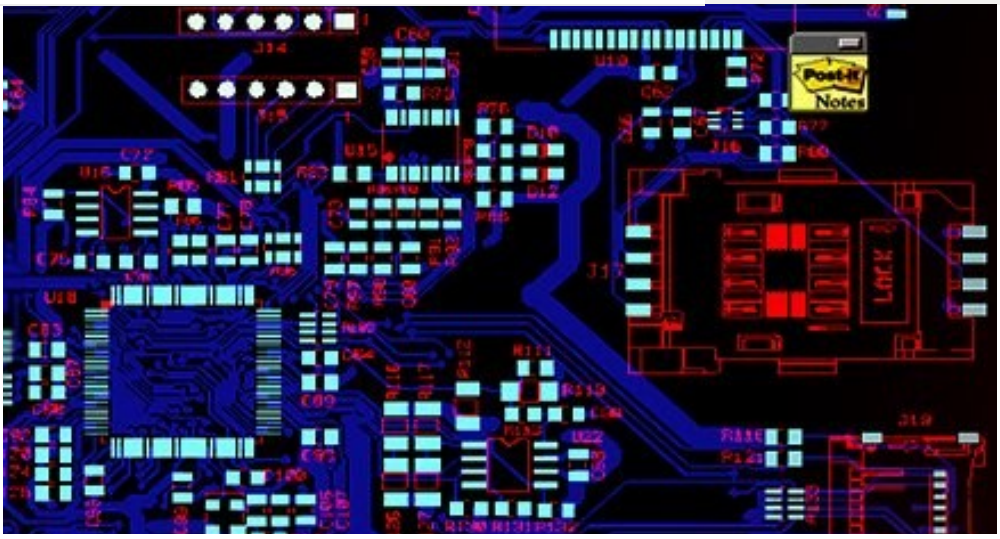


Alan Yang

EMI Control Technology in Diglital PCB Design

Publicerades 20 juli 2016

Yu Ping
PCB Manufacturing, Assembly & Design Services

1. EMI generation and suppression principle

EMI occurs due to electromagnetic interference sources by coupling path to transfer energy to the cause of sensitive systems. It includes a common ground via a wire or conduction, radiation or coupling through space three basic forms through the near field. EMI harm is done to reduce the transmission of the signal quality of the circuit or device causing interference or even destroy, the equipment can not meet the technical requirements of EMC standards required.

To suppress EMI, EMI digital circuit design according to the following principles:

* According to relevant EMC / EMI specifications, indicators of decomposition to the circuit board, hierarchical control.

* That the three elements of the source of interference from EMI, the energy coupling path and sensitive control systems of these three aspects, the circuit has a flat frequency response, ensure that the circuit is normal, stable operation.

* Starting from the front-end design equipment Follow EMC / EMI design, reduce design costs.

2 digital control circuit of the PCB EMI

In dealing with various forms of EMI, we must analyze specific issues. In the PCB design of digital circuits, EMI can be controlled from the following aspects.

EMI during the design, we must first consider the choice of the rate of the
circuit, if the rise time of the device into 5ns rise time of 2.5ns devices, EM

bandwidth, it is the rise time of the signal rather than a function of signal f
0.35 / Tr (where Tr is the rise time of the signal of the device) this type of
frequency range of 30MHz to several GHz, at this frequency, the wavelen
very short, even if the circuit board wiring may also become a transmitting
high EMI, the circuit is easy to lose normal function. Therefore, in the dev
ensure the circuit performance requirements of the premise, should try to u
chip, using the appropriate drive / receive circuit. Further, since the lead pi
has a parasitic inductance and capacitance, and therefore high-speed desig
device package form on the signal can not be ignored, because it is also an
in EMI radiated. Parasitic Generally, the patch device is smaller than the ir
parasitic BGA package is less than the QFP package.

Select the signal terminals Definition 2.2 connector

Connector is the key to high-speed signal transmission, but also easy to pr
link. In the connector terminal is designed to be multi-pin arrangement, an
signal distance, reduce the radiation generated in the active signal connect
provide a low impedance return path. If necessary, to consider some of the
signal isolation.

2.3 stack design

In their costs through licensing, increasing the number of ground plane layer adjacent to the signal ground plane layer may reduce EMI radiation. For high-speed PCB, power and ground planes close coupling, to reduce the impedance of the power supply, which reduces EMI.

2.4 Layout

The signal current flow, reasonable layout, can reduce interference signals. Rational distribution is the key to control of EMI. The basic layout principles are:

* Analog signal susceptible to interference digital signals, analog circuits and digital circuits should be separated;

* Clock line is major interference and radiation, away from sensitive circuitry, and clock traces the shortest;

* High-current, high-power circuits to avoid the center of the plate is arranged in the region, taking into account the impact of heat and radiation;

* Connector plate as far as possible in the side and away from the high-frequency circuit;

* Fully consider the feasibility of the layout of the power split, multi-powe
power across the divided region boundary placement, in order to effectivel
impact on EMI plane division;

* Plane at reflux (route) is not divided.

2.5 Cabling

* Impedance control: high-speed signal line will show the characteristics o
line, the need for impedance control, in order to avoid signal reflection, ov
ringing, reduce EMI radiation.

* Signals are classified according to the different signal (an analog signal,
O signals, bus, power, etc.) of the radiation intensity and the sensitivity of
the interference source separation sensitive systems as much as possible, r
coupling.

* Strict control clock signal (especially high-speed clock signal) trace leng
vias across partitions, termination, wiring layer, return path and so on.

* Signal loop that signals flowing to the loop formed by the inflow signal,
key to control of EMI, in the wiring must be controlled. To understand the
the key signal to the key signal return path near the wiring to ensure that it
minimized.

Low-frequency signals, to make the current flowing through the path of least resistance; high-frequency signal, high-frequency current flows through the path of least inductance to make, not the path of least resistance (see Figure 1). For differential-mode radiation, EMI radiation intensity (E) proportional to the current, the current loop area and the square of the frequency. (Where I is the current, A is the loop area, f is the frequency, r is the distance to the center of the loop, k is a constant.)

Therefore, when the minimum inductance in the return path signal wire just below the area of the loop current can be reduced, thereby reducing EMI radiated energy.

* The key signal can not be split across the region.

* High-speed differential signal traces as a tight coupling.

* Ensure stripline, microstrip line and the reference plane to meet the requirements.

* Lead decoupling capacitors should be short and wide.

* All signal traces should be kept away from the edge of the board.

2.6 division processing power plane

* Power supply layer divided

On a main power plane has one or more sub power supply, to ensure conti
each region and adequate width of the copper foil. Dividing line should no
usually 20 ~ 50mil width can be, in order to reduce the gap radiation.

* Ground plane split

Ground plane layer should maintain integrity and avoid division. If you mu
distinguish between digital, analog and ground noise, and at the outlet via
point connected to an external ground.

In order to reduce the edge of the radiation power, the power / ground plan
principles should be followed, i.e., a power plane ground plane size than th
(see FIG. 2), so that the edge of the radiation field intensity can be decreas

Other means of control of the 3 EMI

3.1 Power System Design

* Design a low impedance power supply system to ensure that the impedar
frequency range below the fknee power distribution system is lower than t
impedance.

* Use filters to control the conduction interference.

* Power supply decoupling.In EMI design to provide reasonable decoupling capacitors, chip enable reliable operation and reduce the power of the high-frequency noise, reducing EMI. Due to the effects of lead and other parasitic inductance of the power supply conductors and slow to respond, so that will make high-speed circuit in less than instantaneous current drive needed. Rational design of bypass capacitors or decoupling capacitors on the supply and distribution layer, before power can respond using capacitive energy storage to provide fast-acting current devices. Proper decoupling capacitor provides a low impedance path to power, which is the key to reduce the common-mode EMI.

3.2 Ground

Grounding the entire board design is the key to reduce the EMI.

* Determine the use of single-point ground, mixed or multi-point grounding grounding.

* Digital, analog noise to be separated and to identify a suitable common ground.

nd grid d

width. A large area may also be paving the way, but be careful on the same area of consistency is better.

* For multilayer design should ensure that there is a ground plane layer, re common ground impedance.

3.3 damping resistor in series

In the circuit timing requirements permitting, the basic sources of interfere technology is the key signal output terminal small value resistor in series, of resistance. These outputs are small series resistor can slow down the ris overshoot and undershoot enable signal becomes smoother, thereby reduci amplitude of the output waveform, to effectively suppress EMI purposes.

3.4 shield

Key components can use EMI shielding material or shielded.

* Shielding of critical signals, can be designed to strip line or both sides of signals to isolate the ground.

3.5 Spread Spectrum

The method of spread spectrum (spread) is a new and effective method to spread spectrum signal is modulated, the spread signal energy onto a relati of frequencies. In fact, the process is controlled by a clock signal modulati does not significantly increase the jitter clock signal. The practical application of spread spectrum technology proved to be effective, radiation can be reduced from 7 to 20dB.

3.6 EMI analysis and testing

* simulation analysis

After completion of PCB layout, you can use EM I simulation software and expert system simulation analysis, simulation, EMC / EMI environment, to assess whether the product meets the relevant EMC standards.

* Scan Test

Using electromagnetic radiation scanners, disk scanning machine and associated loading after power to give the PCB of electromagnetic field distribution (Fig. 3, shown in red, green, green white area represents the energy of electromagnetic radiation from low to high), according to the test results improved PCB design.

4 Summary

ɛed chip,
frequency is also increasing, and the board of the PCB board may be getti
design EMI will face more severe challenges, can only continue to explore
order to make the PCB EMC / EMI design success.

Publicerades av

Yu Ping
PCB Manufacturing, Assembly & Design Services
Publicerades • 4 år

Reaktioner

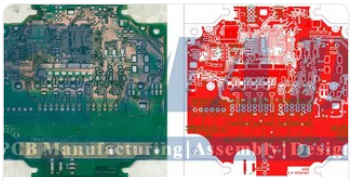
0 kommentarer

Lägg till kommentar ...

Yu Ping
PCB Manufacturing, Assembly & Design Services

+ Följ

Mer från Yu Ping



PCB Reverse Engineering Methods
Yu Ping på LinkedIn

8 Classic Q&As Concerning PCB Layout
Yu Ping på LinkedIn

Six Tips for Choosing PCB components
Yu Ping på LinkedIn

New Patent of 3D Circuit Board Manufacturing
Yu Ping på LinkedIn

Se alla 18 artiklar