



Analysis of Disturbance Factors in Design of Printed Circuit Board (PCB)

In the continuous development of information technology, electronic products, functions, types, structures are increasingly complex, to promote the PCB design gradually multi-level and high density, PCB design of the electromagnetic compatibility issues have also been valued and attention. Electromagnetic compatibility (EMC) design not only to ensure that all the PCB board circuit is normal, stable work, do not interfere with each other, but also effectively reduce the PCB external radiation and transmission, to ensure that the PCB circuit from external radiation and conduction interference. Therefore, it is very important to study the electromagnetic compatibility based on PCB design.

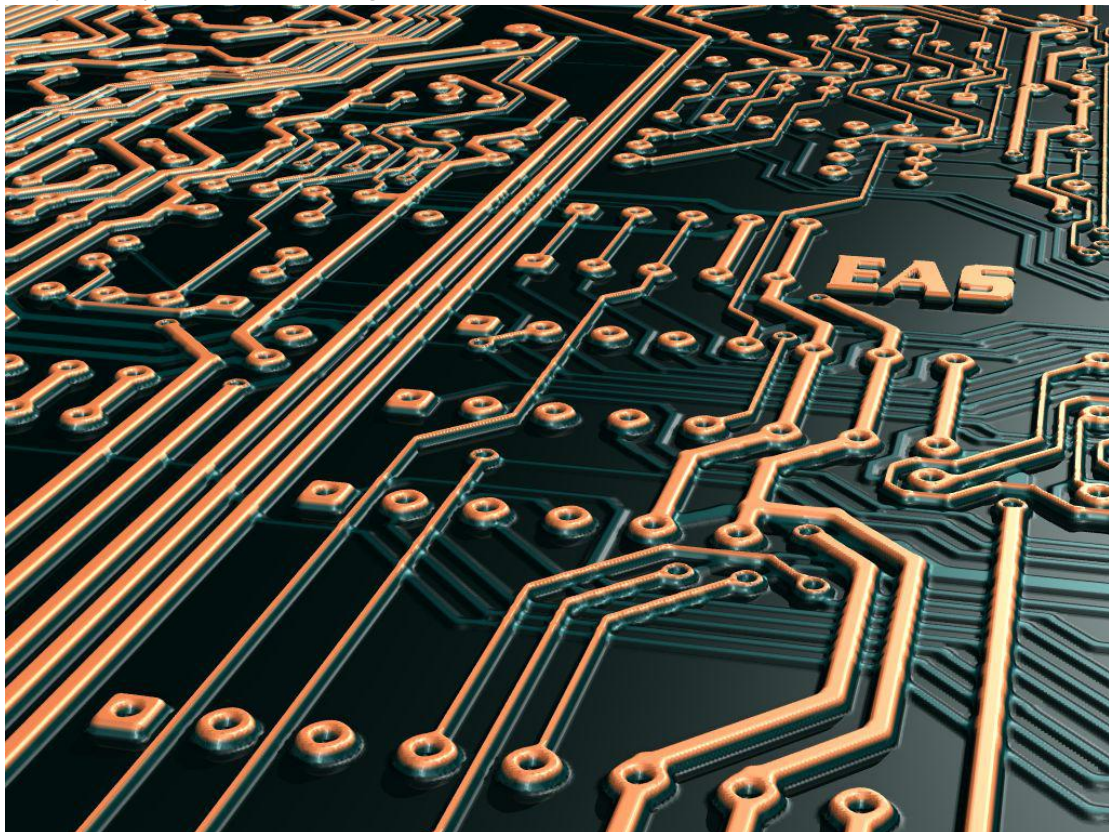


Fig PCB

PCB interference type

PCB interference there are three main, one is the layout of class interference, usually on the PCB board components placed inappropriate interference; the second layer of interference, usually due to unscientific settings caused by noise interference; Third, , Usually the PCB signal line, the power line and the ground line distance or line width settings unreasonable or PCB wiring methods such as improper interference. On the type of PCB interference, it can be taken to the layout rules, hierarchical countermeasures, alignment rules to suppress measures to weaken or even eliminate the impact of PCB design interference, to ensure compliance with the electromagnetic compatibility design standards.





Corresponding Suppression Countermeasures of PCB Interference

Inhibition of Layout Class Interference

To suppress layout interference must ensure that the PCB layout is reasonable, PCB layout should follow the following six points:

First, according to the signal flow position reasonable set each function module circuit position, try to ensure the same direction;

Second, the layout center is locked as the core components of the module circuit, as far as possible between the various components, especially between high-frequency devices lead;

Third, the integration of thermal components, chips, etc. must be away from the heat elements;

Fourth, the location of the components on the PCB board to determine the location of the connector, as far as possible the connector placed on the side of the PCB board to prevent the cable from both sides lead to reduce the common mode current radiation;

Five is the I / O drive and the connector firmly close together to prevent the board I / O signal long distance alignment;

six is how sensitive components can not rely on too close, the output and input components to stay away.

Suppression of Slab Layer Interference

First of all, we should grasp the circuit board design information, taking into account the signal line-intensive, power and land types and other factors, in order to determine the protection of circuit functions required power and wiring layer. The quality of the stratified countermeasure has an important influence on the transient voltage of the ground plane or the power supply layer and the electromagnetic field shielding of the power supply and the signal. According to the practical experience to give the total stratification strategy, the ground plane and the power layer should be adjacent and the spacing between the two as small as possible, the signal layer should be close to the ground layer or the power layer to use one or more layers. In the design of single and double board to focus on the design of power lines and signal lines. To reduce the supply current loop area, ground and power lines to be close and keep parallel. For single-layer board, should be set in the important signal line on both sides of the defense ground, one to reduce the signal loop area, and second to avoid the signal line and the signal line between the crosstalk. In the case of a bilayer, it is also possible to set up a defensive ground, or to lay a large area on the projection plane of an important signal line. Single and double-layer board manufacturing and assembly debugging Although simple and convenient, but if such as digital circuits and digital-analog mixed circuits and other more complex PCB is not suitable for use. Because of the lack of reference plane, radiation will increase with the increase in loop area, parallel traces are difficult to avoid.

If the cost is sufficient, the proposed multilayer. In the design of multilayer board to follow three points:





First, such as the bus, the clock line and other strong radiation or high sensitivity of the important signal line, the wiring is best in the two layers or adjacent to the formation of the signal layer, wiring close to the ground plane to reduce the signal loop area, reduce radiation intensity, Strengthen anti-interference ability.

The second is to ensure that the edge of radiation is effectively controlled, compared with the adjacent ground plane power plane needs to be reduced inward 5 to 20H (H for the media thickness);

Third, if the bottom and the top of the high-frequency signal line, the need for high-frequency signal line walking in the top and between the formation, to suppress high-frequency signal line on the space radiation.

Suppression of interference of traces

PCB traces need to follow the six principles:

First, the output and input wires should be protected as far as possible adjacent long distance parallel, by reducing the parallel crosstalk by inserting the line between the lines or increasing the line spacing;

The second is not a sudden change in the width of the line, if the corner, turn the place usually go arc or 135 °;

Thirdly, the external radiation of the current-carrying circuit increases (decreases) with the increase of the current and the signal frequency as the loop area, so the wire loop area should be reduced when the current flows;

Fourth, shorten the length of the wire to increase its width, easy to reduce the wire impedance

Fifth, to ensure the same layer of adjacent lines between the noise coupling and crosstalk to a minimum, need to be done between the line isolation, to ensure separation of wiring;

Six is to set the bypass isolation of the key signal, set the protection of the line to protect the key signal. In addition, the signal line, power line and ground when the line, not only to follow the guidelines, but also with their own characteristics and functions of the implementation of wiring.

The public ground as far as possible in the PCB board edge arrangement, the best mesh or ring; ground wire to be as thick as possible, the ground to use more copper foil, to strengthen the shielding effect; analog and digital separation, Of the low-frequency to use more single-point open, the specific wiring problems can be considered part of the series and then parallel, high frequency to the best use of multi-point series.

As much as possible to increase the power cord width, weaken the loop resistance, to ensure that the ground and power lines to the same data transmission to maintain consistency. If it is multi-layer PCB, to reduce the power line to the formation or the length of the power layer. As far as possible so that the power supply to the various functional units of a separate power supply, by the public power supply circuit as far as possible close to each other and compatible with each





other.

Keeping signal lines is short enough to certainly weaken the interference signal coupling path. Firstly decorate clock signal lines and sensitive signal lines, and then decorate signal lines for high speed, and lastly decorate the unimportant signal lines. If signal lines are incompatible, you should prepare isolation processing, preventing the formation of coupling interference. When key signal lines are walking the line, they cannot span marker space, even the gap of reference plane caused by bonding pad and via hole. Otherwise, it will increase the signal circuit area. Meanwhile, to curb fringe radiation, the distance from key lines to the edge of reference plane is less than $3H$ (H means the height from key line to reference plane). Sensitive and intense radiation signal lines keep long distance from outer signal line at the port to prevent signal lines from coupling interference and reduce wrong operation of the system and radiation to the outward. The length of differential signal lines should be the same, at the same level, and parallel walk the line. Keep the impedance same and there are no other walking lines among lines. Ensuring the common mode impedance is the same and strengthening its anti-jamming ability.

In summary, combined with the conclusions of electromagnetic compatibility prediction analysis, for different types of interference should be taken to the corresponding technical measures to suppress, in order to effectively improve the PCB design quality and level. At the beginning of the design of the product when the use of simulation software on the implementation of electromagnetic compatibility prediction of PCB analysis, the basic analysis of the PCB design of electromagnetic compatibility, easy to follow the PCB design of the scientific layer, the appropriate layout and reasonable alignment. In short, PCB-based electromagnetic compatibility design is a technical and practical work is very strong, in the specific design must take into account the performance requirements, functional modules, distribution and other factors, and do predictive analysis and corresponding suppression measures, so as to effectively protect the design quality.

