Electromagnetic Compatibility of Circuit System Design

WEIHUA WANG

ABSTRACT

As automobiles, communication equipment, computers, and electrical equipment enter the home, the electronic devices are leading to a dramatic increase in space electromagnetic energy, and the electromagnetic environment is deteriorating. Therefore, the world attaches great importance to electromagnetic environment which is becoming more and more complex. And the broad influence of this subject is prompting environmental magnetism and electromagnetic compatibility technology to develop rapidly. The article mainly introduces the concept of EMC and emphatically analyzes the shielding design, signal and power integrity of PCB board design. Meanwhile, this article summarizes EMC test technology.

KEYWORDS

EMC, PCB board, signal integrity, power integrity, EMC test.

INTRODUCTION

With the development of electronic technology, the systems of electronic equipment become more and more complex. The electronic equipment will inevitably work in environment with high electrical level surrounded. However, electronic equipment’s inner system may easily cause some electromagnetic interference problems. If the adaptability of electronic equipment in electromagnetic environment cannot be improved and electromagnetic compatibility of electronic equipment cannot be solved properly, the performance of electronic equipment will not be brought into play. How to improve the compatibility design of electronic equipment has become one of the top issues for electronic equipment designer. As shown in the fig. 1, based on the understanding of electromagnetic interference, this article will clarify the basic concept of electromagnetic compatibility. And according to the system design of shielding design, printed circuit board (PCB) design of signal integrity and power integrity design two aspects, this article will analyze the key points of electromagnetic compatibility (EMC) design. Meanwhile, the EMC test techniques are analyzed. Then the test instruments, test sites and test methods are summarized.

Weihua Wang, School of Nanjing University of Posts and Telecommunications, Nanjing 210023, China; 1664496281@qq.com
THE CONCEPT OF ELECTROMAGNETIC COMPATIBILITY

Electromagnetic compatibility technology is developed during the course of studying electromagnetic interference mechanism and electromagnetic interference protection technology. The electromagnetic interference (EMI), an electromagnetic phenomenon, can be one the factors which will affect the electromagnetic compatibility, including the performance of equipment, devices and etc.. It may also damage the inanimate substances. EMI has been discovered for a long time and it was found almost simultaneously with electromagnetic effects. In 1881, British scientists published an article on radio interference, which marked the beginning of the study of interference.

Although the problem of electromagnetic interference is long-standing, the new comprehensive discipline of electromagnetic compatibility is not formed until recent decades. International Electro-technical Commission Standard (IEC) defines the electromagnetic compatibility as a system or device that works in an electromagnetic environment and does not interfere with other systems or equipment. EMC is a subject, studying in a limited space, limited time and limited spectrum resources. And the various electrical equipment (subsystem, system, generalized including biological science) can coexist and do not cause degradation. [1]

ELECTROMAGNETIC COMPATIBILITY SYSTEM DESIGN

The Principles of Electromagnetic Shielding.

Electromagnetic shielding is an application of shield’s absorption or reflection of an interfering electromagnetic waves to reduce interference energy, cutting off the coupling path of electromagnetic waves. It utilized a low resistance conductor material and the property that electromagnetic waves reflect on the surface of the shielded
conductor, absorb and repeatedly reflect inside the conductor to play a shielding role. The purpose is to effectively prevent electromagnetic waves from one space to another.

**Several Factors Affecting Shielding Effectiveness and Their Solutions.**

**Material**

When the electromagnetic wave passes through the metal plate, the metal plate generates an induced eddy current to form an ohmic loss and converts to thermal energy loss. At the same time, the eddy current counter magnetic field cancels the incident wave interference field and produces absorption loss.

When selecting shielding materials, the following principles should be followed:

1) When the frequency of interference electromagnetic field is high, the eddy current produced by the low resistivity metal material forms the bucking effect to extraneous electromagnetic waves, thus achieving the shielding effect;

2) When the frequency of interference electromagnetic field is low, using high permeability materials, the magnetic lines of force will be limited inside the shield and prevented from diffusing into shielding space;

3) In some cases, if the high frequency and low frequency electromagnetic fields are required to have good shielding effect, different metal materials are often used to form multilayer shielding.

**Gap and Void**

The shielding effectiveness of the shield depends not only on the material that constitutes the shield, but also on the structure of the shield. The discussion above is the full continuous shield. But in the reality, a completely closed shield is of no value. The crate or shell usually has plenty of display windows, air vents, gaps in different parts and etc. As the result of these factors that lead to electrical discontinuities, shielding effectiveness of the shield is often very low, or even have no shielding effectiveness. Therefore, it is necessary to study the gaps and voids in shields.

In practical design, the following measures are often used to reduce the electromagnetic leakage of voids:

1) For crates with gaps and voids, the gaps and voids should be positioned as far away from the source of the excitation as possible;

2) Increase the overlap size at the seams;

3) Shorten the space between the screws. When there is no conductive elastic gasket on the joint surface, the number of screws should be increased and the space between the screws should be reduced, so that the length of the gap will decrease accordingly and the shielding effectiveness will be improved;

4) Hole opening design. When the opening area is the same, it should minimize the opening of large holes and long and thin holes. For the crate which can be centralized and separated, the holes shall be respectively opened to reduce the leakage;

5) Add metal conduit. Generally speaking, for large holes, metal ventilation ducts are often used in actual design. When the frequency is higher than 100MHz, the waveguide can be used as the ventilation duct. Because the metal tube is equivalent to a high pass filter, the electromagnetic fields which below the cut-off efficiency of
metal tubes will cause a great attenuation when passing through pipes, so the leakage caused by the hole is reduced.

**ELECTROMAGNETIC COMPATIBILITY PCB BOARD DESIGN**

Printed circuit board (PCB) has been widely used because of its simplicity, good performance and low price. With the development of micro technology and the increasing density of PCB board, the problem of electromagnetic compatibility becomes more and more prominent. At the same time, the challenges caused by the complexity of its design are also increasing.

**Signal Integrity Design of PCB Board.**

Signal integrity is the ability of a signal to respond to the correct time sequence and voltage in a circuit. It is a state in which the signal is not damaged. It indicates the quality of the signal on the signal line. It mainly includes delay, reflection, crosstalk and etc. [3, 4]

**Delay**

Delay means that the signal is transmitted at a limited speed on the wire of the PCB board, and the signal is sent from the sending end to the receiving end, where there is a transmission delay. When the system clock is very high, the transmission time of the signal between the devices and the synchronous preparation time are shortened, and the driving overload and the route too long can lead to delay. High speed circuits are required to meet various gate delays in a very short time, including setup time, holding time, wire delay and etc.

**Reflection**

Reflection is the echo of the signal on the transmission line. When a signal is transmitted on a high-speed PCB board, reflections occur at the discontinuity of impedance. According to the signal transmission theory, when the signal is transmitted to the load through the transmission line, some part of the energy will return to the source-end because of the impedance mismatch. In high-speed PCB, the line mutation, via holes, angles and the source-end / load end which do not match with the impedance of the transmission line, will cause reflection.

**Crosstalk**

An alternating magnetic field is produced when alternating current is passed through the signal circuit of the high-speed circuit. Adjacent signal lines in this field will sense the signal voltage, sometimes causing high frequency resonances, and then coupling to adjacent interconnects, resulting in crosstalk. Crosstalk may cause spurious clocks, intermittent data errors and etc., which affect the quality of the adjacent signals. In fact, we do not need to completely eliminate the crosstalk. As long as we control it within the limits of the system, we can achieve the goal. Crosstalk can be reduced by adding ground wire, increasing distance between lines and reducing parallel distance.
Stimulation Analysis of Signal Integrity

In this paper, the transmission line reflection is taken as an example, and the waveform of the load end is simulated by using the Signal Integrity (SI) tool Hyperlynx. [2]

A 5V level signal is transmitted from the driving end to the receiving end through the microstrip line. And the signal waveform of the driving end and the receiving end is observed on the oscilloscope.

Through the analysis and simulation of signal reflection, the following conclusions can be obtained:
1) The energy on the transmission line begins at the source end and ends at the load end, and the ideal transmission line does not consume energy;
2) Whether the load end will overshoot or the degree of overshoot is related to the degree of impedance mismatch.

Power Integrity Design of PCB Board.

In the study of signal integrity analysis, how to provide a stable and reliable power supply has become one of the key research directions. The power integrity problem means that in high-speed PCB, when a large number of chips are turned on or off at the same time, a large transient current is generated in the circuit. Due to the presence of inductance and resistance on the power line and ground wire, voltage fluctuations occur on both of them. [5, 6]

There are two main ways to solve the power integrity problem: installing the decoupling capacitor and optimizing the layered design of the power supply.

Application of Decoupling Capacitor

First of all, for high-speed devices, we remove the high-frequency noise components by adding decoupling capacitors, thus reducing the transient time of the signal. The decoupling capacitor is also equivalent to the local power supply and provides current for the state switching of the switching device. The chip does not need to obtain current from the far power supply through the power supply line.

When the power supplies a short time to the load, the storage charge in the capacitor can prevent the voltage drop. If the placement of the capacitor is not proper, the line impedance will be too large to influence the power supply. And the capacitor can filter the high frequency noise at the high speed switching of the device. In the high-speed PCB design, we usually add a decoupling capacitor at the output of the power supply and the chip power input, in which the capacitance near the power source is generally large (such as 10 uF). For the decoupling capacitor that is connected to the chip power pin location, the capacitor value is generally small (such as 0.1 uF). Moreover, when the decoupling capacitor is arranged, the length of line between the pad and the power is minimized, and we should use a wide line.

The Design of Power Circuit

For the inductance in the loop, we should consider the layer design of the power supply.
First of all, for the design of the power line and ground wire, we need to ensure that the line width is bold (such as 40mil wide while the ordinary signal line is 10mil), so as to minimize the impedance value. With the speed of chip is increasing, according to the 5/5 rule, we use more multilayer board. Through the special power supply layer supplying electricity and special ground layer forming a loop, the circuit inductance will be reduced.

For high integrated PCB design, loop area may be relatively large, because of the complexity of the signal line. In order to solve this problem, we add a capacitance between power and ground planes, where is near the signal line. Thus, for high frequency signals, a signal line at the top will produce a mirror circuit on the ground layer, and the signal line of the ground layer will produce a mirror circuit on the power supply layer. These two mirror circuits will form a loop with the capacitance between the power supply layer and the ground layer. In this way, we can utilize the power supply layer and the ground layer as a loop to reduce the area of loop as much as possible, thus reducing the possibility of power integrity and interring board electromagnetic compatibility problems.

Simulation Analysis of Power Supply Integrity

The power supply Integrity simulation software SpectraQuest PI is used to simulate the circuit. This article mainly describes the single node simulation and multi node simulation of decoupling capacitor.

The single node simulation can determine the capacitance and quantity of the decoupling capacitor. However, the impedance characteristics of each region of the PDS depend not only on the capacitance and quantity of the capacitors, but also on the distribution of capacitors. In the single node simulation, the selection of decoupling capacitor can be adjusted according to the simulation wave form.

The multi-node simulation will determine the impedance near each grid node based on the user's separated grid size of power supply layer, Voltage Regulator Module (VRM) noise source location, parameter settings, and the distribution of decoupling capacitors. In multi-node simulation, the power supply layer is divided into grids. The power plane within the grid shall belong to the lumped system, the effective range of the decoupling capacitor is the 1/6 effective length of the PCB signal. In order to make the effective range of the decoupling capacitor cover the whole power supply layer, the spacing of the decoupling capacitors should be less than L/12 effective radius. To make the simulation more accurate, the size of the grid should be less than the effective radius of the decoupling capacitor.

ELECTROMAGNETIC COMPATIBILITY TEST

Electromagnetic compatibility (EMC) tests include test instruments, test sites, and test methods. All EMC tests must be carried out using the appropriate test instruments and in the specified test sites.

Test Instrument.

In the field of testing instruments, the automatic detection system based on spectrum analyzer can provide the relevant parameters of EMC quickly and accurately. A new type of EMC scanner combined with spectrum analyzer can realize the visualization of electromagnetic radiation. It can carry on the omni-directional
three-dimensional test to individual component of the system, the PCB board, the complete machine, the cable and etc. Thus, it can display the real electromagnetic radiation condition.

Using interference receivers with quasi peak and average detectors, the performance should comply with the requirements of CISPR16-1 or corresponding GB GB/T 6113.1 (< radio interference and immunity measurement equipment specification >). In the standard frequency range, the 2 general frequencies of interference receiver are 10 kHz ~ 30MHz and 30 ~ 1000MHz.

Test Site.

Generally, there are two types of labs for EMC testing: one is a comprehensive design and test laboratory or testing center with the approval of the EMC authority, the quality system certification as well as the statutory testing qualification. The other type is the EMC laboratory which has a certain testing function based on the actual needs and funding conditions of the unit. At present, the commonly used test sites at home and abroad are: open field, shielding room, anechoic chamber, reverberation room, transverse electromagnetic wave room and etc.

In EMC test, the influence of the site on the test result is obvious. The same instrument will get different test results in different test sites. This is mainly due to the difference of reflected waves in different test sites, resulting in inconsistent superimposed field strength. This paper mainly introduces the open field and shielding chamber in two experimental fields.

When measuring radiation interference with an open experimental field, the basic structure is the open and non-reflecting objects around the ground. The ground is flat and uniformly distributed on the metal ground surface. A radiation test site that meets the standard should be a field surrounded by an ellipse with a long axis equal to two times of the focal length (F) and a short axis equal to \( \sqrt{3} \) times of the focal length. When testing, the Equipment under Test (EUT) and the measuring antenna will be placed at the two focuses respectively. The open experimental field is mainly used in the range of 10~30 MHz frequency in the field of electromagnetic compatibility. It can be used for electromagnetic radiation emission test of EUT, and can be applied to large-scale EUT test.

The shield room is a large hexahedron, made of metal grids or metal boards. Because of the shielding effect of the metal material, testing in the shielding room will not be affected by the external electromagnetic environment (passive shielding). Moreover, we can test the sensitivity of the electromagnetic radiation field strength in the shielding room without fearing of causing electromagnetic interference to the outside world (active shield). [7]

Test Method.

EMC test methods are divided into direct test method, alternative test method and automatic measurement method. This section displays two examples of the direct test method and the alternative test method.
Direct Test Method

The direct test method refers to the EMC test, carried out in accordance with the laboratory configuration and test conditions required by a EMC standard. What it pursues is the accuracy of the test results and the comparability with the testing institutions at home and abroad, while the cost is higher.

Alternative Test Method

As the aforementioned laboratory configuration is expensive, if the test site is involved, it is not acceptable for the general production enterprises. The alternative test method, in ensuring the certainty of premise comparability, can minimize the cost of configuration that more companies can accept. The advantage of this method is its simpleness and convenience. The other advantage is its requirement of environment is not high. It can be carried out in the shielding room, and the test results have good repeatability and comparability.

CONCLUSION

To achieve the reliable operation of electronic equipment, electromagnetic compatibility technology must be studied deeply. In this paper, the concept, design and test of electromagnetic compatibility are summarized. The article describes the electromagnetic compatibility mainly through the design, the simulation, the experimental verification and the test verification. And the simulation design can reduce the cost of experiment and find the problem of the experiment in time. The reliability of the simulation is verified by tests, thus ensuring the regular work of electronic equipment.

REFERENCES