USB PCB Layout Guidelines

Introduction

USB PCB boards are the core of the USB serial interface devices which are widely used due to their fast data transmission, convenient interface, and support for hot-swapping. At present, there are many products with USB2.0 as the interface in the market, but many hardware novices have encountered many problems in the USB application, and often the USB interface has various problems after the PCB board assembly.

For example, if the communication is unstable or unable to communicate, meanwhile there is no problem in checking the schematic and soldering. Perhaps it is necessary to suspect that the USB PCB design is unreasonable at this time. Drawing a circuit board that meets the USB 2.0 data transfer requirements is extremely important for product performance and reliability.

The USB protocol defines the transmission of digital signals by two differential signal lines (D+, D-). If the USB device is to operate stably, the differential signal lines must be laid out in strict accordance with the rules of the differential signals.

According to the engineers' many years of experience on USB related circuit design, summarize the following points,

- When placing components, try to make the differential line as short as possible to shorten the distance of the differential line (this is a reasonable way, × is an unreasonable way),

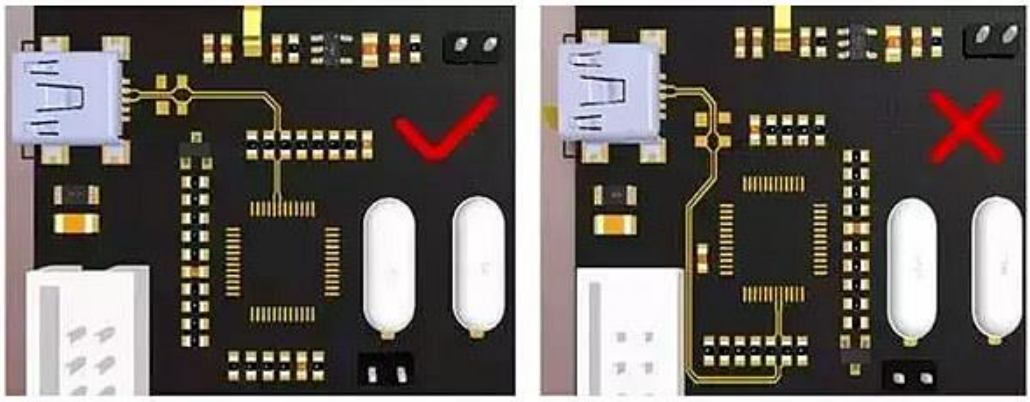

Figure 1 USB PCB Layout

- Plot the differential lines first. Do not exceed two pairs of vias on a pair of differential lines (the vias will increase the parasitic inductance of the line, thus affecting the signal integrity of the line), and need to be placed symmetrically (in a reasonable way, × in an unreasonable manner)

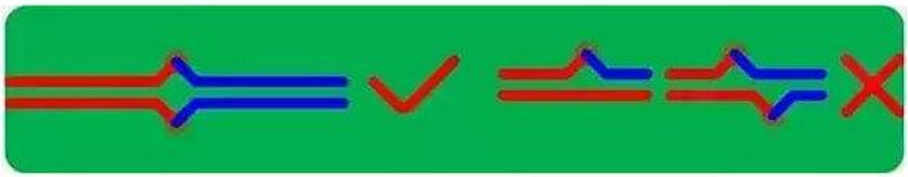

Figure 2

- Symmetrical parallel routing, this can ensure a tight coupling of the two wires, avoiding 90° routing, arc or 45° are better routing methods (√ is a reasonable way, × is an unreasonable way),

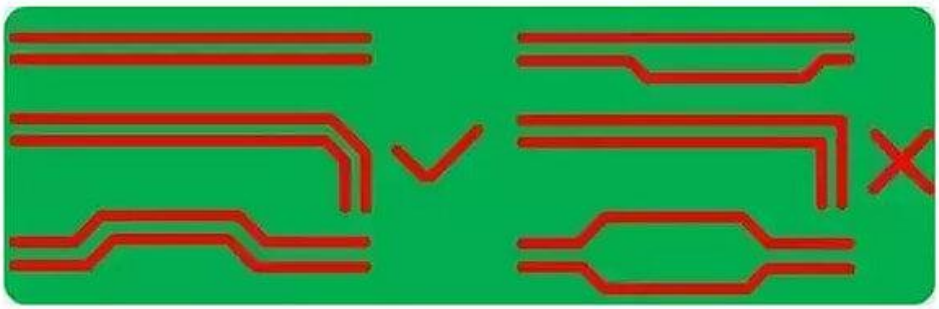

Figure 3

- Differential series connection resistance, test point, placement of the pull-up resistor (√ is a reasonable way, × is an unreasonable way),

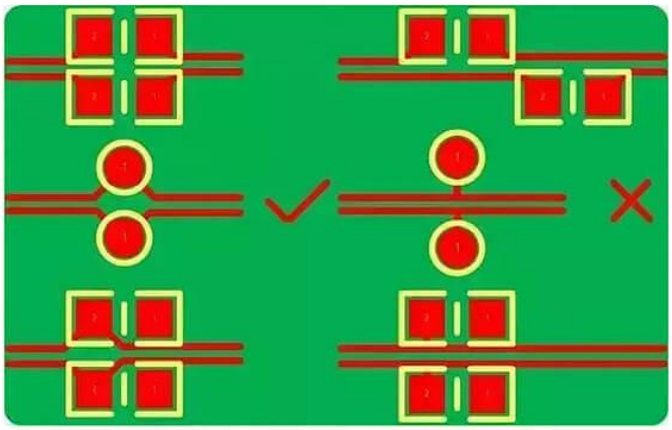

Figure 4

- Due to factors such as pin distribution, via, and trace space, the length of the differential line is not easy to match, and once the line length is not matched, the timing will shift, and common mode interference will be introduced to reduce the signal quality. Therefore, the corresponding compensation for the mismatch of the differential pair is made to match the line length, and the length difference is usually controlled within 5 mils. The compensation principle is where the length difference compensation occurs,

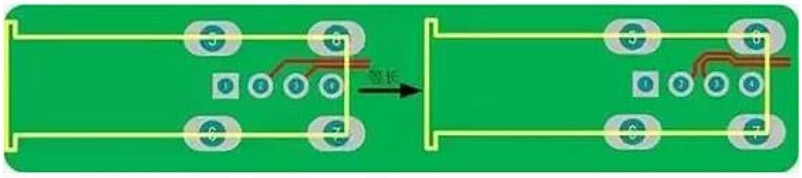

Figure 5

- In order to reduce crosstalk, the spacing of other signal networks and grounds from the differential lines is at least 20 mils (20 mils is the empirical value) when space permits. The distance between the overlay and the differential lines will affect the impedance of the differential lines,
- USB output current is 500mA, pay attention to the line width of VBUS and GND. If the 1Oz copper foil is used, the line width is greater than 20mil to meet the current-carrying requirements. Of course, the wider the line width, the better the integrity of the power supply.

Ordinary USB device differential line signal line width and line spacing can be consistent with the entire PCB board signal line width and line spacing. However, when the USB device works at 480 Mbits/s, it is not enough to do the above. We also need to control the impedance of the differential signal. Controlling the impedance of the differential signal line is very important for the integrity of the high-speed digital signal.

Because the differential impedance affects the eye diagram of the differential signal, the signal bandwidth, the signal jitter, and the interference voltage on the signal line. The differential line impedance is generally controlled at 90 (±10%) ohms.

The differential line impedance is inversely proportional to the line widths W1, W2, and T1. It is inversely proportional to the dielectric constant Er1 and proportional to the line spacing S1. The distance from the reference layer is proportional to H1, figure 6 is a cross-sectional view of the differential line.

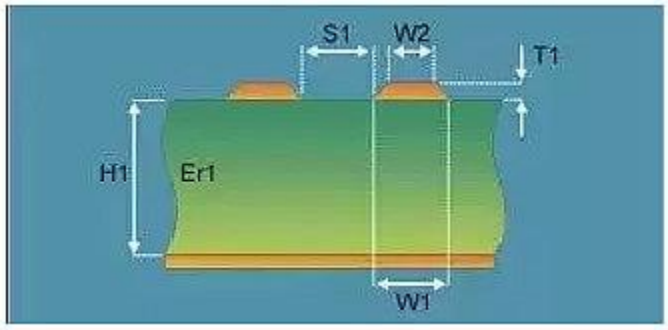

Figure 6

If you aren't sure on the trace/spacing for impedance control, feel free to contact us. PS Electronics is a 26 years' PCB manufacturer, we will come back with optimized suggestions according to the required impedance value and the actual situation of the production and the distance between the USB PCB board and the line width and the reference layer.

References

Pinsheng Electronics Co., Ltd