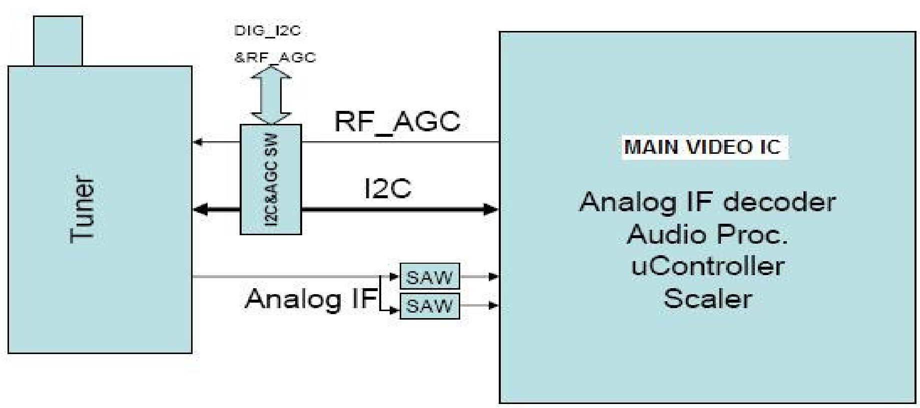


Figure 4.3 Tuner section for analog broadcasting

IF demodulation and decoding inside main IC. There are other control busses seen as I2C in the block diagram. But this is not critical for raditaing EMI while no high frequency just control bits are considered in this part. Demodulated QSS signals are proceed in audio processor and then feeds speakers and other audio output ports. Demodulated CVBS signals are proceed in main (video) IC and then feeds LCD panel over LVDS lines with useful data.

4.3 IDTV (Integrated Digital TV) Section

In recent times, digital broadcasting is much more preferred according to analog broadcasting because of picture quality and signal transmission. Therefore most TVs provide digital broadcasting with built-in structures.

This structure is mostly similar with analog broadcasting. But in this structrure, Digital IF signal is provided by tuner and this signal is processed by another IC named Cable QAM (Quadrature Amplitude Modulation) demodulator in Figure 4.4

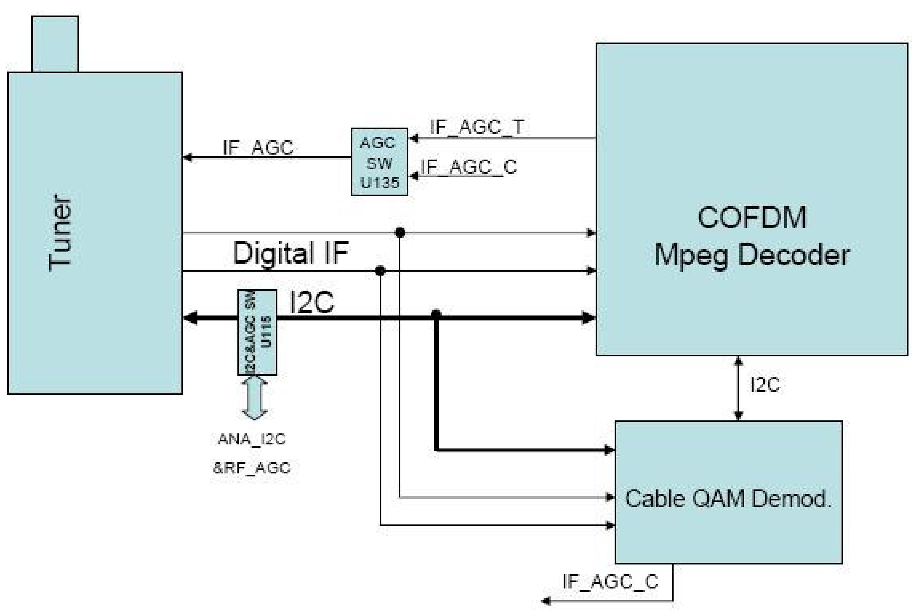


Figure 4.4 Digital broadcasting

The signal comes from tuner is demodulated after filtering and impedance matching process. If the video is mpeg decoded which is a type of coding that increases picture quality, that is decoded by Mpeg decoder and then send to main video IC. Audio and video signals are sent to main video IC after demodulation and decoding stage. Useful data which is processed by main video IC is sent to LCD panel over LVDS lines.

4.4 Digital Common Interface (CI)

This section has the function of providing broadcating which is coded. In recent times, it is very popular to supply channels with money among TV-channel owners. They give a special card for the users that pay the money to receive broadcasting. That card has some standarts all over the world. That means types of cards are same in mechanical but coding of each is different. That type of cards can be used in Common Interface (CI) socket.

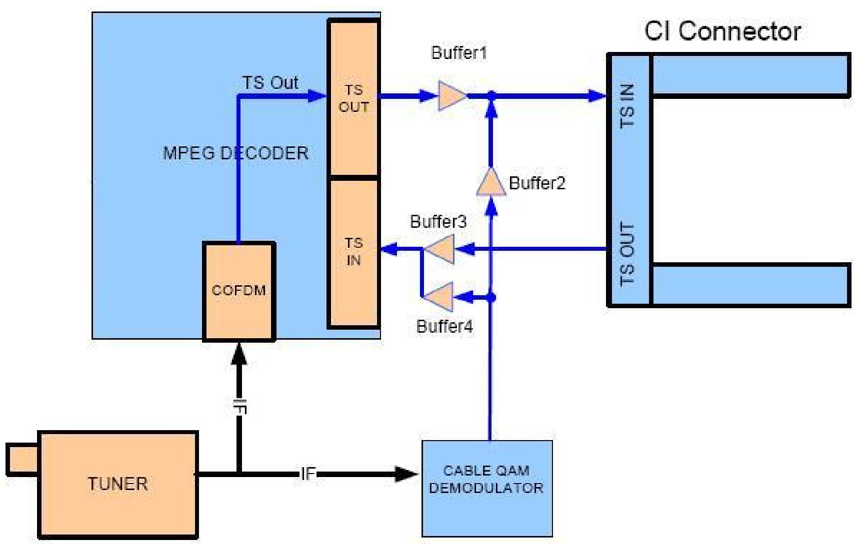


Figure 4.5 Common interface and smart card interface

Received signal from tuner is coded and it is necessary to use CI card which is obtained from TV-channels by money. CI card contains required decoding datas which is also used and communicated by Mpeg decoder. Decoded signal is sent to main video IC. Audio and video signals are sent to main video IC. Useful data which is processed by main video IC is sent to LCD panel over LVDS lines which is similar in above sections. Digital broadcasting and CI interface block diagrams are related with main video IC like in Figure 4.6

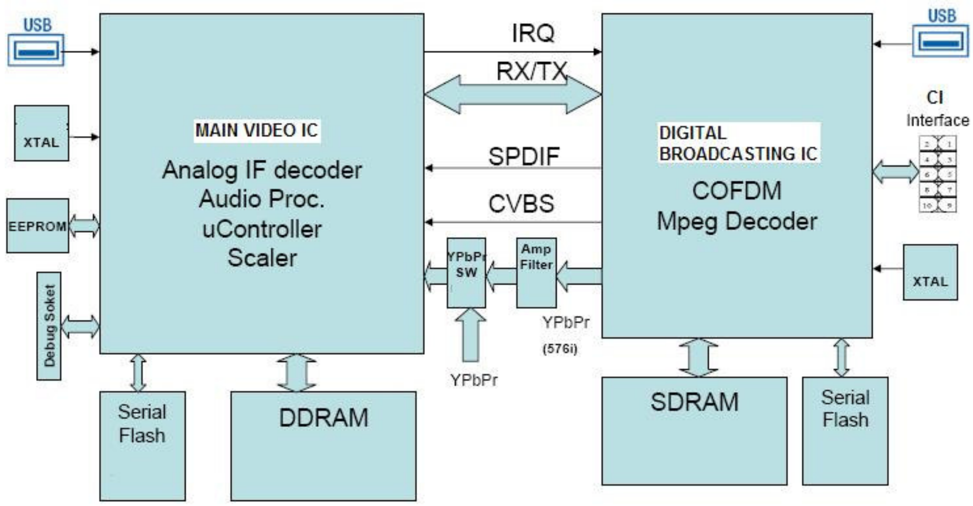


Figure 4.6 Digital broadcasting and CI interface connection with main IC

4.5 LVDS (Low Voltage Differential Signal)

LVDS is a technology of high performance data transmission application. This signalling method can be used in all high performance data transmissions. But in television, it used to give required data to pixels on LCD panel. It means, it is used to provide the picture on television by sending data to each pixel on television. LVDS delivers high data rates while consuming significantly low power. Therefore this system is egnerally called ‘Gigabits @ miliwatts’. (National Semiconductor, 2004)

It has many advantages such as low voltage power supply compatibility, low noise generation, high noise rejection, and robust transmission signal.

But this is one of th most critical parts for EMI design while high frequency of signal transmission is performed. Signal routing should be very carefully done.

LVDS is differential, using two signal lines to convey information. This is actually a benefit in cost. It uses two traces to convey a signal and it requires too short rise time which enables faster data lines.

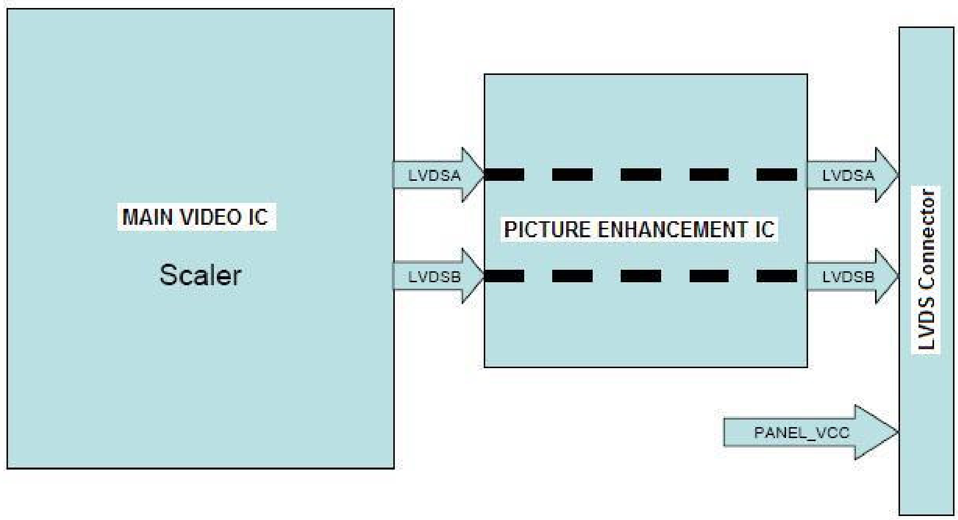


Figure 4.7 LVDS Interface

LVDS output consists of a current source that drives the differential pair lines. The basic receiver has a high DC input impedance, so the majority of driver current flows across termination resistance generating about 350mV across the receiver inputs. When the driver switches, it changes the direction of current flow across the resistor, therefore creating a ‘one’ or ‘zero’ logic state.

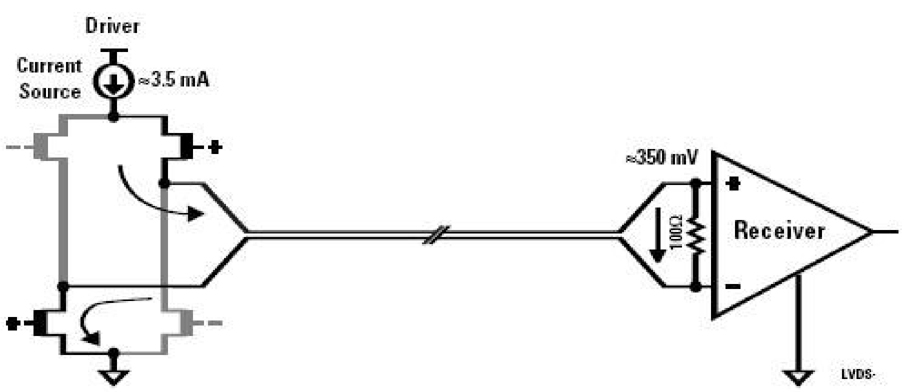


Figure 4.8 How LVDS works

LVDS data ranges from 100MHz to 2 GHz. At these frequencies, PCB can no longer be simple traces. Traces carrying these high-speed signals need to be treated like transmission lines. These transmission lines should be designed with appropriate impedance and they need to be correctly terminated for reduced EMI and signal integrity.

There are two major considerations of LVDS design for reduced EMI. One is eliminating coupling between the traces of each pair and second is minimizing imbalances between the conductors of each pair. (National Semiconductor, 1998)

To eliminate coupling between two pairs, it is necessary to keep the distance between the pairs as minimum as possible. Since differential pair is a current loop as defined in previous paragraphs, this reaction should reduce the aera of antenna loop. It is also important to minimize imbalances in order to reduce EMI. The impedance of signal traces should be well controlled. If the impedances of two trace in a pair are different, it should lead to an imbalance. The voltage and fields of one signal will be different from its pair. This should create more field strength and therefore more EMI. The idea of design rules about LVDS which is going to be shown in the next chapter is based on these two purposes.

4.6 HDMI (High Definition Multimedia Interface)

HDMI is a digital interface for both audio and video signals designed as a small-socket, single-cable solution for home theater and consumer electronics equipment. HDMI is electrically identical to DVI, which is video only, and an HDMI source can detect a DVI device on the other end and switch to the DVI protocol.

HDMI supports a TV video format, including enhanced and high-definition video, additionally to multi-channel digital audio on a single cable. HDMI is a critical issue of EMI management while ITE tests should be done in HDMI mode. That means the source of input signal is HDMI instead of tuner or VGA signal. Test can be performed while a DVD player connected to television by HDMI input. And integrated TMDS (Transition Minimized Differential Signaling) receiver and transmitter cores capable of receiving and transmitting 2.25 Gbps. That means a high speed of serial data is transmitted in HDMI mode. Therefore care should be taken during HDMI signal routing.

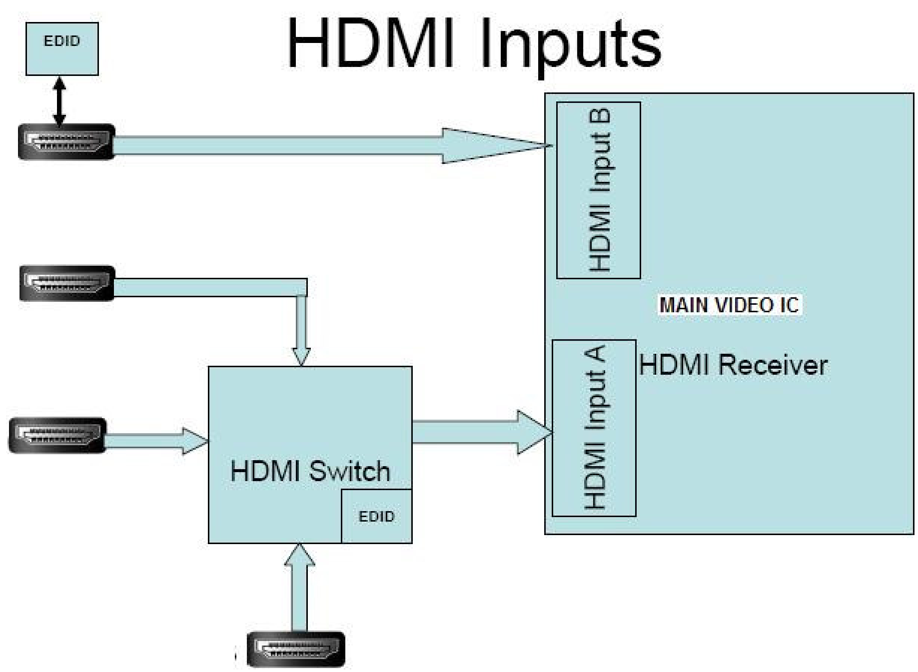


Figure 4.9 HDMI switching and HDMI receiver diagram

With the help of these high speed serial data, higher resolution video performance can be obtained. This provides a high definition picture quality.

4.7 USB (Universal Serial Bus)

This is a well known data transfer method but it is included in television technology in recen times. First generation of USB (USB 1.1) specifies 12 Mbit/sec data transfer rate at full speed mode.

Later on, full speed rate was no longer sufficient due to increasing processing speed such as televisions. Thus USB Implementers provides second generation USB

(USB 2.0) with data transfer rate up to 480 Mbit/sec. At this time, EMI problems should occur during a communication of USB with main IC in a television. USB port is used to view pictures, play tracks in television. As the technology and customer demands increase, USB technology gets involved in televisions. Block diagram is represented in Figure 4.6.

4.8 VGA (PC Input)

VGA is a traditional signal tranfer method but the importance for EMI design during a television comes from the test methods. Because when a television consists of a VGA input, it can be used as a monitor which a PC or laptop can be connected. During the use of monitor, it differs from audio/video equipment with broadcasting and it is respected as a IT equipment. As it is stated in previous chapters, IT equipment has its own standard which is EN 550022 and testing procedures. A television which tries to have compliance in European market should satisfy and meets the requirements of EN 55022. Below VGA interface and other analog audio/video interfaces are shown.

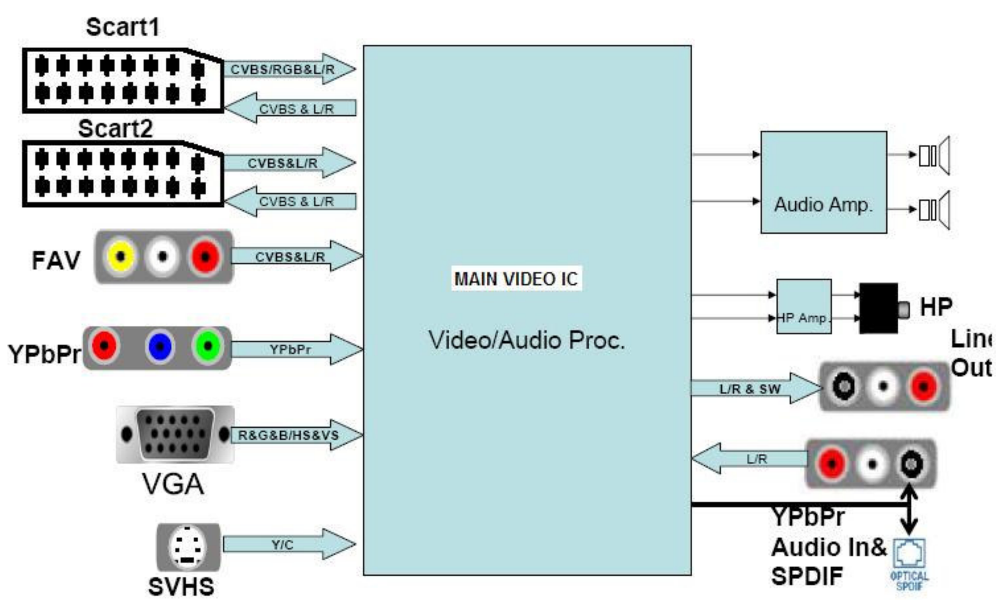


Figure 4.10 Analog inputs and outputs

Although the signal in VGA mode is analog, it's critical for EMI design with its own standard. A computer or laptop is connected to television during the testing according to EN 55022.

4.9 DRAMs, Serial Flash and EEPROMs

These components are used for storage of temporary data and software used by main IC. They are used for the same purpose with personal computers. They are considered in EMI design rules while a high speed data transfer is performed. Especially DRAMs has the capability of fast clock rate. This provides a high speed of working but it indicates a problem during design stage. Very special considerations have to be taken during these data lines routing. The frequency of clock rate varies from different suppliers but it can be up to 170 MHz. Data and address lines of these memory devices have higher data transfer rates up to 300 MHz. Harmonics of clock rates are also potential problem in EMI design. These components are used with main video IC and digital IC which is shown in Figure 4.6.

Clock routing of these components should be done very carefully. This is also indicated in the next chapter. Details will be given in the design procedure.

CHAPTER FIVE
DESIGN PROCEDURE FOR REDUCED EMI

The design rules dedicated in this chapter is just related to prevent EMI radiated from PCB. The criterion is EMI. But in a television design stage, all design procedure given in this chapter can not be followed one by one with different restrictions such as proper work of all circuits together, cost, mechanical problems and etc.. It is an engineering work to find optimum application level of design rules. It is possible to decide weight for each rule and examine maturity of design for low EMI radiation at the beginning of the design.

The design procedure given in this chapter should guide any television design with reduced EMI. This design procedure includes general layout and wiring techniques. These design techniques is prepared generally for televisions but they can be also used or converted to any electronic equipment which has analog and/or digital communication. These design rules can be implemented to project after suitable theoratical calculations because of proper operation of television.

Design procedure is going to start with orientation of blocks defined previous chapter on a PCB. This is the most critical and hardest part of the design. Because there are several factors which influence orientation of PCB such as mechanical, industrial, customer demands and others besides EMI. This part has to be applied with optimum level. Because for instance it's hard to determine LVDS connector coordinate on a PCB just based on EMI concept. There are several restrictions for LVDS connector. After decision of PCB orientation, routing techniques is going to be given. Applying signal routing techniques is a little bit easy rather than orientation. Limitations are related to PCB layer and size. For instance when intersection of two signals is mandatory, one of the signals has to be passed to other layer wih a via and then routed. And after PCB routing techniques, lastly design rules for cabling between intersections of television. The hardness of this part is mostly related with cost and procurement of cables.

5.1 PCB Orientation

This part is related with how the subcircuits which have its own functionality should be placed on an empty PCB. This orientation should eb decided according to defined criterions below before signal routing. Before a PCB design, all functional groups have to be determined. Connection of all supply voltages and grounds of functional groups has to be star distribution from one point. This shouldn't be forgotten. High frequency signals have to be placed and routed at first. Second supply voltages and laslty analog signals have to be placed and routed.

The chronological list of design procedure related to PCB orientation:

1. Firstly, functional groups which will be on the PCB should be determined.
Exp: HDMI, LVDS, Audio circuits, RF inputs, Power stage

2. Supply voltages and operating frequency of these functional groups should be listed.
Exp: HDMI ➔ 3.3V ➔ 27.00 MHz
LVDS ➔ 5.0V ➔ 75.00 MHz
IDTV ➔ 2.8V ➔ 33.15 MHz

3. According to order below, it should be determined where the functional blocks are oriented on the PCB.

- High Frequency Signals
- Supply Voltages
- Analog and low frequency signals

After a quick overview of orientation according to above statements, below design rules related to PCB orientation given with detailed theoratical information should be applied to the system to obtain reduced EMI.

- The most important criterion in EMC design is the characteristic impedance of PCB medium. This is related with the amount of PCB layer. There should be a signal layer and supply/ground layer on a PCB.

But a television consists of lots of signal that must be routed. Therefore in two layer PCBs, it becomes mandatory to use low impedance supply/ground layer for signal routing. This increases the impedance of layer and also possibility of noise coupling from signal traces to supply voltages. But in four layer PCBs, one layer can be used as totally ground which help backcurrents resturn in the shortest way and not produce common mode current for the generation of EMI. Two layers can be used for signal routing and one layer for supply voltages. This helps the reduction in the total spectrum of EMI.

Below, a design with two layer and a design with four layer PCB is given. The only difference between two PCBs is the addition of one total ground layer and one supply layer.

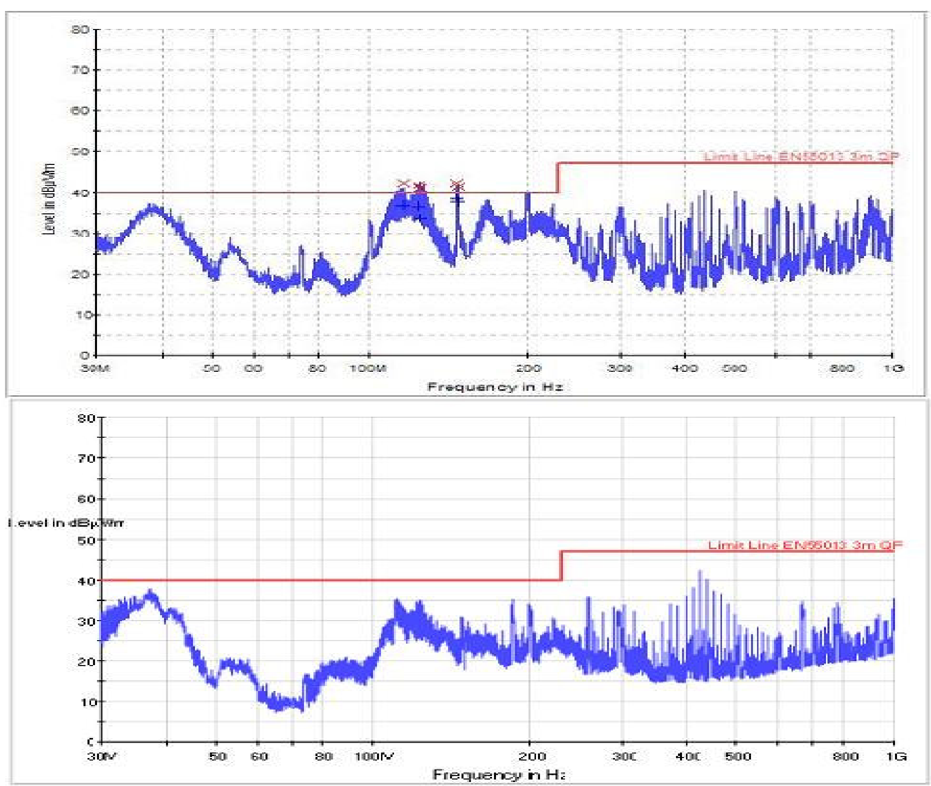


Figure 5.1 Above two layer PCB result and below four layer

• To prevent crosstalks, overlaps and interaction between different supply voltages, PCB should be seperated for different voltage regions. For instance when there are functional circuits which have 5V supply voltage, these should be oriented in 5V region just like given in the below figure.

If there are some ICs which uses different supply voltages, orientation of these Ics and their components should be decided according to operation frequency. For instance an IC with 1.8V supply voltage which has a high frequency operation is connected to main IC. It also uses tha same 5V with main IC. At this condition 1.8V IC should be located in 5V region due to high frequency operation while high frequency signals should be routed as short as possible.

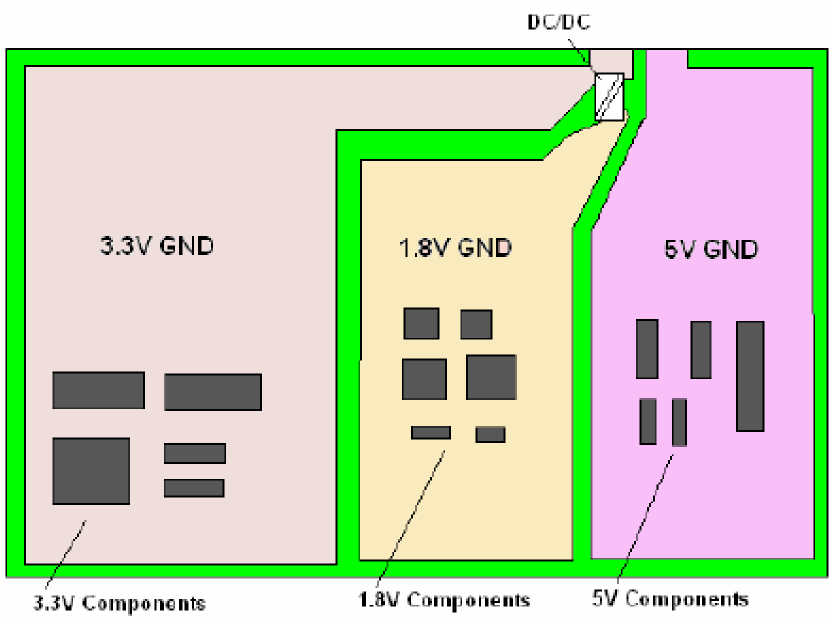


Figure 5.2 Different supply voltage regions to prevent crosstalks

For those kind of situations, optimum solution has to be discovered considering overall system rules.

• Back currents of high frequency circuits should be controlled and defined in the shortest way where common mode radiation mostly occurs. To define back current, all connection of PCB to metal chasis shoulcn't be with conductive bolts or screws. If they are all electrically connected to metal chasis, it is hard to define back currents of high frequency signal and interactions, crosstalks between functional groups should occur. This is defined with given figure below.

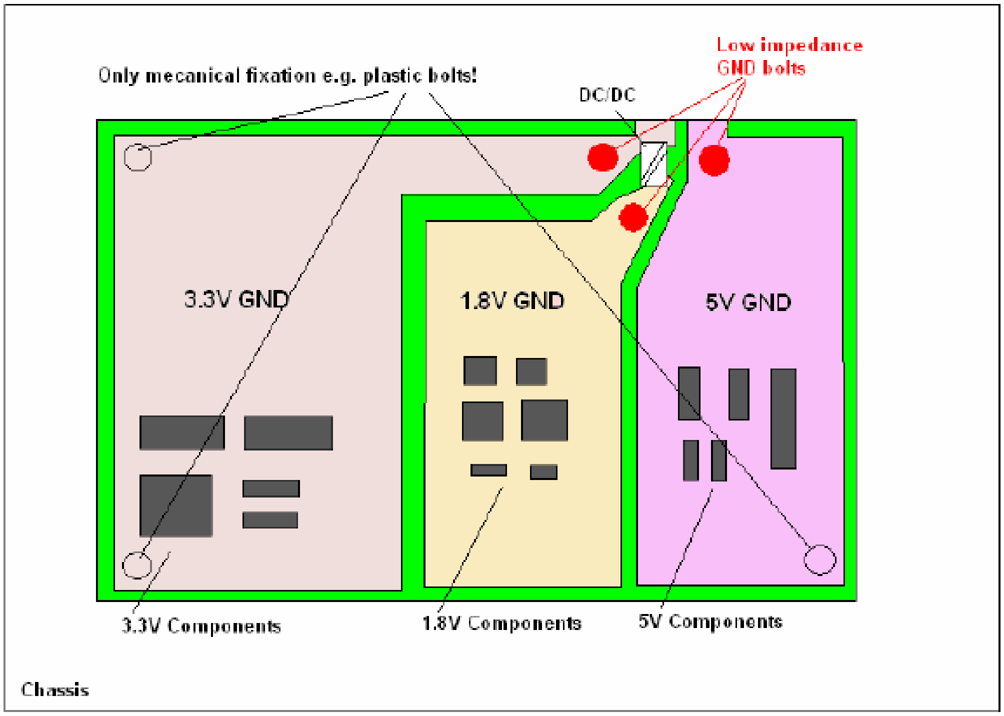


Figure 5.3 PCB connection with metal chasis to define back currents

It is better to connect electrically PCB to metal chasis with the red points given in Figure 5.3. If red low impedance GND points are connected to metal chasis, it is easy to define back currents of different supply voltage circuits inside its region just like in Figure 5.4. While high frequency signals which are also critical with their back currents are located close to DC/DC, it is also useful for high frequency circuit back currents. But when all connections of PCB to metal chasis provide electrical contact, control of back currents is getting harder. And at that point interactions between different functional groups, crosstalks, common mode radiation should occur just like in Figure 5.5.

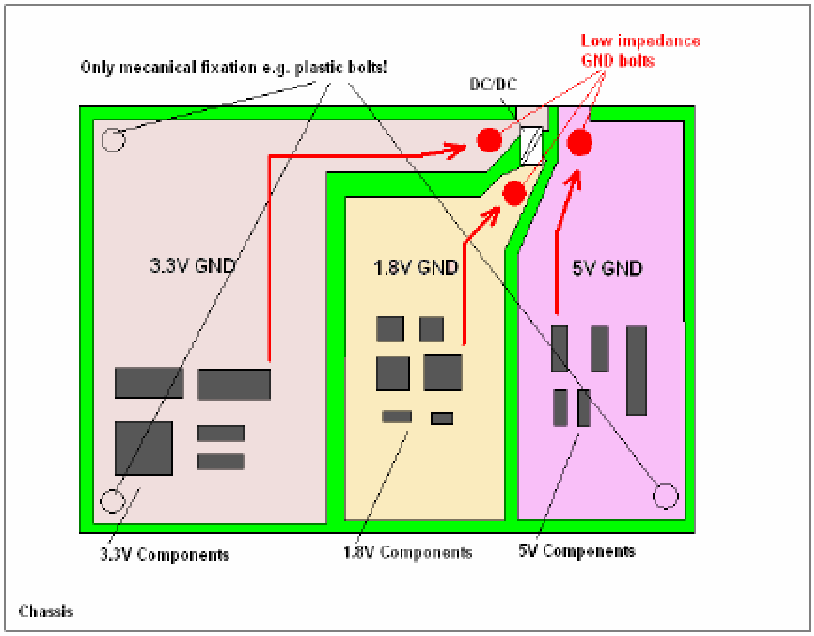


Figure 5.4 Defined back currents

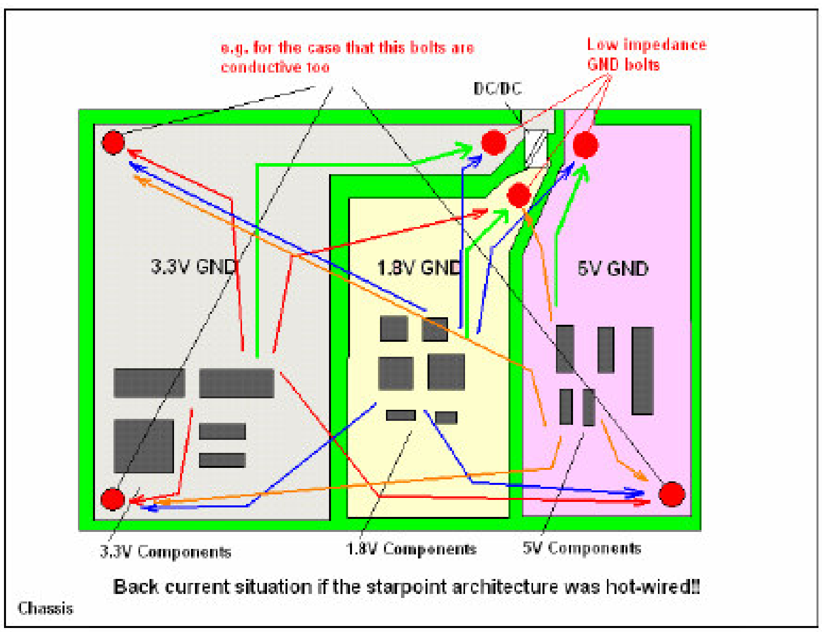


Figure 5.5 Undefined back currents

- High frequency signals inside its supply voltage should be located closest to low impedance GND part of the system which is power stage of PCB. It should be followed by lower frequency signal circuits and then lowest as stated in Figure 5.6

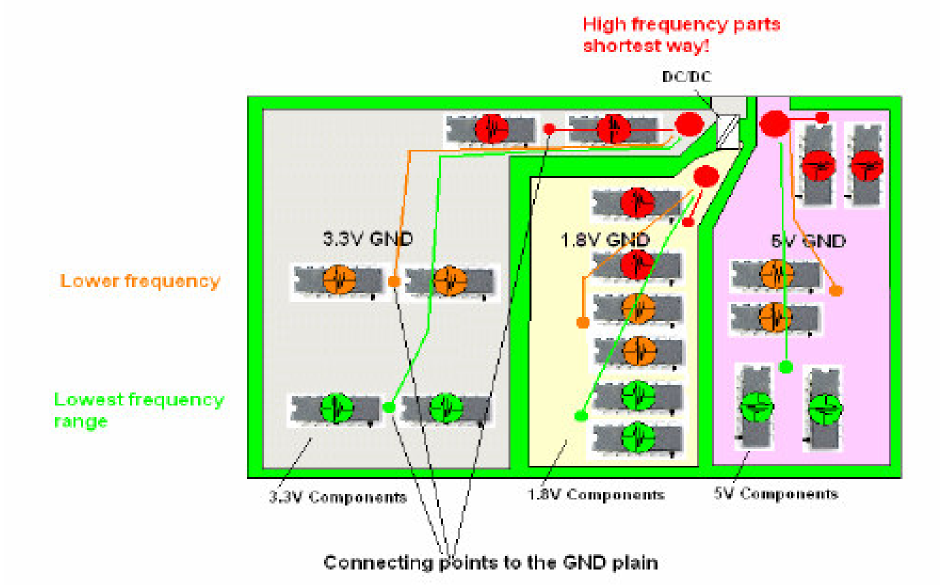


Figure 5.6 High frequency circuits closest to low impedance GND stage.

This should achieve the system minimum radiation. Because while high frequency signals should have low impedance GND not to create common mode radiation. Unless high frequency goes to GND, it should effect other circuits and should create more EMI. Therefore it has to be located to low impedance GND stage as close as possible.

- There shouldn't be impedance change cause of geometrical at the connection of same supply voltage regions. During the transition of same voltage region, surface shouldn't become too narrow. It is defined below figure.

This will result impedance change between same voltage regions. By impedance change some reflections for back currents may occur and this wil create unwanted EMI and also crosstalk.

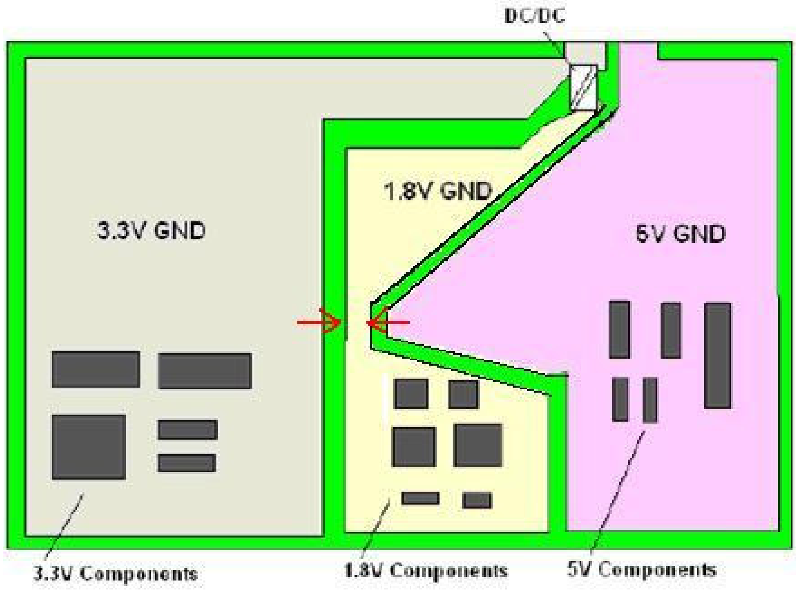


Figure 5.7 Geometrical change created impedance change

- To prevent crosstalks, interactions between fucntional groups like USB, IDTV, etc.. isolation should be provided. That is obtained by seperating GND planes under functional groups seen in below figure.

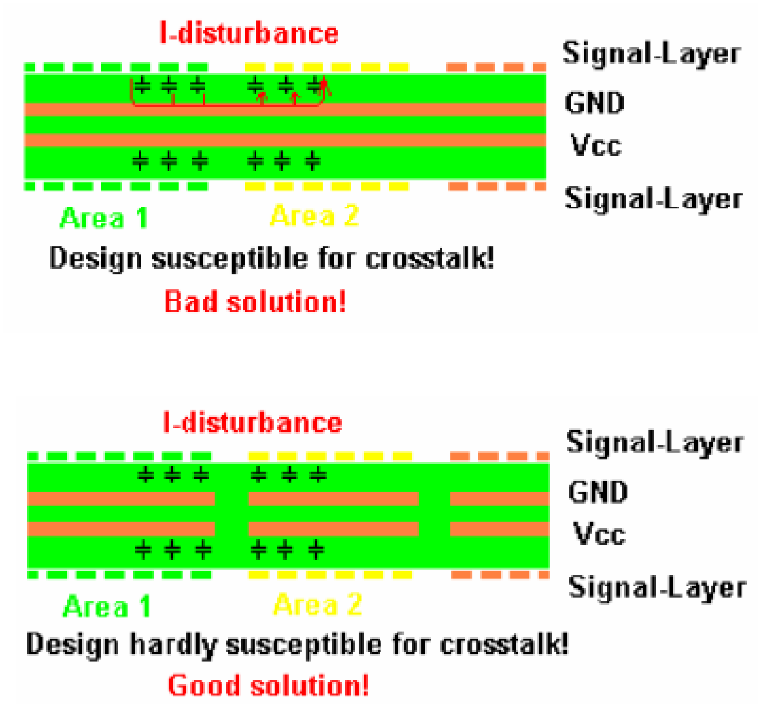


Figure 5.8 Isolation between functional groups

If GND plane under functional groups on below layer is also seperated, EMI should be limited. Because ther shouldn't be any interaction between circuits and therefore there is no imbalance in back current and common mode current.

- Different voltage regions at different PCB layer shouldn't overlap eachother. This will increase the interaction and crosstalk between different groups.

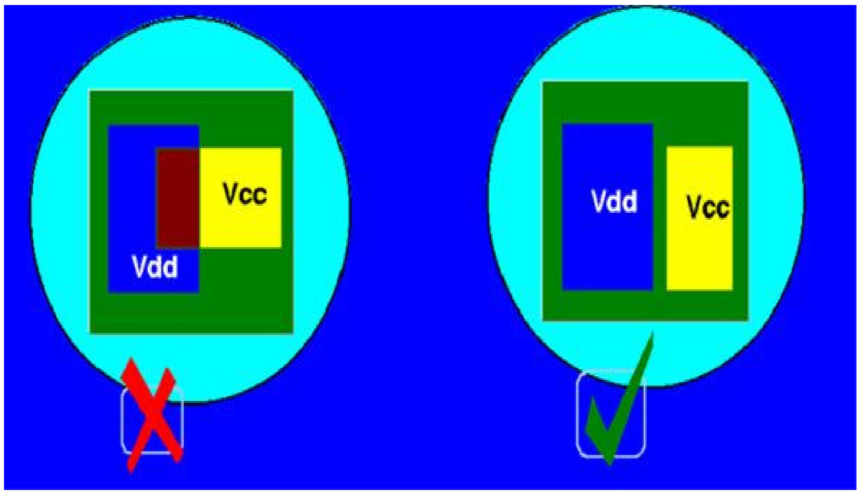


Figure 5.9 Overlap of different voltage regions

- Clock signals should be close to GND layer of the PCB. Clock signals are the most critical signals related to EMI. Therefore they should be carefully oriented and routed. To give closest way for the back current of clock signals, they should be located to low impedance GND layer of the PCB.

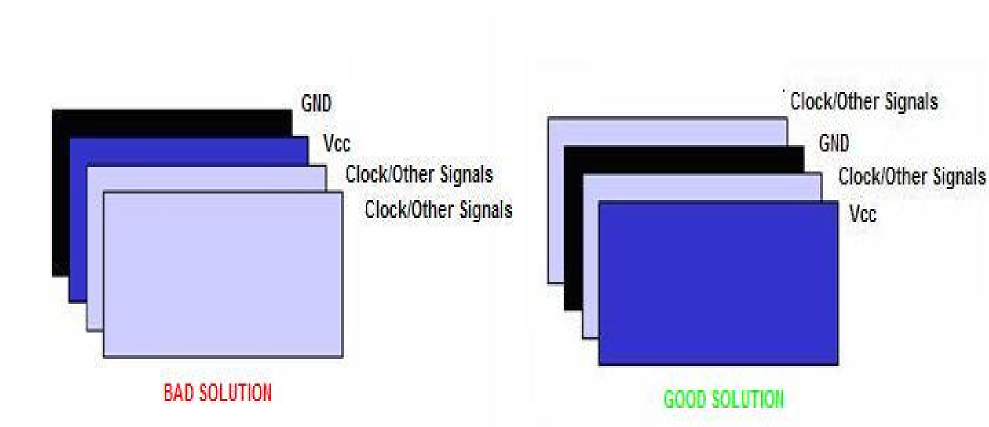


Figure 5.10 Orientation of clock signals

• GND connections of signal layers to GND layer should be high in number and homogenous. This will decrease the impedance of GND at signal layer.

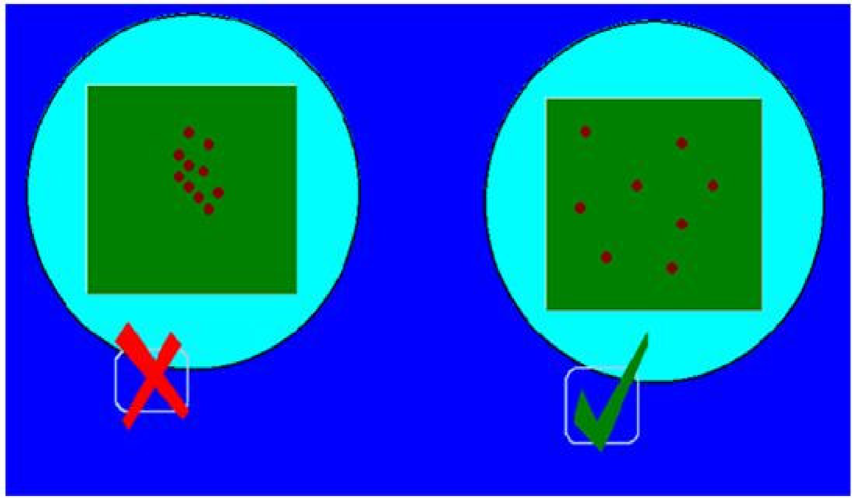


Figure 5.11 Homogenous GND connections decrease the impedance

• If there are several layers PCB available, two layers should be just only used as low impedance layer for Vcc and GND. This wil decrease ripple and EMI unless any functional circuit signal passes through these layers. With the construction of low impedance GND layer, every signal can find its back current path in the shortest way.

• If there are special regulators used for specific IC, for instance IDTV, HDMI IC, these should be located close to related ICs. This will provide a supply voltage without ripple or EMI to IC and reduce EMI of the functional block.

• All filter circuits for audio and video output should be located nearby the connector. Because as it’s stated in previous chapters, while performing test according to EN 55022, all ports are terminated with suitable cables. If there is an EMI radiation from any port, termianation cable will act like an antenna and increase EMI of the system. If the filter circuit placed nearby the connector, EMI can be prevented before signal faces with termination cable.

• All crystals should be located nearby related ICs. This will prevent long traces of clock signals and big EMI radiation from those signals.

• Any component or connector shouldn’t be located close to DDR traces. Because there is a high frequency operation in DDR traces and radiated EMI from these traces can be coupled to other component of another block or to connector and cable behaving like an antenna. This will seriously increase EMI level of the system.

- Oscillator traces should be located seperately from other traces, connectors. This is also a precaution of coupling of EMI to other systems just like above rule.
- Capacitances of crystals should be placed close to related IC reference ground. This will provide a pure clock signal without ripples to the IC. Ripple in the clock signals may hardly cause EMI.

Above rules are all related with PCB orientation. As it is stated before, this is the hardest part of the design. Sometimes it is necessary to find optimum solutions. Next step is related with signal routing. Those are more detailed and specific rules while they indicate how a trace of functional blocks stated in chapter four.

5.2 PCB Signal Routing

After the decision of orientation on PCB according to above rules, signal routing can be performed according to below design procedure and rules. And precautions should be taken for expected problems. Some of below design rules are explained detailly in theoratical and some of them in practical with test results.

At all television PCB designs, main aim is to isolate current conducting traces between eachother and decrease coupling impedances. Coupling mostly occurs because of high current and voltage changes. For this situation, it is necessary to decrease line impedances. Line impedances can be minimized by short trace length, wide trace width, big trace surface and small current loops.

Generally routing should be performed in below order;

- High frequency signals routing
- Vcc-GND routing
- Signals routing according to decreasing frequency
- Analog signals routing.

After listing of signals in order to route, below rules should be followed;

• All GND structures should generate a closed loop and be terminated. Unterminated GND structures shown in red lines in below figure would behave like an antenna. Back current of function groups would intend to follow this GND structure but if it's not terminated, connected to GND layer, it would act like an antenna.

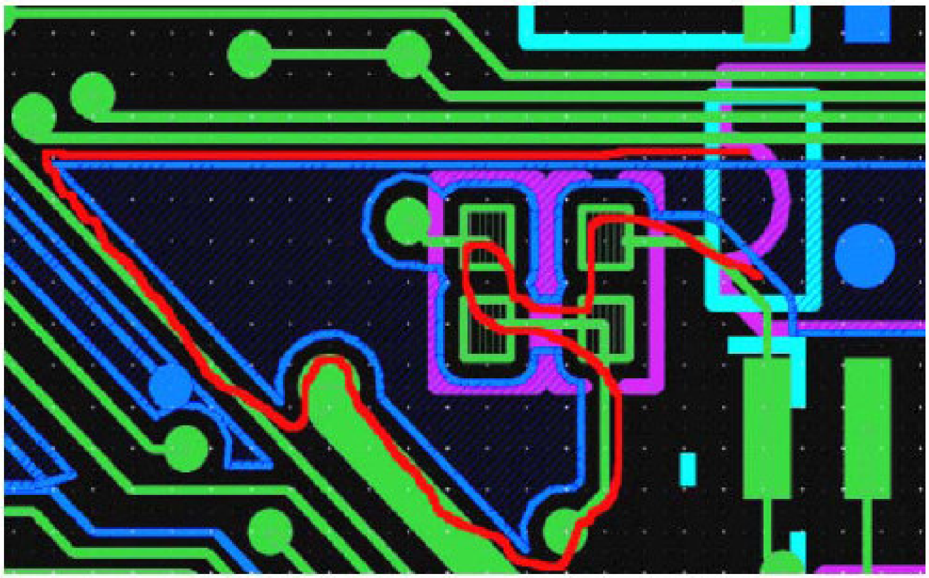


Figure 5.12 Unterminated GND sturctures

• Digital ground ve digital supply must be isolated from other grounds and supplies such as analog ground and supply that are used for the rest of the system. Because EMI can be easily transferred by supply traces while they have low impedance.

• When noisy paths are surrounded by GND, ground paths must be terminated. This have the same idea with above rule. EMI would intend to flow to low impedance ground. If GND isolations are terminated, EMI would not exist.

• For the GND connections of ICs and decoupling capacitors which carries high currents, there should be two or more smallest vias or one bigger via. This wil provide a low impedance GND point. Other wise a via created an impedance change and source resistance which will result EMI.

• Number of vias used to strengthen ground connections on multi layer PCBs, should be as high as possible. Especially the ground paths of regions, that can cause magnetic pollution and that can make ripples on supplies, should be done in this way.

- Tuner grounding is important for signal quality. So for the designs in which the tuner is horizontal, the area below tuner should be open and tuner should touch here.

- The ground paths of LVDS signals must be connected to the ground of IC that drives LVDS.

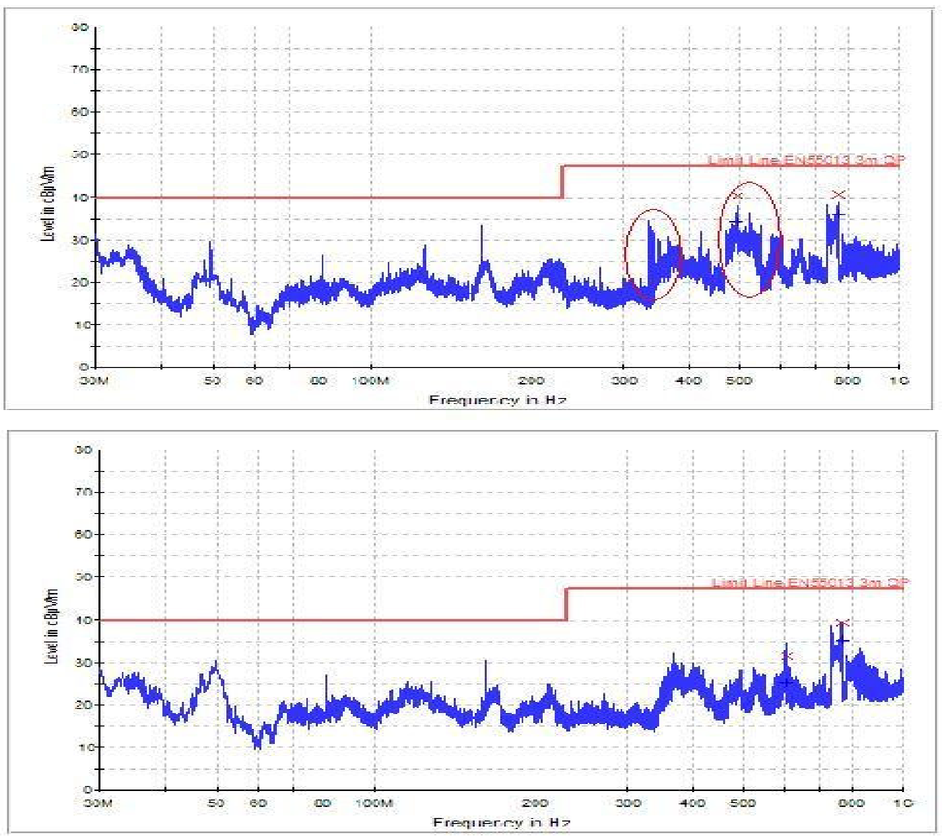


Figure 5.13 When ground paths of LVDS connected to IC ground, emission decreases

In the test results above, it can be easily observed when ground of LVDS signal paths are is connected to ground of IC that drives LVDS, emission decreases. That is because of the defined back current of LVDS signals. Both IC and LVDS has the same reference of ground.

- All of the voltages must be distributed from one main point and sent to ICs as star distribution. Star distribution is more likely to single point distribuiton. It can be thought as collecting all reference levels at one point. That conneciton point should be far away for all related parts and can be routed with wider traces.

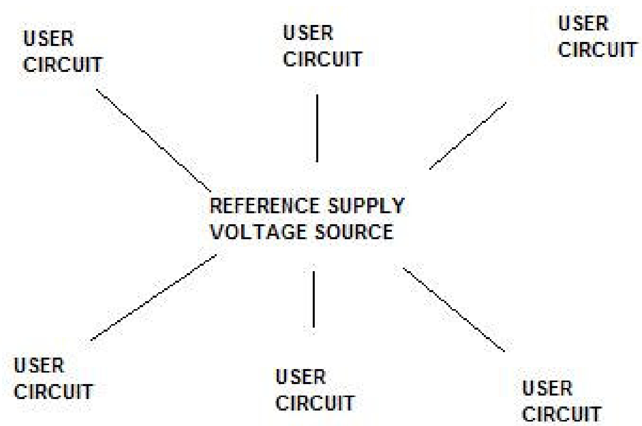


Figure 5.14 Star distribuiton of supply voltage

- High frequency signals like DDR,LVDS,HDMI should not make big cycles. This would bothe increase EMI of the system and reduce immunity to EM waves. Different impedance levels occur at the big loops. With the voltage diference according to these different impedance levels, current also occurs which would create EMI flows in the loop.

- The ground area under high frequency circuits and signals like DDR,LVDS,HDMI must be large enough.

- High frequency signals like DDR,LVDS,HDMI must be terminated.

- High frequency signals like DDR,LVDS,HDMI should be drawn with minimum paths. For instance a high frequency signal shouldn’t be such long as in Figure 5.15.

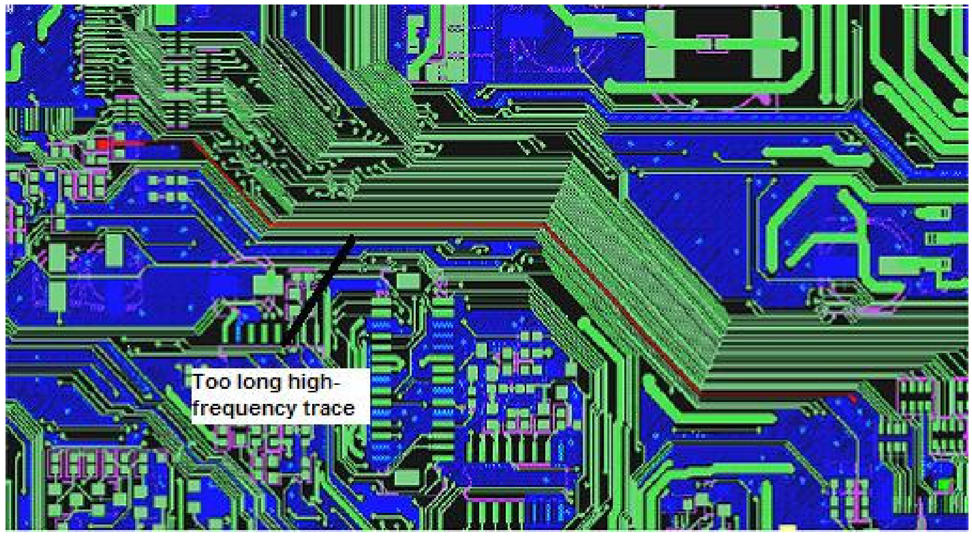


Figure 5.15 Unnecesseraliy long high frquency trace

If the trace in red is a high frequency trace, it becomes unnecesseraliy long and possible to radiate EMI. High frequency traces should be as short as possible. If long is mandatory, a wider GND path should be drawn parallel to high frequency trace. These GND path shoul be connected to GND layer of PCB with more than one via to reduce impedance.

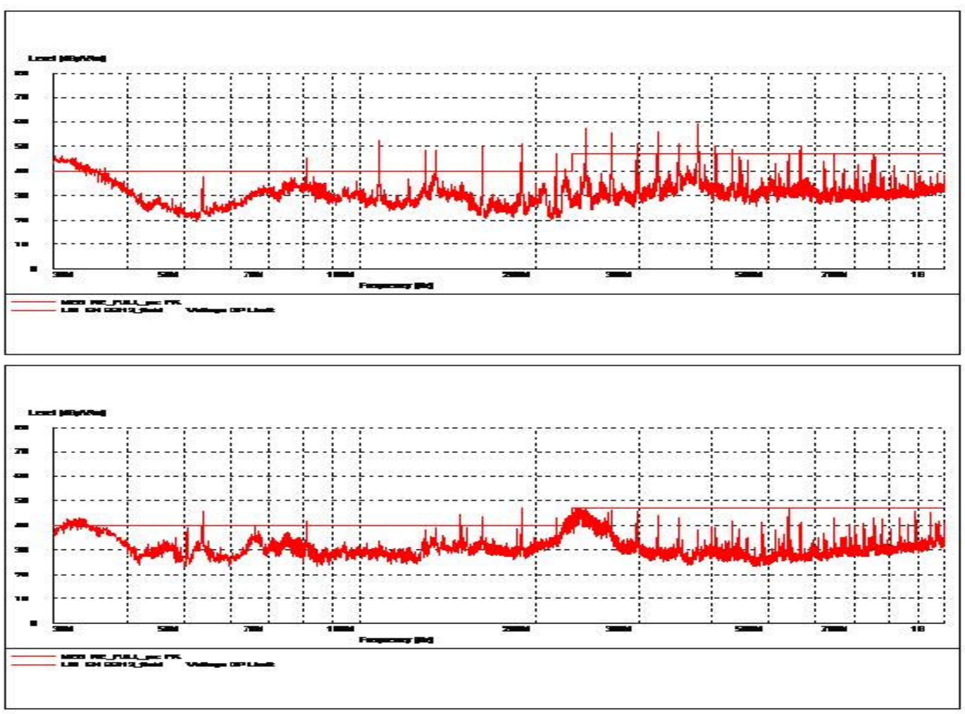


Figure 5.16 Long HDMI traces increase EMI radiation

According to test procedures expressed in previous chapters, television can be tested in HDMI mode. If HDMI traces are routed unnecessarily long, emissions increase because of antenna effect of traces.In this experiment, HDMI mode is performed and after shortening of HDMI traces, most of peak emissions lowered.

- In two layer PCBs, while drawing high frequency signals like DDR,LVDS,HDMI, the ground layer on the buttom layer must be cut into pieces in the regions that these high frequency signals pass.

This method is called ground gridding. It's quite precise method therefore calculations should be made very carefully. This method should be very useful if it is made in the right way. Because ground gridding simulates metal shield on the PCB.

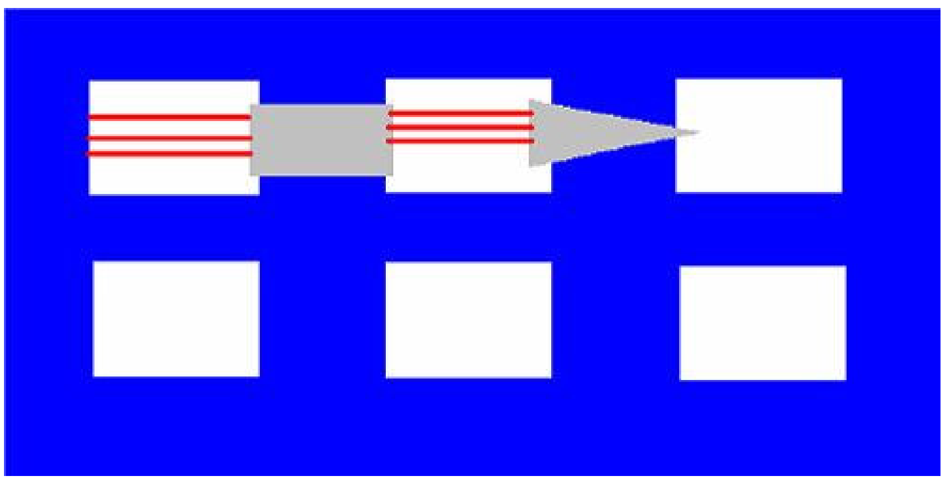


Figure 5.17 Ground gridding method

Layer below the high frequency signals can be cutted into pieces. Length of grids must be smaller that critical length. The distance between two squares must be one fifth of the critical length. Dimension of squres is decided according to highest frequency on the paths. For instance, for a signal at 1 GHz, λ equals to 10 cm and $\lambda/4$ is 2.5 cm. This length is divide by safety factor (3:1) and critical length rquals to approximately 8 mm. Dimension of squares above should be 8mm for a 1 Ghz signal and the distance between squares should be at least 1.6mm. These square should be conneted to GND as high as possible. Otherwise, it would react as an antenna.

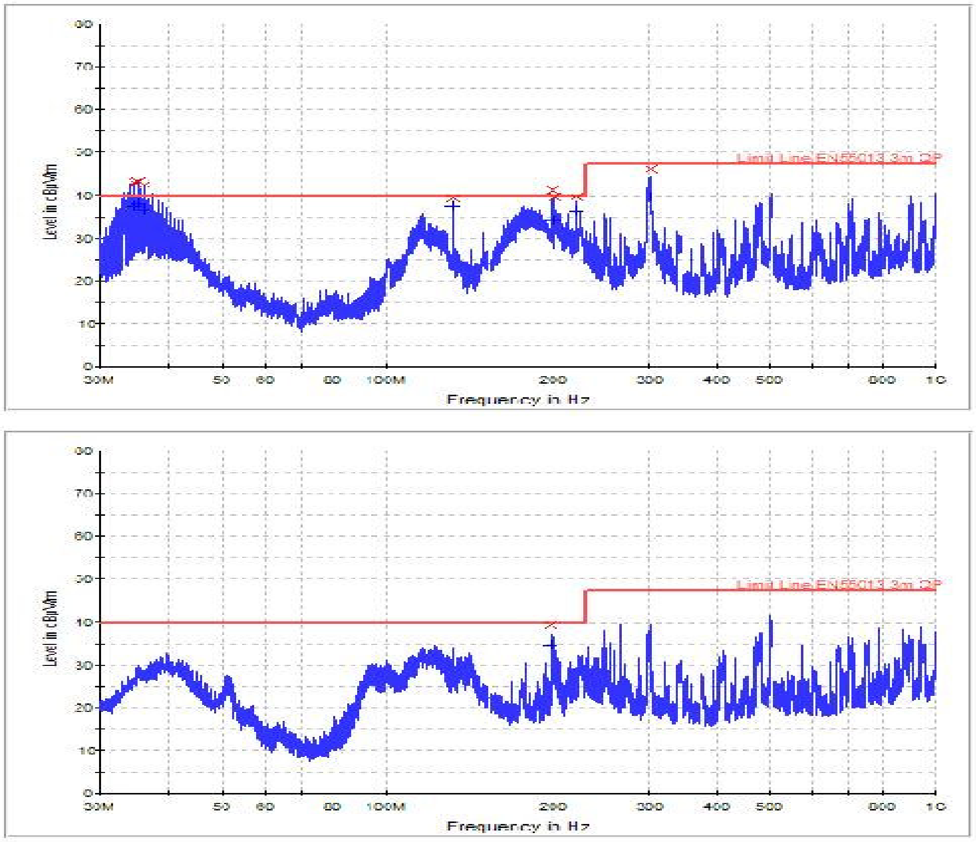


Figure 5.18 Ground gridding has a shielding effect

An emission caused by a functional block in a television at a frequency band 30-300 MHz has been eliminated by ground gridding method. Ground gridding has an effect of shielding EMI as it described above. It is a big risk to remove a complete ground under high frequency signals, but it can be only useful when it is calculated according to above considerations.

- The buttom region of HDMI ve USB signals must be completely ground.

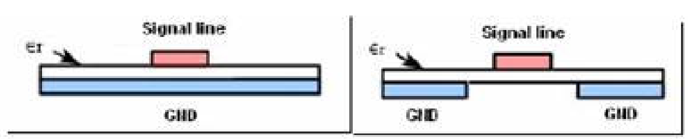


Figure 5.19 Capacitive effect of uncalculated GND pieces

If the GND plane is divided by accidentally without any calculations described above, two GND structure creates a capacitive effect and intend to increase EMI. That is proved with below test results.

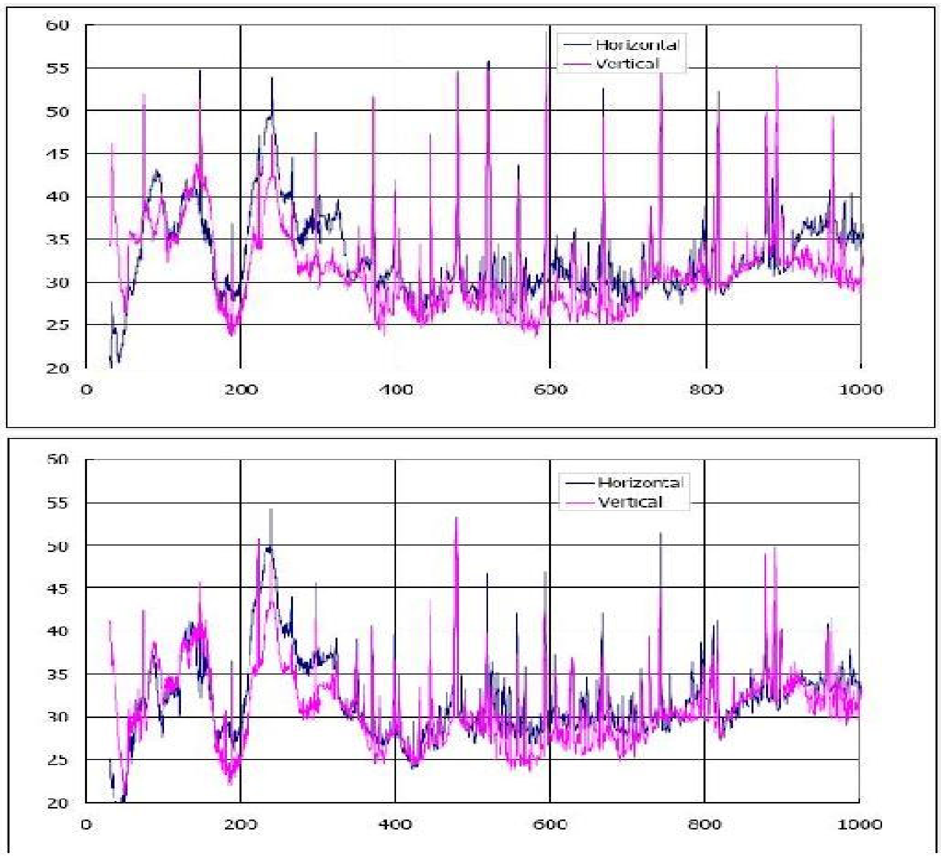


Figure 5.20 Buttom region of high frequency signal should be completely ground

In the test result above, buttom region of HDMI paths of two layer PCB is cutted and complete ground is destroyed without a purpose of ground gridding. That causes backcurrent path loss and back current creates crosstalks to other signals and emission level increases. But when it is grounded completely, emission levels get lower as it is seen in the test results below.

- For high frequency signals like DDR,LVDS,HDMI, there must be serial components on the source side of these signals. There should be a serial resistor of for instance 100 ohm to decrease EMI. Level of wanted signal should be taken care of.

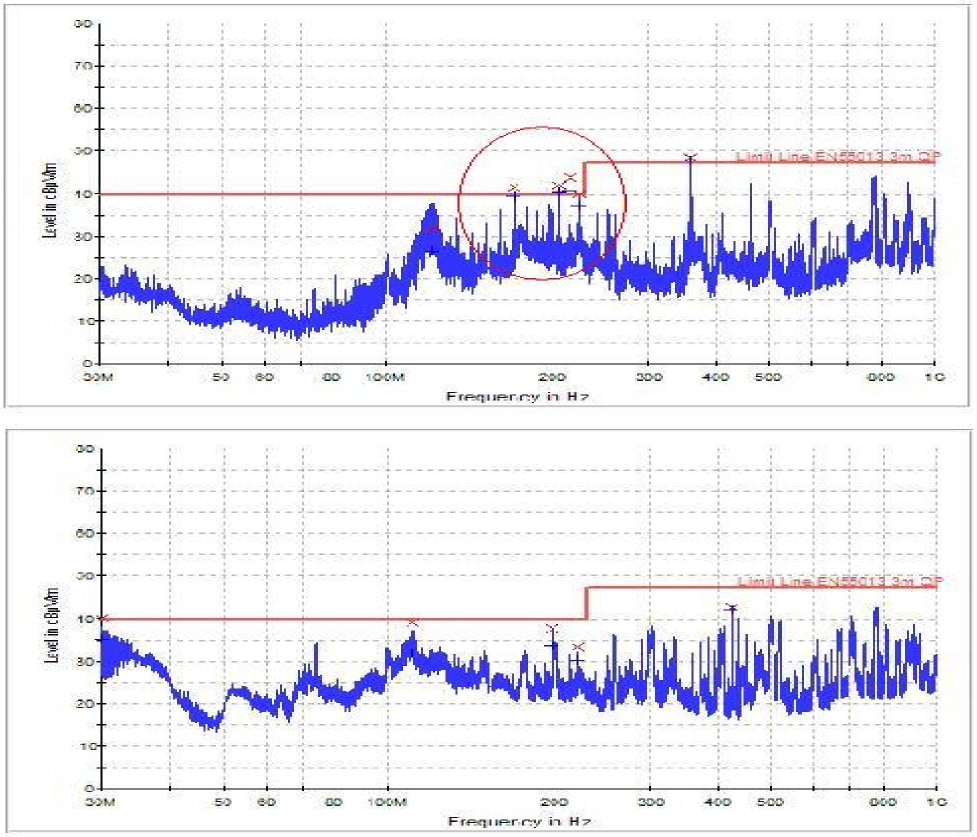


Figure 5.21 High frequency signals should be drawn with a serial component

As it is seen from above test result, when there is no serial component on a high frequency signal, emission level cannor be controlled. But when a serial component is added to circuit, emission level is decreased. But serial component would also reduce the signal level. The level of wanted signal has to be considered.

- High frequency signals that can carry noise, should not be passed through outer sides of PCB.

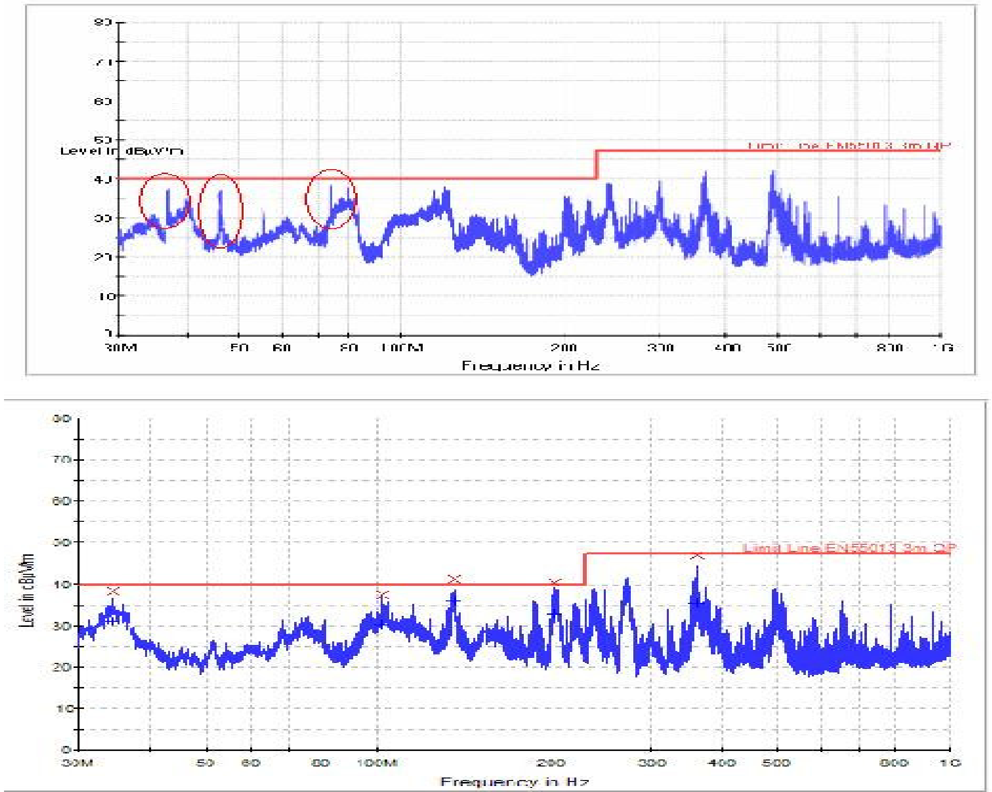


Figure 5.22 High frequency signal cannot be routed from outer side of PCB

For instance, a high frequency block had been drawn from the oouter side of PCB, such as nearby connectors, fundamental frequency and its harmonics have a greater emission level. But when the signal is routed from inner part of PCB with a complete ground consideration, the level has been totally eliminated. High frequency signals nearby connectors and outer side of PCB has much more emission levels.

- The turnings for high frequency signals like DDR,LVDS,HDMI, there should not be sharp cornered. Sharp corners provides high impedance for the circuits and increase EMI

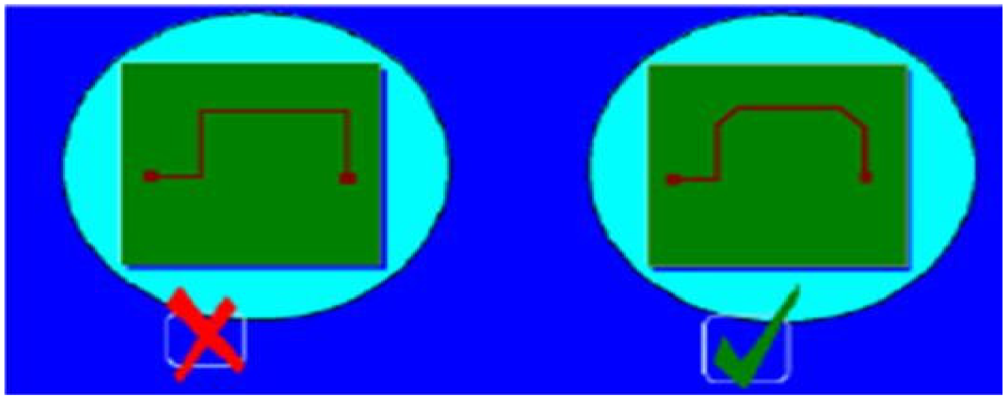


Figure 5.23 Sharp corners increase EMI

- There should not be any long auxilary signal paths for high frequency signals like DDR,LVDS,HDMI.

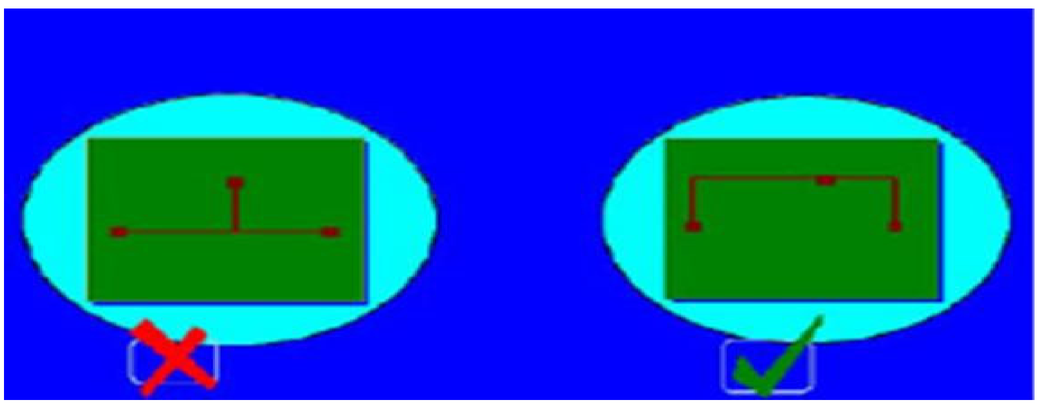


Figure 5.24 Auxilary signal paths increase EMI

- There must be no vias on high frequency paths. If it is mandatory to change layer with via more than one vias must be used.

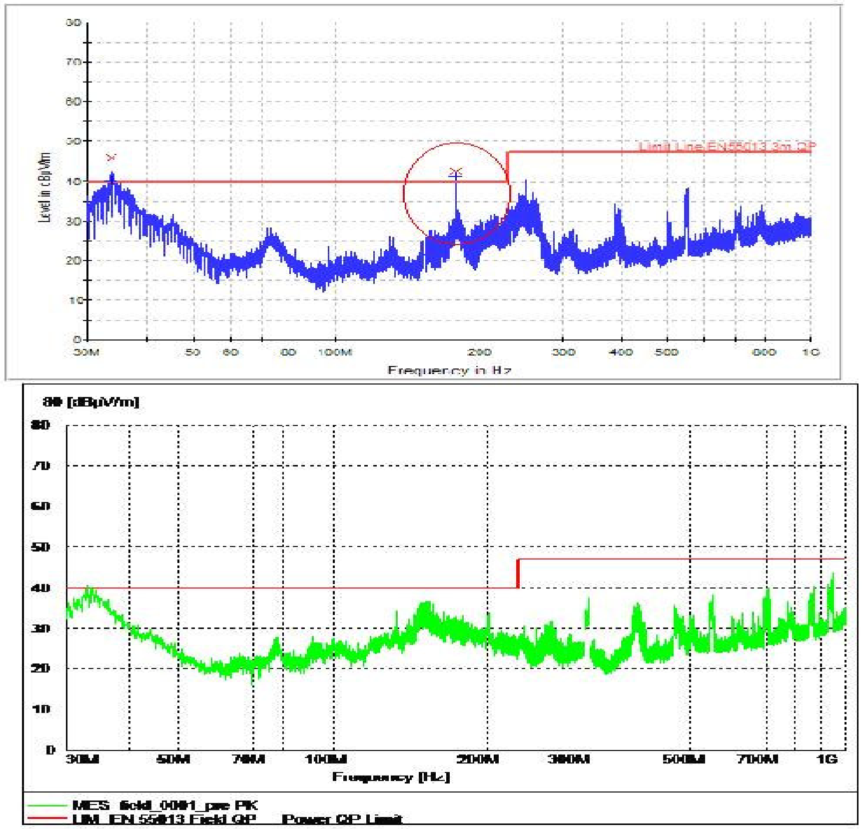


Figure 5.25 A via on a high frequency clock trace increases emission shown in circle

In the test results above, a via used on high frequency clock trace causes an increase in EMI radiation. A clock working in the frequency which is shown in the red circle is reduced by eliminating via on the trace and connect directly.

• LVDS paths must be drawn as pairs and no vias must be used. Paths must be side to side with each other. But there should be a distance of minimum two path thickness between pairs.

• The distance between pairs must be constant all over the path. For example clock,lvds,tmds,..

• None of the paths on chassis must be close to DDR paths and there must be a secure distance.

• Signal paths on different layers should not be parallel. It is easy for coupling noise from one to another on these paths.

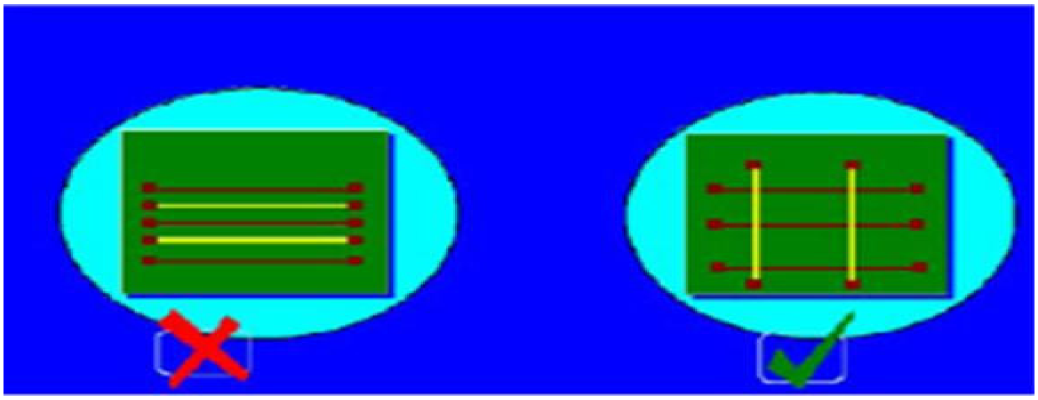


Figure 5.26 Parallel singal paths on different layers increase possibility of coupling

• All clock traces on the PCB should be drawn without via. This is mandatory.

• Bottom part of main IC should be adjusted as a homogen ground plane. This is necessary for both signal integrity and EMI. Because all signal processings are performed in main IC. Lots of components are working with high frequency. Therefore it is necessary to provide low impedance GND plane at the bottom part of main IC.

5.1 Internal Wiring

• All cables should be seperated according to their functional groups. This will reduce the possibility of EMI coupling between different functional cables.

• All cables should be placed closely to metal frame.

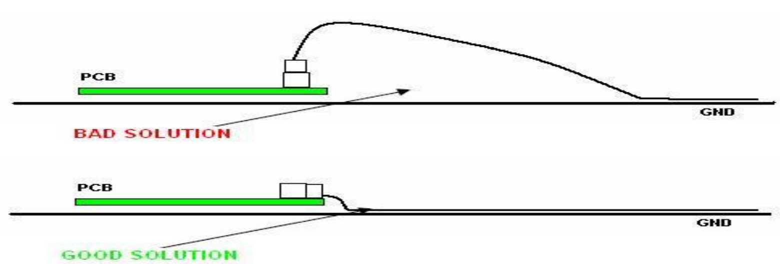


Figure 5.27 Cables should be placed close to metal frame

Metal frame provides a homogen and low impedance ground plane. When there is an EMI radiated by a cable, it can be eliminated by placing it close to metal frame.

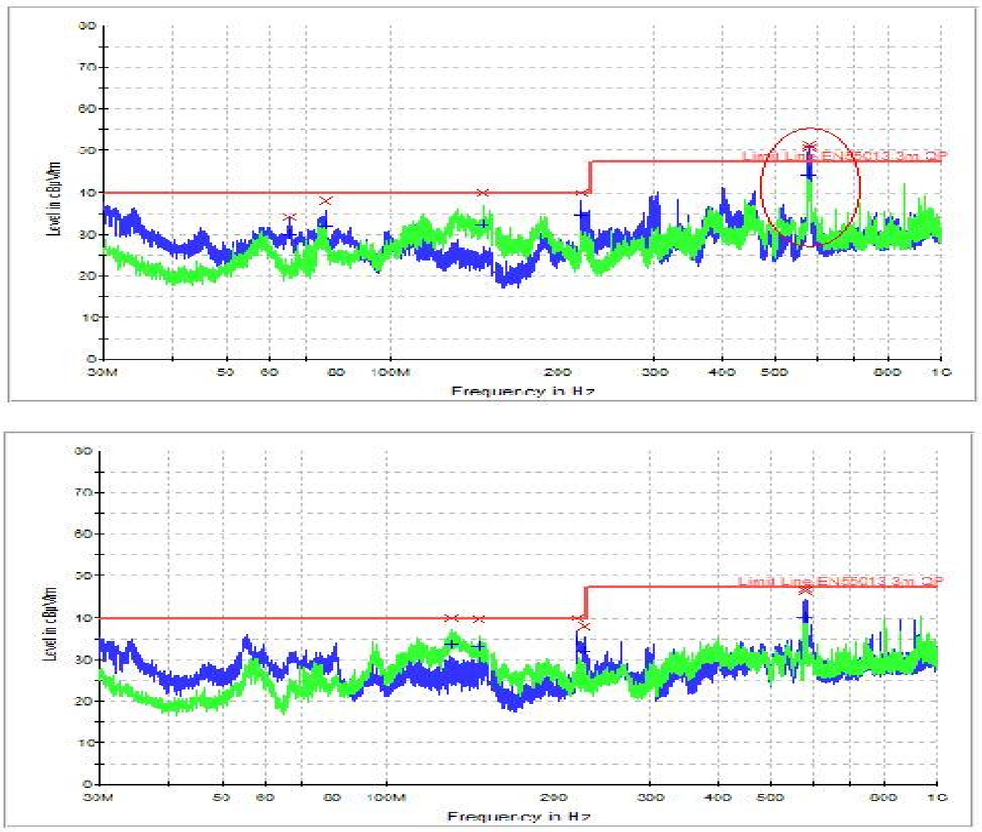


Figure 5.28 Cables should be placed close to metal frame

When a cable is not closed to homogen ground plane, characteristic impedance of the medium increases which results a higher emission level. As it is seen from the above result in red circle, an EMI transmitted by a cable should be reduced by placing the cable close to metal frame or any homogen ground plane.

- All cables shouldn't be unnecessarily long. If the length of the cable is long, the possibility of radiating EMI increases. While generated E field is directly prportional with length of the cable.

$$E_\theta = \frac{-iI_0 \sin\theta}{4\varepsilon_0 cr} \frac{L}{\lambda} e^{i(wt-kr)}$$

- The distance between cables should be as far as possible, the radius of the cable high and characeristic impedance of medium low. These are the factors that effect capacitive coupling.

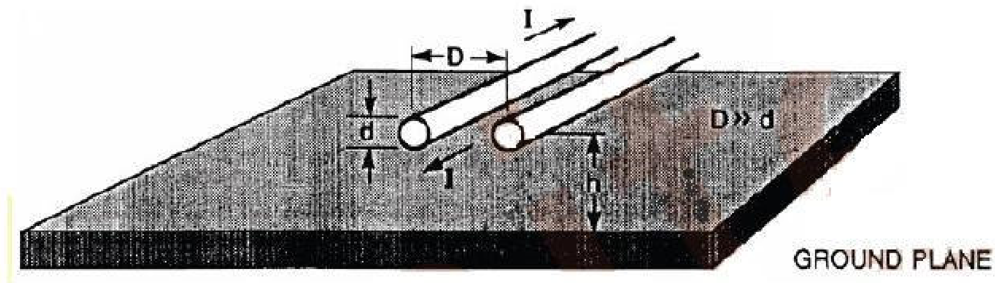


Figure 5.29 Cable parameters that effect coupling

$$C_{12} = \frac{28\varepsilon_r}{\cosh^{-1}(\frac{D}{d})}[pF/m]$$

The angle between cables should be adjusted close to right angle in order to avoid magnetic coupling. Because max coupling occurs when the cables are in paralel because of the direction of radiated field.

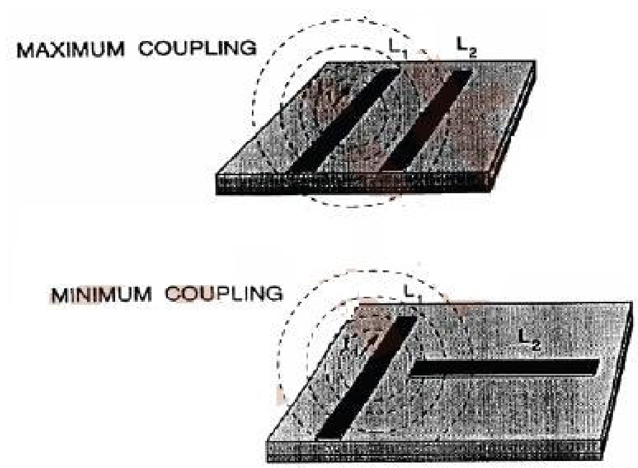


Figure 5.30 Maximum coupling occurs in parallel

- The configuration of usage of flat cable should be determined in the manner of which pins should be GND as described in Figure 5.31.

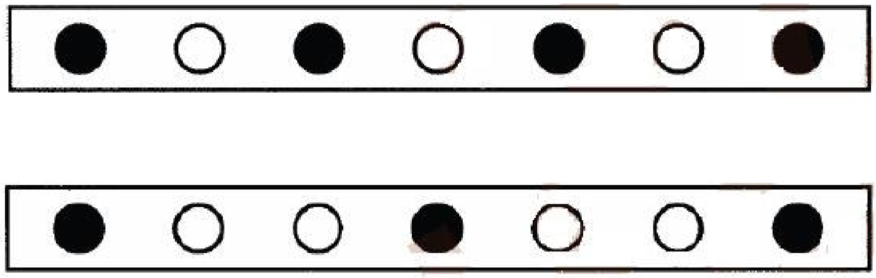


Figure 5.31 GND configuration of flat cables

If it is necessary to use flat cable to carry high frequency signals, above configuration is the best solution while black pins should be ground connections. But when mechanical restrictions and cost up are considered, below configuration can be used even it has more EMI coupling possibility.

- Shielding of LVDS cable should be connected to PCB without data lines, with a ground connection of only itsself. Because if it is carried with data lines, ground that provides shielding over the cable loses low impedance property and grounding effect decreases. This is illustrated in Figure 5.32

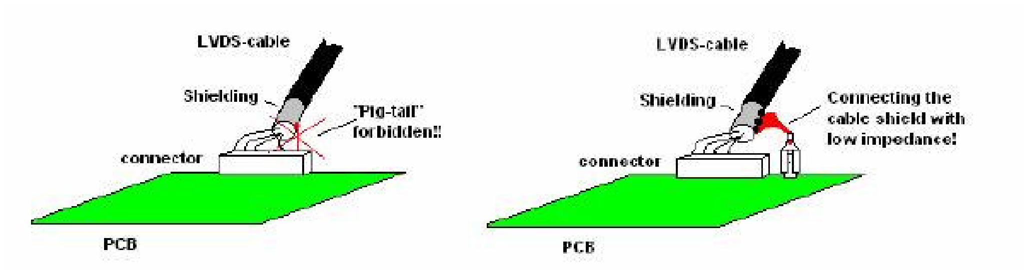


Figure 5.32 GND connection of LVDS cable

- There should be a connection between metal frame and PCB nearby input and output connectors where cables are going through out or in. That will provide a low impedance grounding nearby connectors. This will reduce EMI effect that is radiated by cables.

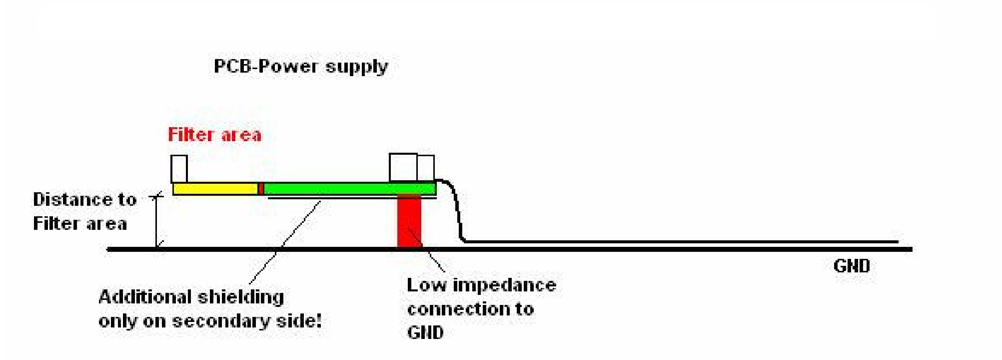


Figure 5.33 Low impedance connection nearby connectors

- Connection of metal frame with LCD panel should have a surface contact not only a point contact which is shown in Figure 5.34

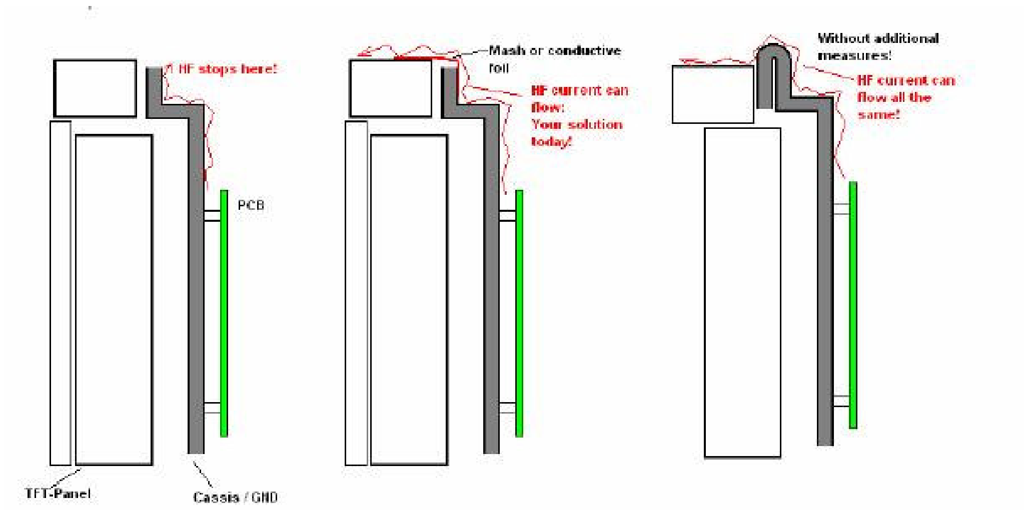


Figure 5.34 Surface contact of metal frame with LCD panel

The theoratical explanation of surface contact is if there is a point contact, high frequency signal that tries to flow to lowest impedance ground stops at the contact point and point contact shown in the figure above behaves like an antenna where sharp corners generally becomes an antenna in electromagnetic theory.

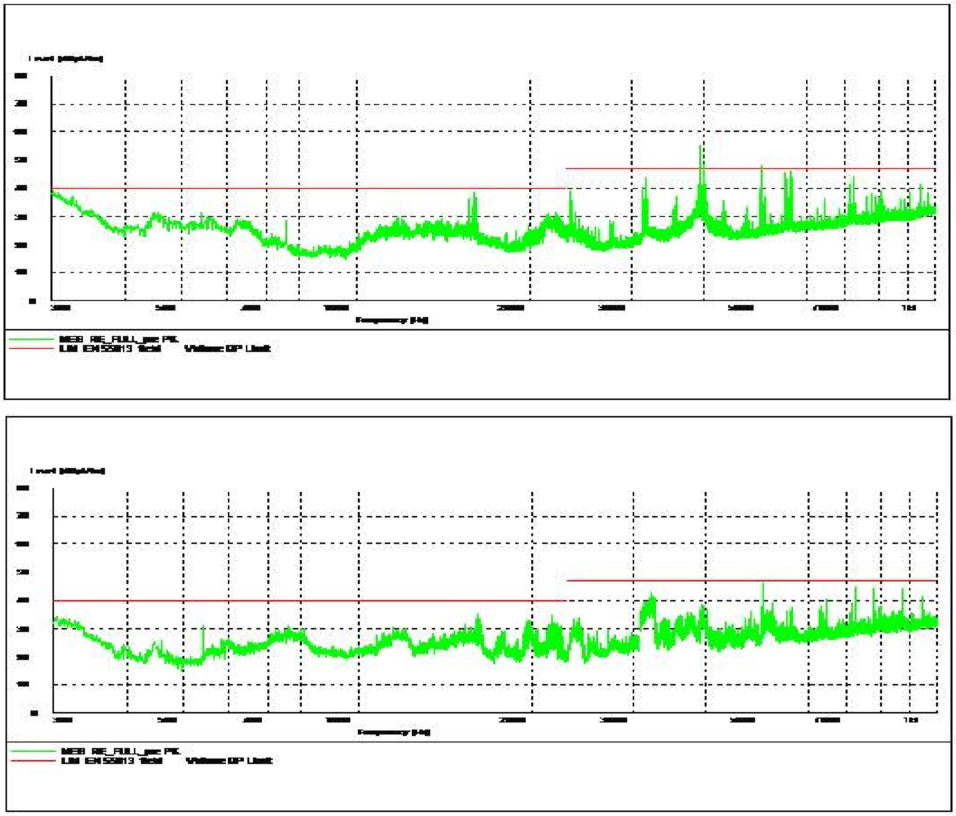


Figure 5.35 Surface contact of metal frame with LCD panel

It can be easily realized that when the contac surface of metal frame is increased, emission level decreases significantly. Therefore mechanical rules are also som important for emc design procedure.

- There is an useful way of transmitting power supply voltages to PCB instead of cables. Board to board connectors can be used to create a low impedance transmission. This would reduce emission level while a cable would react as an antenna.

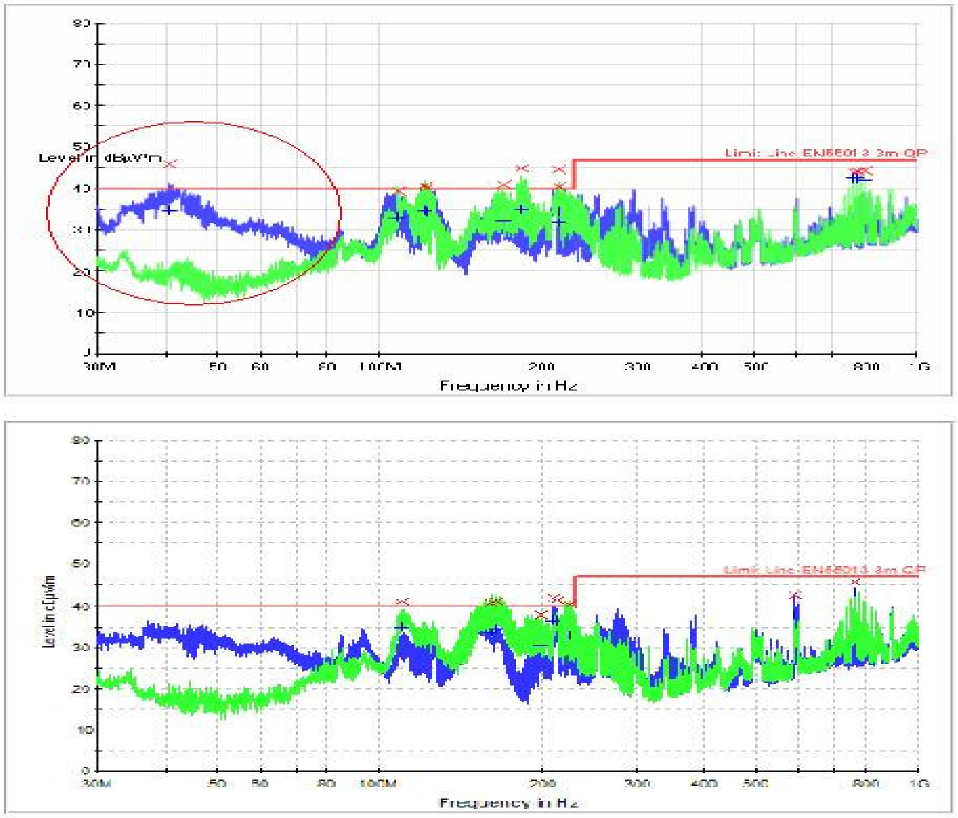


Figure 5.36 Board to board connectors provide a low impedance transmission

Low frequency of the spectrum which is close 30 MHz is mostly related with power supply emissions. It can be realized when a cable is replaced with a board to board connector and tested. Above test result is obtained with a proper cable and below with a board to board connector.

These are the detailed rules for a design with reduced EMI for flat TV applications. Some of them are theoratically explained and some of them are practically proved with test results. There are different types of restrictions in a television design but most of these rules especially proved with test results have to be implemented to get a compliance and stable results. These rules can be grouped according to their importance level. But even it is grouped and most important ones are implemented, it should be known that a risk of uncompliance is taken.

At the next chapter, conclusion and further works are given.

CHAPTER SIX
CONCLUSION

The aim of this thesis is to prepare a guideline for a television PCB and internal wiring design with reduced EMI. The thesis starts with a quick introduction of the necessity of concerning EMI during the design stage. The history of EMI consideration and harmful effects of EMI to other devices or systems had been mentioned. Depending on harmfull effects of EMI to other systems, product testing regulations have been accomplished and the nature of product testing in frquency domain had been observed.

In the thesis, some of the design rules are proved experimentally and some of them proved theoratically. Therefore a basic review for electromagnetic field theory had been presented in the following chapter. The points of electromagnetic theory that is really connected with EMI and EMI based design have been mentioned. For instance, electromagnetic field generation due to common mode currents had been investigated because in most cases EMI is generated due to undefined or unbalanced signal transmission. If the current which is flowed from source to target circuit has to be transferred back totally not to damage balanced transmission. If there is an imbalance in transmission, common mode current and EMI occurs. This kind of theoratical knowledge about electromagnetic field theory has to be considered before the construction of design procedure, some of them have to be verified with theoratical knowledge.

In the third chapter, testing of televisions according to European regulations had been examined. This is a useful idea since it is necessary to know from which test it is tried to pass. These design rules guides for both signal integrity and electromagnetic compatibility. Testing procedure and necessity of compliance have been mentioned to get an idea of how important job is done to take a role in European market. The compliance is mandatory to sell product to Europe. As it can be easliy understood from third chapter that this work is not only related with technical issues but also commercial.

A television design is a very complex job but some critical points of EMI view have been given in chapter four. It is a good idea of mentioning about television design, because critical circuits, signals and functional blocks have to be clearly defined for the design of EMI. There are lots of control signals which consists of very low frequency and no matter for EMI problem. Those kind of signals called I2C busses are not cosidered but for instance LVDS signal pairs which have an approximately 75MHz fundamental frequency and harmonics as a potential problem for EMI. As an analog signal, VGA is only considered because according to standards defined by European Union, it is necessary to test television in VGA mode. Television can be used as a monitor by PC input port and it should be tested whether it effects other devices with harmful EMI or not in monitor mode.

The combination of knowledge about EMC testing and television design concepts generates a design procedure for a flat television with reduced EMI. A design procedure consists of three main groups which are ; PCB orientation, signal routing techniques and internal wiring. Some mechanical considerations are also included because of the importance for reduced EMI.

Design procedure is defined step by step. It starts with PCB orientation which is mainly related with which functional blocks should be placed at which place on a PCB. There are some restrictions of overlapping and interconnecting of different circuits in order to prevent crosstalk and EMI radiation. Sometimes it is necessary to distinct a circuit from others with a ground plane or etc.. PCB orientation deals with the placement of circuits on the PCB and restrictions of applications that shouldn’t be done for reduced EMI. These are mostly derived theoratically because it is hard to change the place of a functional block after all PCB routed. PCB signal routing is the most critical part to prevent EMI at the design stage. There are specific rules for each functional block in television. It is easier to implement and verify experimentally. Some of the design rules belong to this stage of design procedure are implemented and real life results are given. And as a last part of design procedure, internal wiring and mechanical structures are considered. This should be done after PCB is constructed. The cables and mechanical structures can behave like an antenna. This should be prevented with design techniques defined in this part. There are lots of troubleshooting ways of preventing EMI by inernal wiring and mechanical rules. Therefore this part should be clearly understood to find solutions with lowest cost. Otherwise EMI shielding methods just like gaskets, foams, ferrites on cables is neccessary to prevent which means extra cost to the design.

A design procedure for flat television applications is generated in this thesis. This becomes a useful guide for the design for reduced EMI with minimal cost. This is prepared for a specific application of electronics but analogy can be done for other electronic products. The main property and speciality of this thesis is the clear procedure for a specific product which takes a big role in consumer electronics. A television design has lots of considerations such as electrical, safety, performance and etc.. However this guide should be both used in design stage and also in troubleshooting. If there is an EMI problem occured after a prodcut development, specific troubleshooting methods with minimal cost can be found if the problem is localized.

By the help of these desing rules, a checklist can be generated to control design of PCB and internal wiring. A checklist consisting of these rules can be prepared and if the rule is applied, it can be ticked and if it is not applied it can be marked as ‘not applied’. This job should generate a control mechanism for a design to check whether a mistake is done or not. Also if a rule is not applied for other reasons, the risk for EMC compliance and signal integrity can be determined easily. If a rule is not applied and an EMI problem is occured after product development because of that rule, a lessons learned list can be also generated as an experience and not to do same mistakes in further designs.

In addition according to how much of design rules are applied, a maturity level of PCB for EMI design can be generated. That means according to how much percent of rules are applied, it can be compared with the test results. This can be done with experience. For instance, if a percentage of rules are applied and that television cannot give compliance, that percentage should be called as a danger stage. But if it can give compliance with that percentage, it can be called as a secure stage.

These kind of control mechanisms can be generated as further works. But the most important thing to get compliance with minimal cost for a good engineering work.

REFERENCES

Aleaf, A. (11//2000). *EMI Results and PCB Design Considerations*. Retrieved May 2007, from www.parallax.com/dl/appnt/sx/An3Emi.pdf

Armstrong, K. (2000). *Measuring Radiated Emissions.* London: REO UK Ltd.

Armstrong, K. & Williams, T. EMC Testing Radiated Emissions. *EMC Compliance Journal, Part 1,* 1-32. Retrieved October 19, 2007. from associate of EMC UK database

Burr, J. (3/30/2007). *EMC and Circuit Design*. Retrieved January 2007, from http://www.egr.msu.edu/classes/ece480/goodman/spring/group06/files/emi_justin.pdf

Cho, S. (1994). *EMI Preventation in Clock Distribution Circuits*. Retrieved January 2007, from focus.ti.com/lit/an/scaa031/scaa031.pdf

Mercer, S. (01//1999). *Minimizing RF PCB Electromagnetic Emissions*. Retrieved May 2007, from http://images.rfdesign.com/files/4/0199MERCER46.pdf

Micrel, INC. (2004). *General PCB Design and Layout Guidelines*. Retrieved January 2007, http://www.micrel.com/page.do?page=/product-info/app_notes.shtml

Montrose, M. & Nakauchi, E. (2004). *Testing for EMC Compliance* (1st ed.). New Jersey: John Wiley & Sons, Inc.

National Semiconductor. (1998). *Overview of LVDS Technology* . Retrieved May 2008, from www.national.com/an/AN/AN-971.pdf

National Semiconductor. (2004). *LVDS Owner's Manual* (3rd edition). Retrieved May 2008, from www.national.com/appinfo/LOM3.pdf

Schaffner (2000). *RF Emission Testing.* Switzerland: Schaffner EMV AG.