Computer Life (CPL) ISSN: 1819-4818
DELIVERING QUALITY SCIENCE TO THE WORLD

Analysis of Reflection Problems in High Speed Circuit Design

Qiufeng Zhou [a], Yajun Cheng [b] and Jie Cheng [c]

College of Automation, Chongqing University of Posts and Telecommunications, Chongqing 400065, China

[a] qiufeng6018@163.com, [b] jerech@qq.com, [c] 960375024 @qq.com

---

Abstract: This paper introduces the concept of signal integrity in high-speed circuits and the causes of signal reflection. It also discusses the practical ways of reducing reflections in practical engineering. Finally, through the actual design of PCB, the IBIS model of the device is imported and the topology of the signal line is derived. Analyze reflections by simulation.
Keywords: signal integrity, reflection, IBIS, simulation.

---

1. INTRODUCTION

With the continuous development of semiconductor technology, the switching speed of ICs has been increased from the time-honored MHz to hundreds of MHz and even several GHz. In high-speed circuit design, there are always problems such as reflection, ringing, overshoot, undershoot, crosstalk, and ground bombs. This article will focus on the causes of reflections in high-speed PCB designs, methods for attenuating reflections, practical methods for operation, and corresponding simulations.

2. SIGNAL INTEGRITY DEFINITION

Signal integrity (SI) refers to all signal quality issues and delays caused by interconnects, power supplies, and devices[1].Signal integrity includes many factors, the most common of which are reflections, ringing, crosstalk, and ground bounce. These unfavorable factors will cause the degradation of signal quality and signal delay. The signal obtained at the signal receiving end is different from the original signal at the transmitting end, and even the error information at the receiving end. With the increase of signal operating frequency, the integrity issue has attracted more attention from high-speed PCB engineers.

3. FORMATION AND CALCULATION OF REFLECTIONS

The reflection of the signal is closely related to the impedance of the interconnection line. The most direct cause of reflection is an abrupt change in the impedance of the interconnect, and there is an impedance discontinuity in the trace, where reflection occurs. The impedance of the two parts of the interconnecting line is different. The signal does not pass through the original signal voltage at the impedance change point. The voltage is reflected and incident and propagates to both ends respectively.

The most direct cause of reflection is the impedance, the capacitance and inductance on the unit-length signal that causes the impedance change. In design PCB design and production processes, the amount of capacitance and inductance that can be changed on a signal line per unit length is the width of the wire, the thickness of the wire, the dielectric constant of the medium, and the distance between the wire and the reference plane. Figure 1 shows the schematic diagram of reflection due to abrupt changes in the width of the wire.

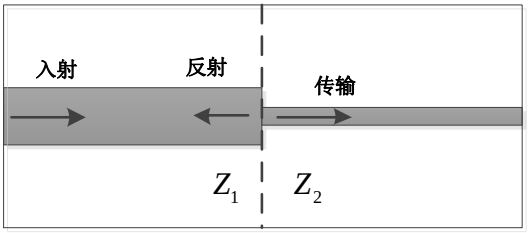

Fig. 1 Reflection caused by abrupt changes in wire width

Understanding the reflection from the perspective of electromagnetic waves, at the interface, part of the positive propagation, part of the backward propagation. From the perspective of voltage and current, at the interface, a part of the voltage is transmitted in the forward direction, a part of the voltage is transmitted in the reverse direction, the incident voltage forms a forward current, and the reflected voltage forms a reflected current, which is calculated by Ohm's law:

$$\Gamma = \frac{V_{ref}}{V_{inc}} = \frac{Z_2 - Z_1}{Z_2 + Z_1} \tag{1}$$

$$T = \frac{V_{trans}}{V_{inc}} = \frac{2Z_2}{Z_2 + Z_1} \tag{2}$$

In formula 1, Γ is the reflection coefficient, and in formula 2, T is the transmission coefficient, which is closely related to the transmission line impedance. In case of an open end, $Z_2 = \infty$, the calculated reflection coefficient is 1, all voltages are all reflected, the amplitude is the same as the amplitude of the incident voltage, and the polarity is the same. At this point the end voltage is 2 times the incident voltage. In the case of terminal short-circuit, $Z_2 = 0$, the reflection coefficient is -1, amplitude is the same as the amplitude of the incident voltage, and the polarity is opposite. Both of these extreme conditions should be avoided in the actual high-speed circuit design. In order to obtain better signal quality, the reflection coefficient should be close to 0, and the incident coefficient should be close to unity. Calculated by Equation 1 and Equation 2, the signal quality is best when $Z_2 = Z_1$[2].

4. REFLECTION PROBLEM SOLUTION IN HIGH SPEED CIRCUIT DESIGN

4.1 Solutions to Reflective Problems in Principle Design

Termination technology is used in principle design and design to improve the signal quality by connecting resistors and capacitors in series and parallel on the signal line. Possible termination methods in the project include series termination, parallel termination, David South termination, and AC parallel termination.

(1) Series termination

Series termination is a small series resistor at the source, as shown in Figure 2. If the impedance of the transmission line is 50 ohm, the nominal value of the series resistance is usually 33Ω, and the actual value varies according to the output impedance of the device.

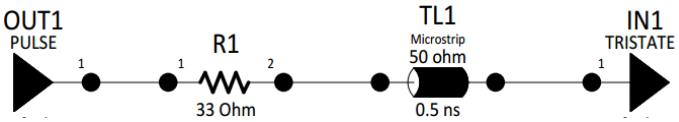

Fig. 2 Series termination

Series termination requires only one resistor, no extra DC power dissipation, elimination of secondary reflections at the driver's end, and unaffected by load changes at the receiver. However, there are also disadvantages: one reflection at the receiving end still exists; the signal edge is affected; the resistor is placed close to the driving source and is not suitable for bidirectional signal transmission; when the high and low levels of impedance are inconsistent, the reflection cannot be completely eliminated; it is not suitable for a multi-load structure[3].

(2) Parallel termination

Parallel termination is also called terminal matching. A resistor is connected in parallel with the terminal. The termination resistance is the same as the characteristic impedance value Z0, as shown in Figure 3.

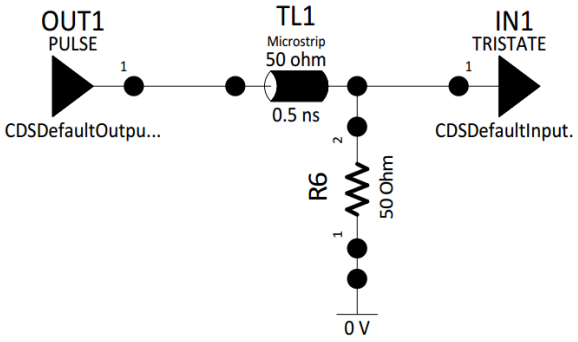

Fig. 3 Parallel termination

Parallel termination is suitable for multiple loads. Only one resistor is needed and the resistance is easy to select. However, the DC power consumption is increased. Since the pull-up and pull-down to the power supply or ground reduces the noise margin, the output current is not large enough when pulled down to ground. Can cause logic errors [4].

(3) David South termination

The South termination terminal of David South also terminates the pull-up and pull-down resistors. As shown in Figure 4, the pull-up and pull-down resistors satisfy the relationship: R1R2/(R1+R2) is equal to the impedance of the transmission line.

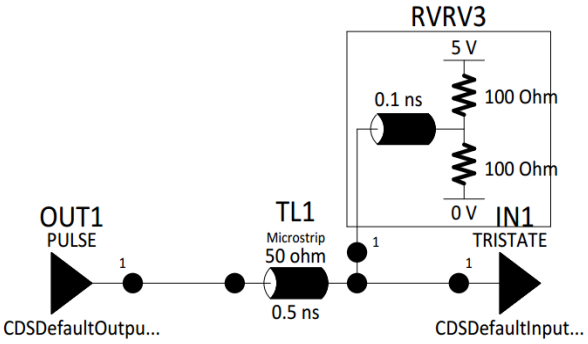

Fig. 4 David South termination

David South termination is suitable for multiple loads, but requires two devices, increasing the DC power consumption, selecting the resistance value is difficult, and it is difficult to balance power consumption and signal reflection[5].

(4) AC parallel termination

AC parallel termination, also called RC termination, requires a resistor and a capacitor, as shown in Figure 5.

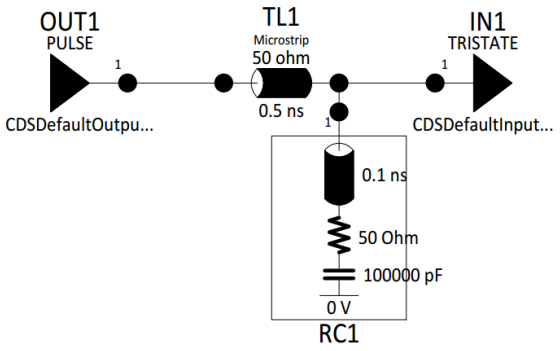

Fig. 5 AC parallel termination

The AC shunt termination is suitable for multiple loads without DC power loss, but requires two devices, increased capacitive loading of the terminal, increased delay caused by the RC circuit, and is not suitable for non-periodic signals.

4.2 Solution to Reflection Problems in PCB Design

In the PCB design, the wire width, the wire thickness, the dielectric constant of the medium, and the distance between the wire and the reference plane can all affect the impedance of the transmission line. The thickness of the wire and the dielectric constant of the medium are determined by the process of the PCB manufacturer. The distance between the reference planes is provided by the PCB manufacturer and the PCB engineer chooses. Only the wire width can be controlled by the PCB engineer. Different impedance values are calculated by different line widths. The line width corresponding to the appropriate impedance value is selected for design. After the design is completed, the IBIS model of the IC device is imported to extract the required wire topology. The model, whether the simulated signal waveform meets the requirements.

The most common impedance of the transmission line in actual design is 50 ohms. Use the Polar Si9000 to calculate the impedance. When other parameters are fixed, change the line width to get the impedance of the transmission line. As shown in Figure 6, the impedance of the transmission line is about 50 ohm when the line width is 4.5 mils.

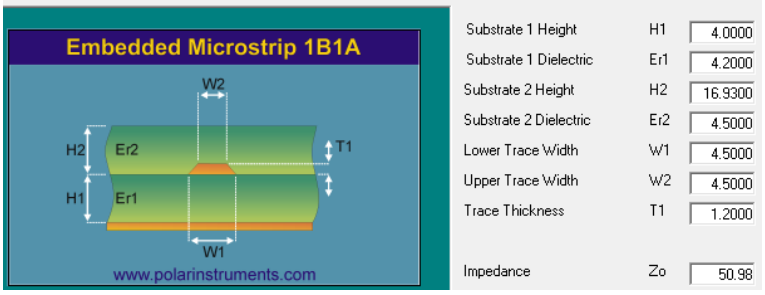

Fig. 6 Impedance calculation

5. MODEL EXTRACTION AND SIMULATION

The IBIS model can be added to the completed PCB. The Allegro PCB SI can convert the IBIS model to a DML model. The DML model is added to the corresponding device to extract a 50 ohm transmission line topology, as shown in Figure 7.

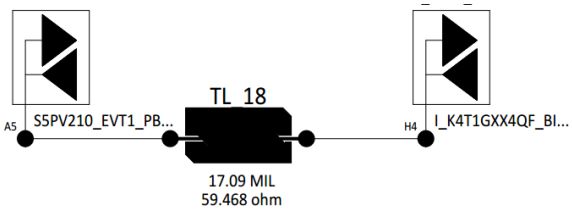

Fig. 7 Topology

Change the line width, change the characteristic impedance of the transmission line, use 55mil, 20mil, 10mil, 4.5mil, 2mil, 1mil line width to establish the topology, the source load 400MHz pulse signal, get the simulation waveform as shown in Figure 8.

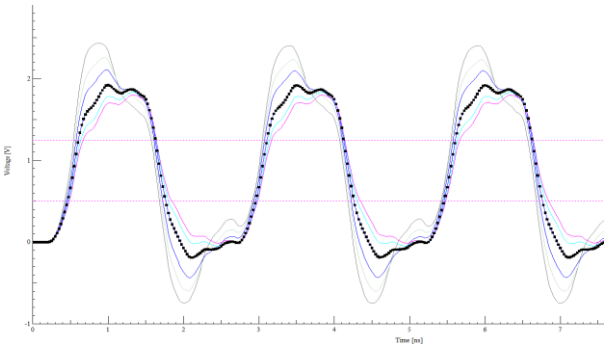

Fig. 8 Simulation waveform

When the line width is 4.5 mils, the signal quality obtained with an impedance of 50 ohm is the best, The signal is stable at a high level above VIH (min), and the low level is stable below VIL (max); $V_{top} < \sqrt{2}VCC$. Line width is too wide and too narrow causes the impedance to be too small and too large, resulting in a large overshoot of the waveform or a large step in the rising edge. In the case of impedance matching, the signal reflection effect is small and the signal quality is significantly improved.

6. CONCLUSION

The main contributing factor to the transmission line reflection effect is continuous impedance. Proper termination techniques can improve the impedance signal quality and eliminate the adverse effects caused by reflections. In the case of other PCB structure determinations, the appropriate line width is matched to the impedance, and the reflection is reduced to a very small amount, effectively improving the signal quality.

ACKNOWLEDGEMENTS

ChongQing postgraduate scientific research and innovation projects (CYS16169).

REFERENCES

[1] De Paulis F, Nisanci M H, Orlandi A. Signal integrity [J]. Electromagnetic Compatibility Magazine IEEE, 2012, 1(3):60-65.

[2] Thierauf S C. High-Speed Circuit Board Signal Integrity / S.C. Thierauf [J]. 2004.

[3] Liang B J, Sun S B, Zhang Y L. Analysis of the Signal Integrality in the Design of High Speed Circuit Board [J]. Journal of Xian University of Arts & Science, 2006.

[4] Sonoda H. The transmission line noise automatic decision system of the high speed circuit board [J]. Ipsj Sig Notes, 1997, 97:65-73.

[5] Engel A, Yu P, Giessler K, et al. Printed circuit board having improved high speed transmission lines[J]. 2017.