



Analysis on Key Points of PCB Design

Introduction

In the current society, a large number of electronic products are widely used in everyone's daily work and life, so they need to ensure the reliability, and the vast majority of electronic systems through printed circuit board, must have a reasonable design schematic, correct printed circuit board, can fundamentally improve their reliability. For example: Suppose the distance of the two printed parallel lines is very small, will cause the signal waveform delay, the final formation of a large number of reflected noise in the terminal equipment. This paper has a series of analysis, hoping to play a role to initiate.

The key points of grounding line design

In electrical equipment, the vast majority of interference problems can be properly shielded and a reasonable grounding to solve, so we must have enough attention to grounded design work. Grounding system consists of four major components, analog ground, digital ground, chassis ground and the system ground, of which the digital ground is also called the logical ground, chassis ground is also known as shield ground. The following is to introduce several aspects to need pay attention to in the ground design.

Choosing grounding way reasonably

Usually there are two kinds of grounding, multi-point grounding and single-point grounding, so to make a reasonable choice. If the working frequency of the equipment exceeds 10MHz, it will bring adverse effects to the normal operation of the equipment due to the ground resistance is too large, so should try to choose multi-point grounding to reduce the ground impedance to achieve the purpose. Similarly, when the circuit's operating frequency is less than 1MHz, in this case, we must take a little grounding to avoid the formation of the circulation affect the interference. Therefore, the circuit is within 1 ~ 10MHz working frequency and can use multi-point grounding when the wavelength is less than 20 times than other grounding wire length, otherwise, need to use a single point grounding method.

Separation analog circuits and digital circuits

As the circuit board is very complex, the above both linear circuits and logic circuits, so they should be separated to avoid the confusion between the two, and the mixing is avoided by separately grounding the power supply terminal, at the same time, also talk about the linear circuit grounding area as much as possible to expand.

Select a thicker ground wire

Production. And the amount of inductance is inversely proportional to the width of the wire, and is proportional to the length of the inverted, so we should try to choose some thick and short wires,





which is very effective in suppressing interference. Because the bus driver, the row driver and the clock lead signal often appears the very big passing current, therefore, when we select the above line, should choose the short electric wire. For those integrated circuits, the width of the wire should be controlled between 1 ~ 0.2mm, for discrete components circuit, will control the width of about 1.5mm.

Design Essentials of Electromagnetic Compatibility

As the working environment of electronic equipment is complex and varied, it requires a better ability to adapt to the electromagnetic environment, and also reduce the electromagnetic interference to other electronic equipment which requires the design of the electromagnetic compatibility aspects, so the electronic equipment electromagnetic compatibility design is also one of the focuses of our work.

Select the correct wiring method

Through the use of parallel wiring method can significantly reduce the inductance of the wire, but will lead to Constantly increasing of distributed capacitance between the wires and mutual inductance, so the conditions permitting, we can use the well-shaped wiring structure, the specific wiring method is to take a different wiring on both sides of the printed board, one side is vertical, one side is the horizontal line, using the metal hole to connect in the cross-hole. Because there are crosstalk between the PCB conductor, so we should control the situation that appearing long-distance parallel alignment when it not appeared.

Select the correct width of the wire

Because of the frequent impact of interference, so when we printed wire to control the transient current, the main method is to control the inductance of printed wire production. And the amount of inductance is inversely proportional to the width of the wire, and is proportional to the length of the inverted, so we should try to choose some thick and short wires, which is very effective in suppressing interference. Because the bus driver, the row driver and the clock lead signal often appears the very big passing current, therefore, when we select the above line, should choose the short electric wire. For those integrated circuits, the width of the wire should be controlled between 1 ~ 0.2mm, for discrete components circuit, will control the width of about 1.5mm.

The Design Points of Device and Dimension on Board

Printed circuit board size should be moderate, if it is too large and printed lines are long, increasing impedance, not only anti-noise ability is declined, cost is also high; if it is too small, the heat is not good, easy to be disturbed by adjacent lines. In the device layout, it is like other logic circuits, as should be mutually related devices as close as possible, so you can get a better anti-noise effect. Clock generator, crystal oscillator and CPU clock input are easy to produce noise, to be close to each other. For device to easy to produce noise, low-current circuits, and high-current





circuits as far as possible from the logic circuit. If possible, it is important to make circuit boards.

Thermal design points

From the perspective of favorable heat dissipation, the printing plate is preferably installed upright, the distance between the board and the board generally should not be less than 2cm, and devices in the printed version of the arrangement should follow certain rules:

For devices employing free convection air cooling, it is preferred that the integrated circuit (or other device) be arranged in a lengthwise fashion; for equipment employing forced air cooling, it is preferred that the integrated circuit (or other device) is horizontally arranged.

The devices on the same printed circuit board should be arranged in the order of their calorific value and degree of heat dissipation. The devices with small heat or heat resistance (such as small signal transistors, small scale integrated circuits, electrolytic capacitors, etc.) placed in the most upstream (entrance) of cooling air flow .The device having a large heat capacity or heat resistance (such as power transistors, large scale integrated circuits, etc.) placed in the downstream of cooling air flow .

In the horizontal direction, the high-power devices are placed as close to the edge of the PCB as possible to shorten the heat transfer path; in the vertical direction, the high-power devices are placed as close as possible to the printed circuit board, in order to reduce the impact of these devices on the temperature of other devices .

Devices that are sensitive to temperature are best placed in the lowest temperature region (such as the bottom of the device),do not place the device directly above the heating element,multiple devices are preferably staggered in the horizontal plane.

The cooling of equipment within the printed circuit board mainly rely on air flow, so it need to study the air flow path when designing ,reasonable configuration of devices or printed circuit boards.Air flow tends to flow when the resistance is always small, so when configuring the printed circuit board on devices, to avoid leaving a larger area in the airspace.

In summary, due to the continuous development of China's electronic equipment industry, China is facing an increasingly fierce market competition in the fierce competition, we need to improve product quality, take the road of quality, so PCB design work has become a link everyone can not be ignored.

