Crosstalk Analysis in Signal Integrity

Jiangwei Wang
School of Optoelectronic Information & Communication Engineering
Beijing Information Science and Technology University
Beijing, China
wang_jiang_wei@sina.com

Muyan Ma
School of Optoelectronic Information & Communication Engineering
Beijing Information Science and Technology University
Beijing, China
ma_my@bistu.edu.cn

***Abstract*—The signal transmission and processing is becoming faster and faster, and more and more problems occur during the high-speed circuit design. Signal Integrity is an engineering discipline that uses measurements, mathematics and simulation modeling techniques to ensure reliable electrical signaling between electronic devices. The crosstalk is one of the main issues of concern for signal integrity. The mechanism of crosstalk is introduced in this paper, and the simulation of crosstalk by ADS (Advanced Design System) is also given to analyze the method of reducing crosstalk.**

Keywords-signal integrity; crosstalk; ADS; high-speed circuit design

I. Introduction

The signal integrity problems are usually produced when the system frequency is up to 50MHz. The high-speed digital design emphasizes the behavior of passive circuit elements, and these passive elements may include the wires, circuit boards, and integrated-circuit packages that make up a digital product. At low speeds, passive circuit elements are just part of a product's packaging, but at higher speeds they directly affect electrical performance. Signal Integrity engineering fundamentally ensures that electronic circuits interconnect reliably without unintended variations in signal quality. Poor signal quality can cause computers and electronic equipment to crash and that can result in corrupt bank accounts, airplane accidents, electric grid failures, and communications blackouts. So it is necessary to analyze the signal integrity. The main issues of concern for signal integrity are loss, crosstalk, ISI (Inter Symbol Interference), skew and jitter and so on.

Crosstalk, which is the coupling of energy from one line to another, will occur whenever the electromagnetic fields from different structures interact. It will occur on the chip, on the PCB board, on the connectors, on the chip package, and on the connector cables. Furthermore, as technology and consumer demands push for physically smaller and faster products, the amount of crosstalk in digital systems is increasing dramatically. In this circumstance, more attention should be paid to the crosstalk and it is important for the engineer to learn the mechanisms that cause crosstalk and the method to avoid the crosstalk.

Supported by Beijing Science and Technology Development Program (Grant No. KM201010772008)

II. MECHANISM OF CROSSTALK

A. *Mutual Inductance and Capacitance*

Whenever a signal is driven along a wire, a magnetic field is developed around that wire. If two wires are placed adjacent to each other, it is possible that the two magnetic fields will interact with the each other, causing a cross-coupling of energy between signals, known as crosstalk. There are two types of coupling that exist which predominantly cause crosstalk.

- Mutual Inductance: This is the effect of induced current from a driven wire, or aggressor, appearing on the quiet wire, or victim, by means of a magnetic field. Mutual inductance causes positive waves to appear on the near end of the victim line (closest to the transmitter) causing near-end inductance, while negative waves appear at the far end of the transmission line (nearer to the receiver), causing far-end cross-talk. The circuit element can be described as

$$V_{Lm} = L_m \frac{dI}{dt}$$

- Mutual Capacitance: This is the coupling of two electric fields, where electrical current proportional to the rate of change of voltage in the driver is injected into the victim line. Mutual capacitance causes positive waves to appear at both ends of the transmission line. The circuit element can be described as

$$I_{Cm} = C_m \frac{dV}{dt}$$

B. *Transmission Mode*

A 2-conductor system will have 2 propagation modes: even mode and odd mode.

1) *Odd mode:*

Odd propagation mode occurs when two coupled transmission lines are driven with equal magnitude and 180° out of phase with one another. The effective mutual capacitance of the transmission line will increase by twice the mutual capacitance, and the total inductance will decrease by the mutual inductance.

978-1-61284-774-0/11/$26.00 ©2011 IEEE

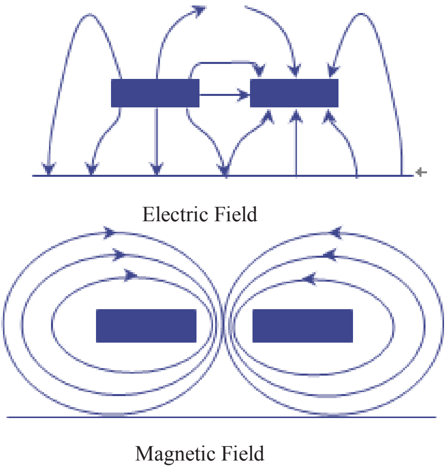

Figure 1. The odd-mode electric and magnetic field patterns

2) Even mode:

Even-mode propagation mode occurs when two coupled transmission lines are driven with equal magnitude and are in phase with one another. The effective capacitance of the transmission line will decrease by the mutual capacitance and the equivalent inductance will increase by the mutual inductance.

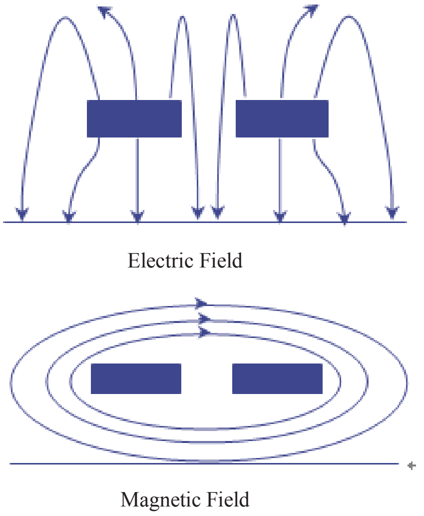

Figure 2. The even-mode electric and magnetic field patterns

C. Termination Techniques

In most aspects of a design, it is usually prudent to minimize coupling between all traces. In some designs, however, it is beneficial to have a high degree of coupling between two traces. A differential transmission line consists of two tightly coupled traces, and it is beneficial because they tend to exhibit better signal integrity than that of a single trace because the differential transmission line are more immune to noise. There are two termination techniques:

1) PI Termination

One way to properly terminate a two-line coupled pair and prevent reflection in both the odd and even modes is to use a pi network. This particular termination scheme is useful when differential receivers are used. 3 resistor networks can be designed to terminate both odd and even mode.

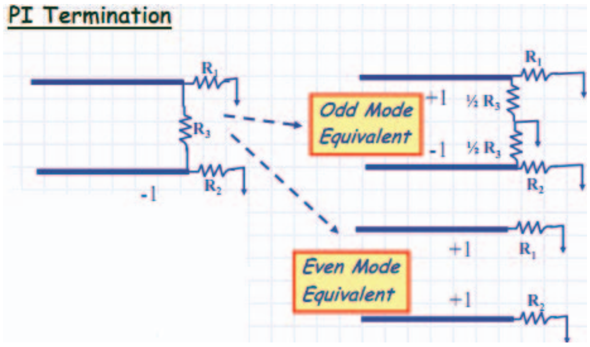

Figure 3. PI termination

The required values of R1, R2, and R3 in PI termination are as follows:

$$R_1 = R_2 = Z_{even} \tag{1}$$

$$R_3 = 2\frac{Z_{even}Z_{odd}}{Z_{even} - Z_{odd}} \tag{2}$$

2) T Termination

Another method of terminating both odd and even modes is a T-network of resistors.

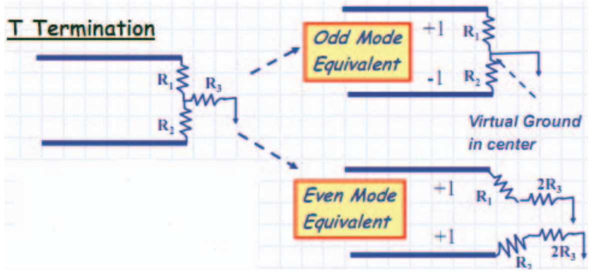

Figure 4. T termination

The required values of R1, R2, and R3 in pi termination are as follows:

$$R_1 = R_2 = Z_{odd} \tag{3}$$

$$R_3 = \frac{1}{2}(Z_{even} - Z_{odd}) \tag{4}$$

III. SIMULATION OF CROSSTALK WITH ADS

Crosstalk can be significantly reduced if some rules are followed.

1) Add matched resistors in the ports as routing restrictions.

2) Widen spacing between signal lines as much as routing restrictions will allow.

3) Design the transmission line so the conductor is as close to the ground plane as possible(less than 10 mils), this will couple the transmission line tightly to the ground plane and help decouple it from adjacent signals

4) Use differential routing techniques wherever possible, especially for critical PCB trace.

These rules followed in crosstalk can be verified by ADS (Advanced Design System). ADS allows to perform signal integrity analysis of designs such as PCI Express, DDR3, DDR4, Rapid IO, Serial ATA and 10 Gigabit Ethernet.

In the simulation, the basic parameter is set as follows:

1) The width of micro-strip is 0.3mm, the distance between traces is 0.5mm(larger than the width), the coupling length is 8.3mm, the thickness of medium is 0.25mm, and each port resistance is 50(almost equal to the characteristic impedance 45.37 of the micro-strip trace), the source is a step function. The result can be seen in figure 5.

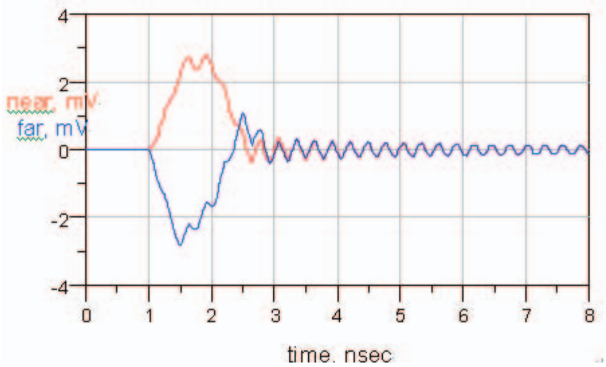

Figure 5. Crosstalk in near and far end

As can be seen, there is crosstalk in the victim line which has no source. And the crosstalk in near end is obvious larger than the far end. Furthermore, the far end crosstalk can be positive.

2) Change the impedance to 10, and the same time keep the rest parameters unchanged, the simulation result can be seen in figure 6.

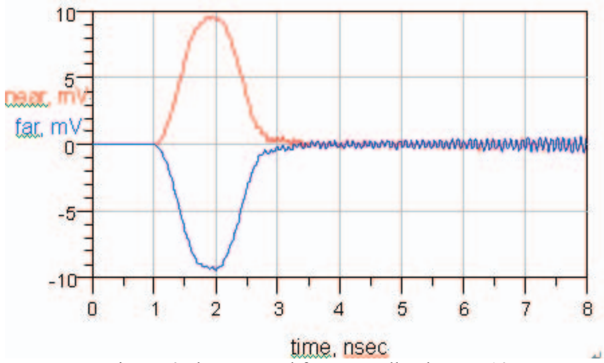

Figure 6 The near and far crosstalk when R=10

As can be seen, the crosstalk is larger since the impedance doesn't match the characteristic impedance.

3) Change the distance between traces to 1.5mm, and the other parameters are the same with condition 1.The result can be seen in figure 7.

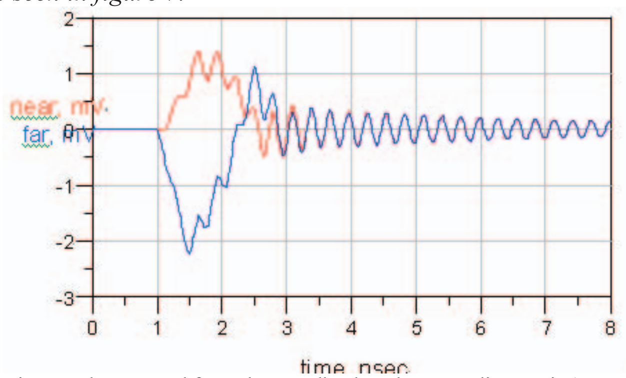

Figure 7 the near and far end crosstalk when the space distance is 1.5mm

As can be seen, the crosstalk noise is reduced since the distance becomes wider.

4) Change the thickness of medium to 0.4, other parameters are the same with condition 1.The result can be seen in figure 8.

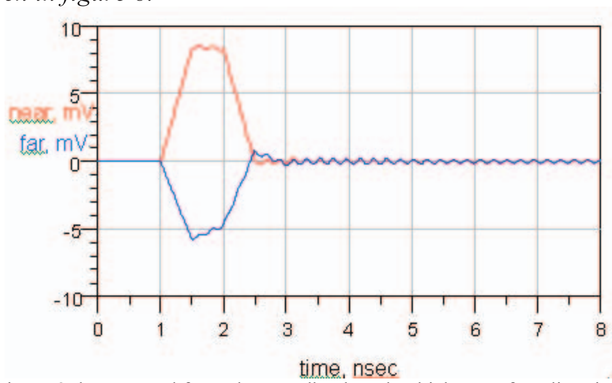

Figure 8 the near and far end crosstalk when the thickness of medium is 0.4

As can be see, the crosstalk become worse since the thickness of medium is larger.

In words, from above simulation we know that crosstalk noise can be reduced by careful PCB design. Widen the space between transmission lines, add matched impedance and minimize the thickness of medium and etc can reduce the crosstalk and improve the design.

IV. CONCLUSION

Crosstalk is one of the main problems in signal integrity, and it will affect the quality of signals directly. Signal jumping is the main cause to crosstalk, but in practical design, we can reduce the negative effect by widening spacing between signal lines, using differential routing techniques, shortening parallel run lengths between signals and so on.

ACKNOWLEDGMENT

I would like to express my sincere gratitude to the teacher Ma for her strong support and help. From the entrance to the

present, she has always been tirelessly guiding and teaching me.

REFERENCES

[1] Stephen H. Hall, Garrett W. Hall, and James A. McCall. "High speed digital system design : a handbook of interconnect theory and design practices," New York, 2000.

[2] Howard Johnson, Martin Graham. "High-speed signal propagation: advanced black magic," New Jersey, 2002.

[3] http://en.wikipedia.org/wiki/Signal_integrity.

[4] http://www.altera.com/technology/signal/fundamentals/basics/sgl-basics-cont.html#_Cross-talk

[5] Kaer Huang, Wenyi Liu, Yan Zhang, and Hongcheng Yan, "The Signal Integrity of The High-speed IC Design," Frontier in Education Conference, 2009. FIE'09.39th IEEE, Oct.2009, doi: 10.1109/FIE.2009.5350548.