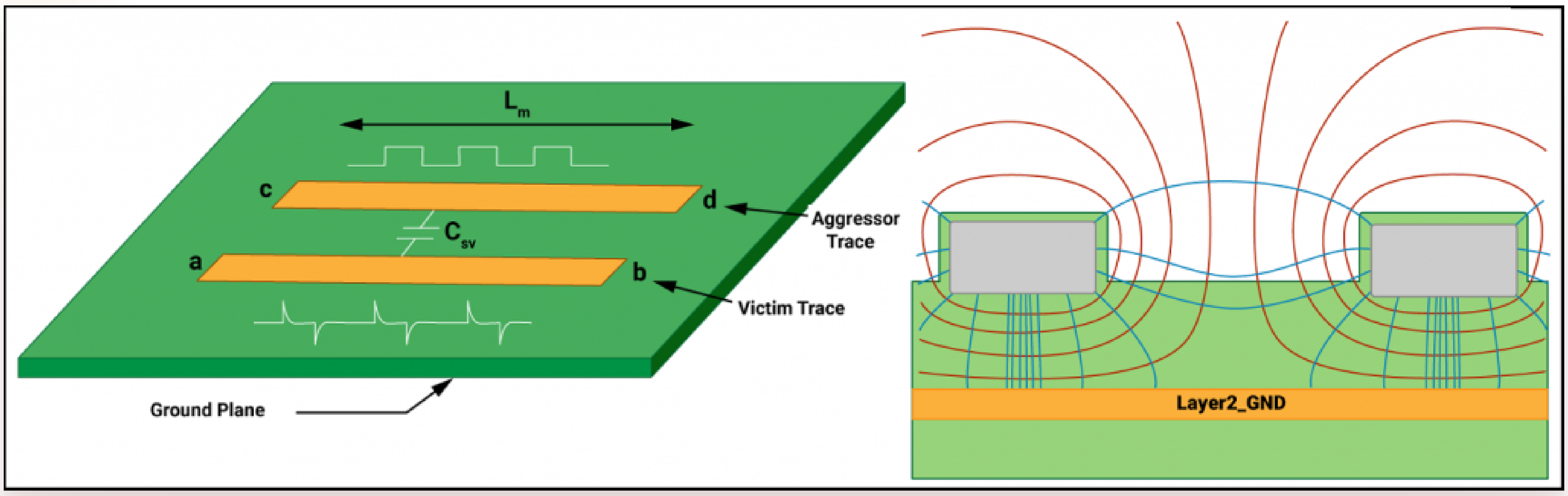
PCB DESIGN PCB TIPS SIGNAL INTEGRITY

Handle Crosstalk in High-Speed PCB Design

Signal integrity measurements have become a critical step in the process of developing digital systems. Signal integrity problems such as crosstalk, signal attenuation, ground bounce, etc., increase at higher frequencies where transmission line effects are also critical.

By **The Sierra Circuits Team**
April 19, 2022 | 0 Comments



Today's high-level functionality electronics stipulate the use of HDI PCBs that work with high-frequency range (RF/Microwave) of 1 to 10 GHz. This increased frequency causes steeper edges in signal response. Boards become more compact and gradually tight wiring density gives rise to the importance of elementary analysis of crosstalk.

EMI goes up as faster edge speed produces shorter wavelengths relative to the bus length, creating unintended radiated emissions. These emissions increase crosstalk and can cause a high-speed PCB design to fail during EMI/EMC testing.

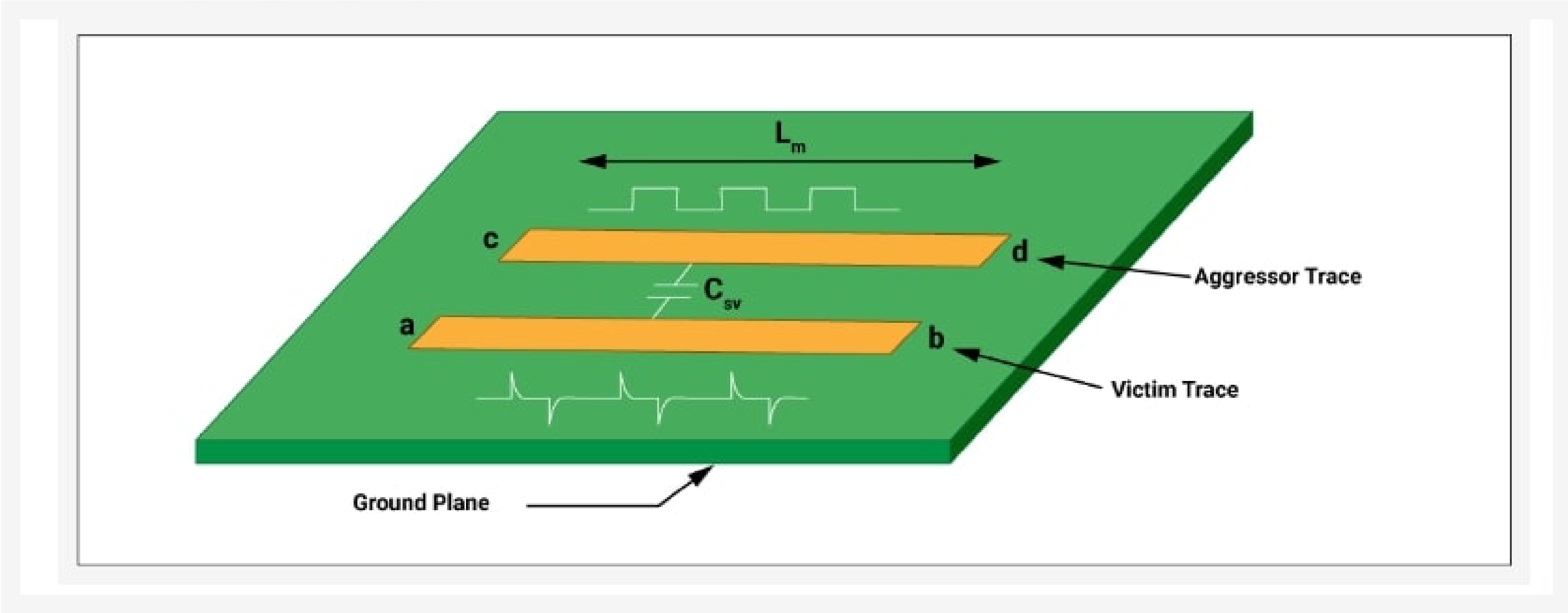
What is crosstalk in a PCB?

Crosstalk is the disturbance caused by energy coupling from one PCB trace to another even if they are not in contact. In other words, unwanted electro-magnetic coupling between closely spaced traces in high-speed PCBs is referred to as crosstalk. It happens due to the interference of electric (capacitive coupling) and magnetic fields (inductive coupling). The magnetic field generates mutual inductance, and the electric field generates mutual capacitance between the traces in the vicinity. Mutual inductance is responsible for inducing current on the adjacent (victim) line, which is opposite of the current in the aggressor line. And the capacitor formed due to mutual capacitance will pass the current in both directions on the victim line. It hampers the signal integrity of a circuit board.

Simulation, using SI tools, can determine crosstalk after the initial layout routing. Although, it remains undetected sometimes. If not controlled at the initial stage, crosstalk may degrade the board functionality. Basically, it is all about fringe electric and magnetic fields that are not confined but spread out in the vicinity region.

SHARE THIS ARTICLE

Search our Education Center



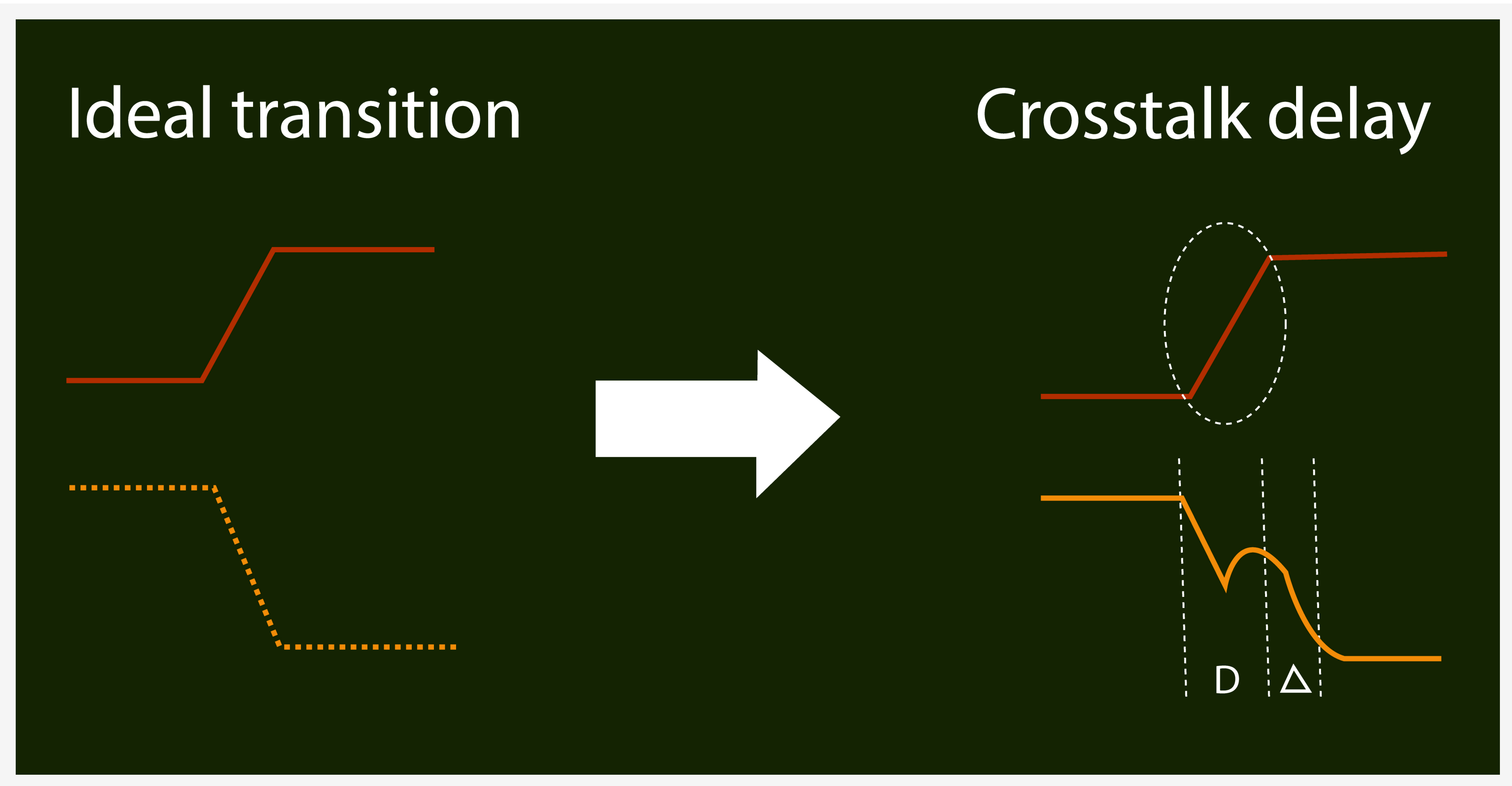
Crosstalk induced in the victim trace (ab)

Crosstalk arises when two traces run next to each other in the same layer or one over the top of the other in adjacent layers. Consider two traces running in the same direction. If the signal flowing through one trace has a higher amplitude than the other, it could affect the signal flowing through the other trace. Here, the trace with a higher amplitude will be called "aggressor" and the other trace is referred to as "victim."

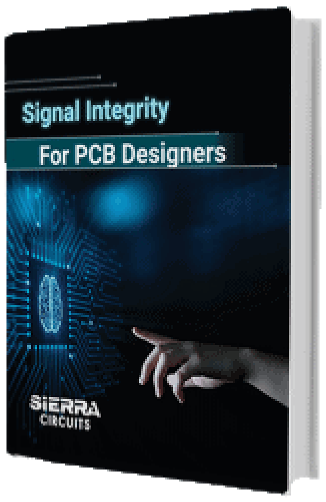
In such a scenario, the signal in the victim trace will start imitating the characteristic impedance of the aggressor trace instead of conducting its own signal. When this happens, it means crosstalk has invaded the system.

Impact of crosstalk on a PCB

- Degrades signal integrity of the board
- Timing delay
- Overshoot of voltage
- Chaos in logic functions due to false triggering

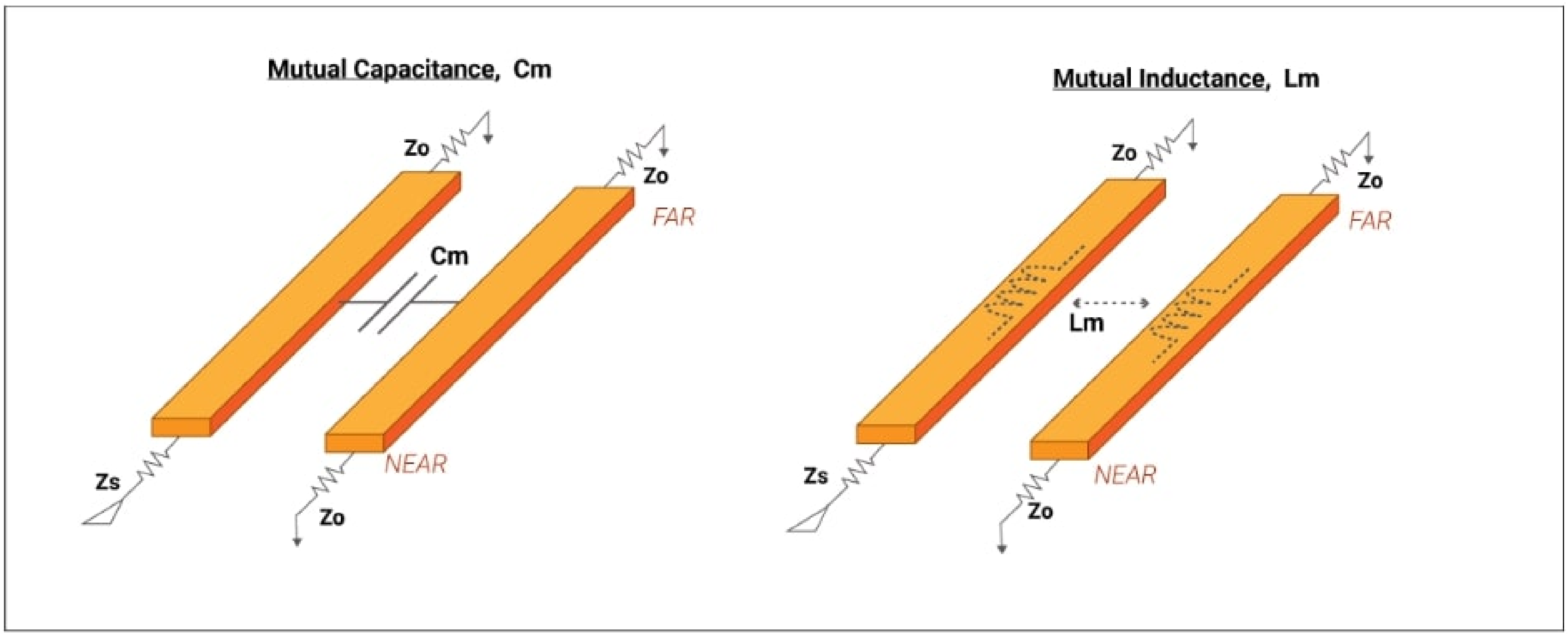


Timing delay due to crosstalk in a PCB



How does crosstalk induce noise in a system?

Every electrical signal has a varied EM field. Whenever these fields overlay, they produce inductive, capacitive, or conductive coupling resulting in EMI. Crosstalk makes the aggressor signal overpower the victim signal, even though they are not physically attached but are in close proximity. Crosstalk is divided into **common impedance coupling** and **electromagnetic field coupling**. The easy-to-analyze common impedance coupling happens when multiple signals share a common return path. Electromagnetic field coupling is further categorized into **capacitive** and **inductive** coupling, which are the most concerning factors.

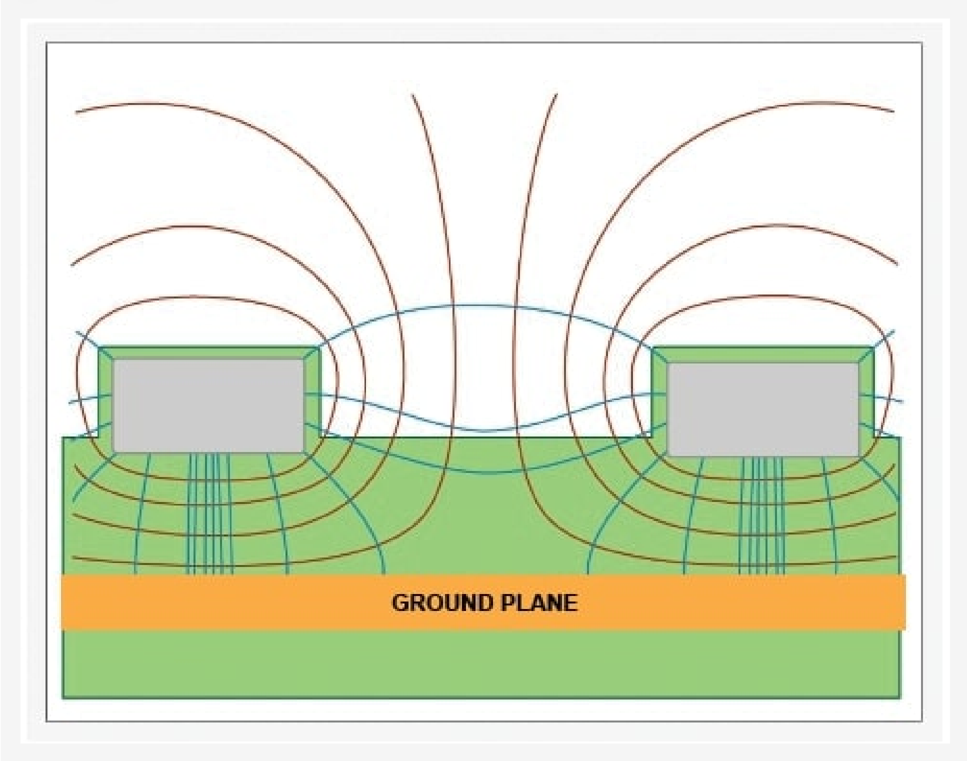


Crosstalk induced due to mutual capacitance and inductance

Traditional analysis of crosstalk with a pen and paper is not only troublesome but also a time-consuming process. What we can do is estimate the coupling considering the circuit elements. The coupling due to fringe electric fields is approximated by a capacitor (mutual capacitance) and fringe magnetic field caused coupling is assessed by an inductor (mutual inductance).

Capacitive coupling due to change in electric field

Two traces working parallelly on a plane act as parallel plates of a capacitor separated by a dielectric. The change in the electric field between aggressor and victim is modeled as a capacitor. We know if there is a change in voltage in the capacitor, the electric field changes, and the displacement current is induced. Here, the capacitively coupled current is the crosstalk we refer to. This effect is known as parasitic capacitance. To learn more read how to reduce parasitic capacitance in PCB layout.



Mutual capacitive and inductive coupling

Inductive coupling through magnetic field rings

Magnetic coupling or inductive coupling is approximated by how much magnetic field rings revolve around the victim. The magnetic field change generated by the aggressor surrounding the victim results in changing the conduction current. Thus, voltage is induced on the victim line as per Faraday's law of induction. This induced voltage drives the current in the form of crosstalk.

Broadside coupling

Generally, crosstalk happens between two adjacent traces on the same layer. In addition to that, parallel traces in the surroundings separated by a small dielectric contribute to crosstalk. This dielectric thickness can be either **4 mils (0.1mm) or less than the spacing between two traces**. This is referred to as broadside coupling.

What are the different types of crosstalk?

Here are different types of crosstalk:

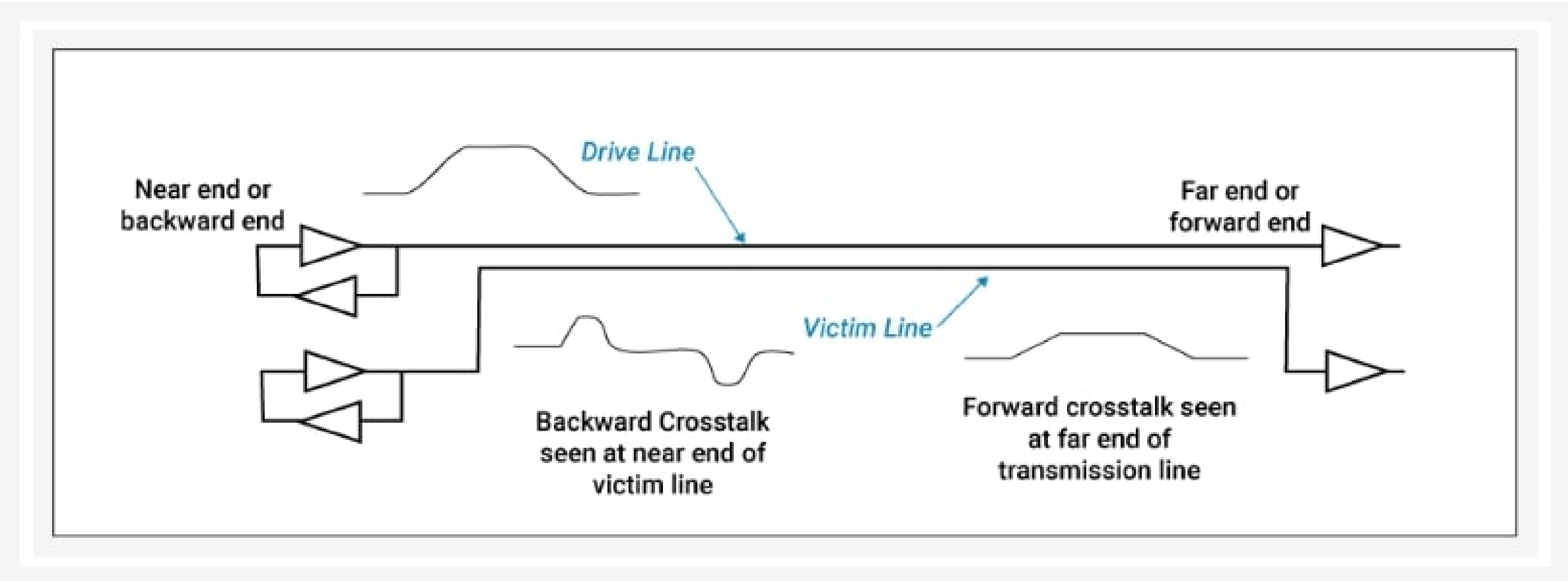
Based on the direction of propagation

- **Forward crosstalk:** It propagates along the direction of the aggressor signal.

  Forward crosstalk = Capacitive coupling – Inductive coupling

- **Backward crosstalk:** It propagates in the opposite direction of the aggressor signal.

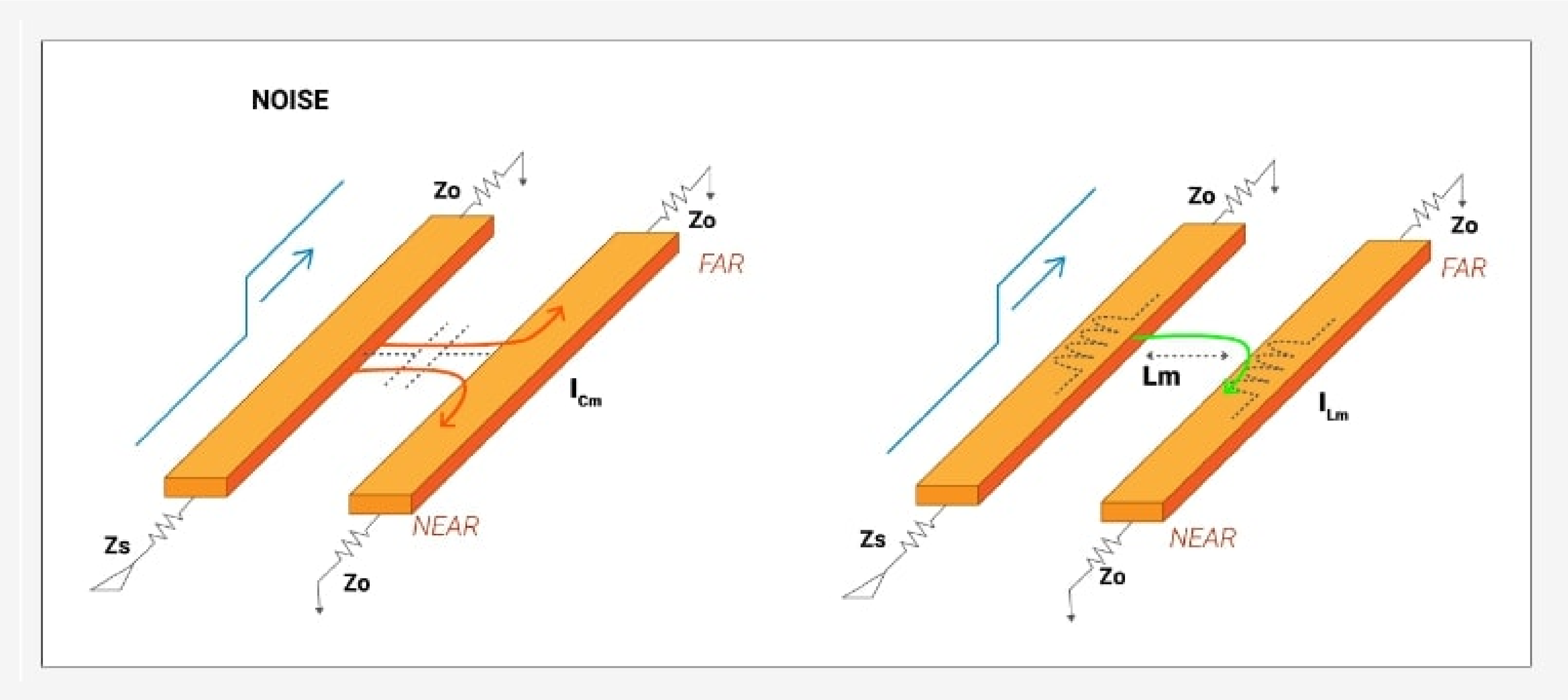
  Backward crosstalk = Capacitive coupling + Inductive coupling



Forward and backward crosstalk

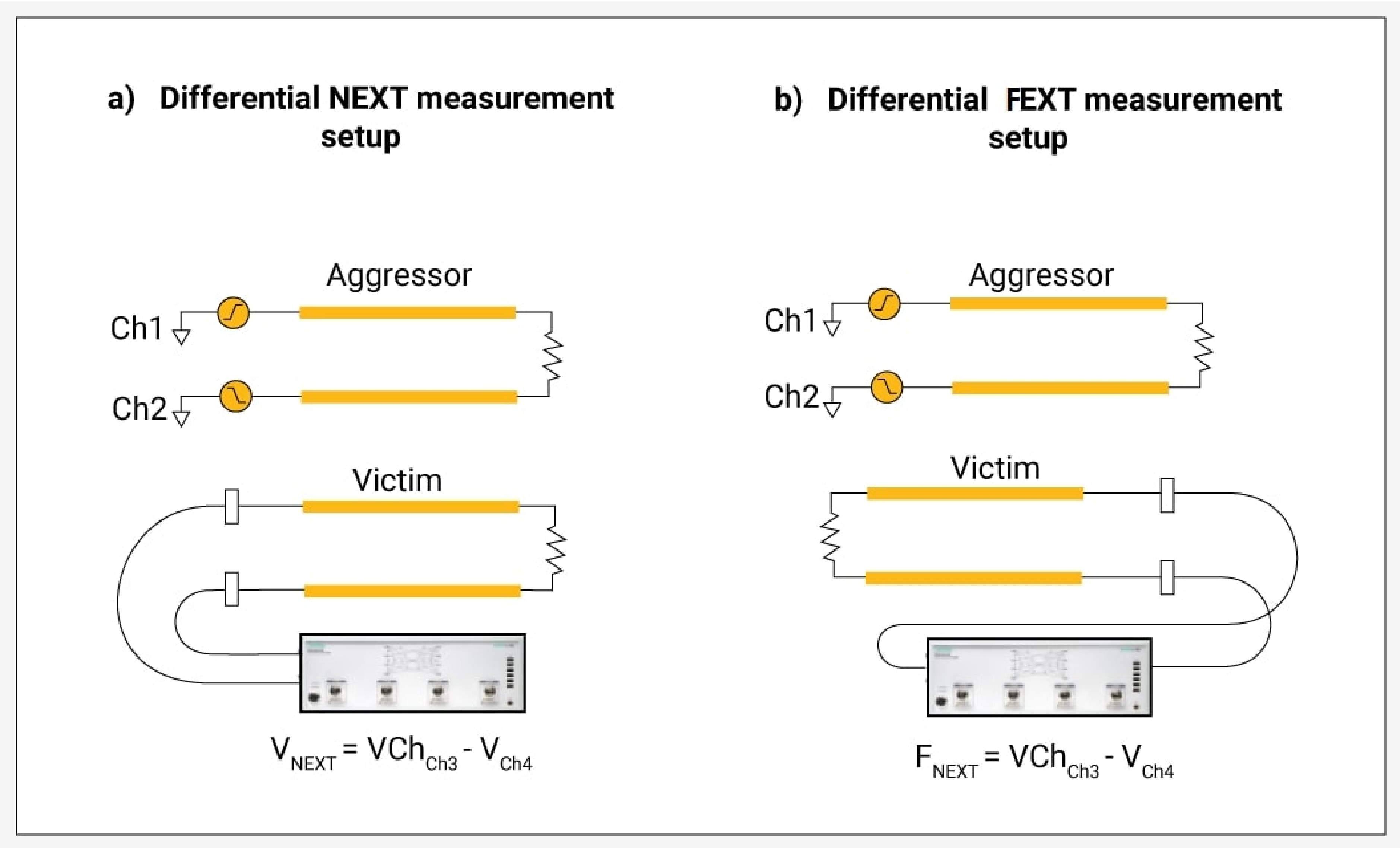
Based on measurement zone

- **Far-end crosstalk (FEXT):** It is the disturbance on the receiver side of the victim line.



Near-end and far-end crosstalk

NEXT and FEXT are measured with respect to the port to which the stimulus is applied. It can occur anywhere along a line, whether it is a dual conductor or single-ended.



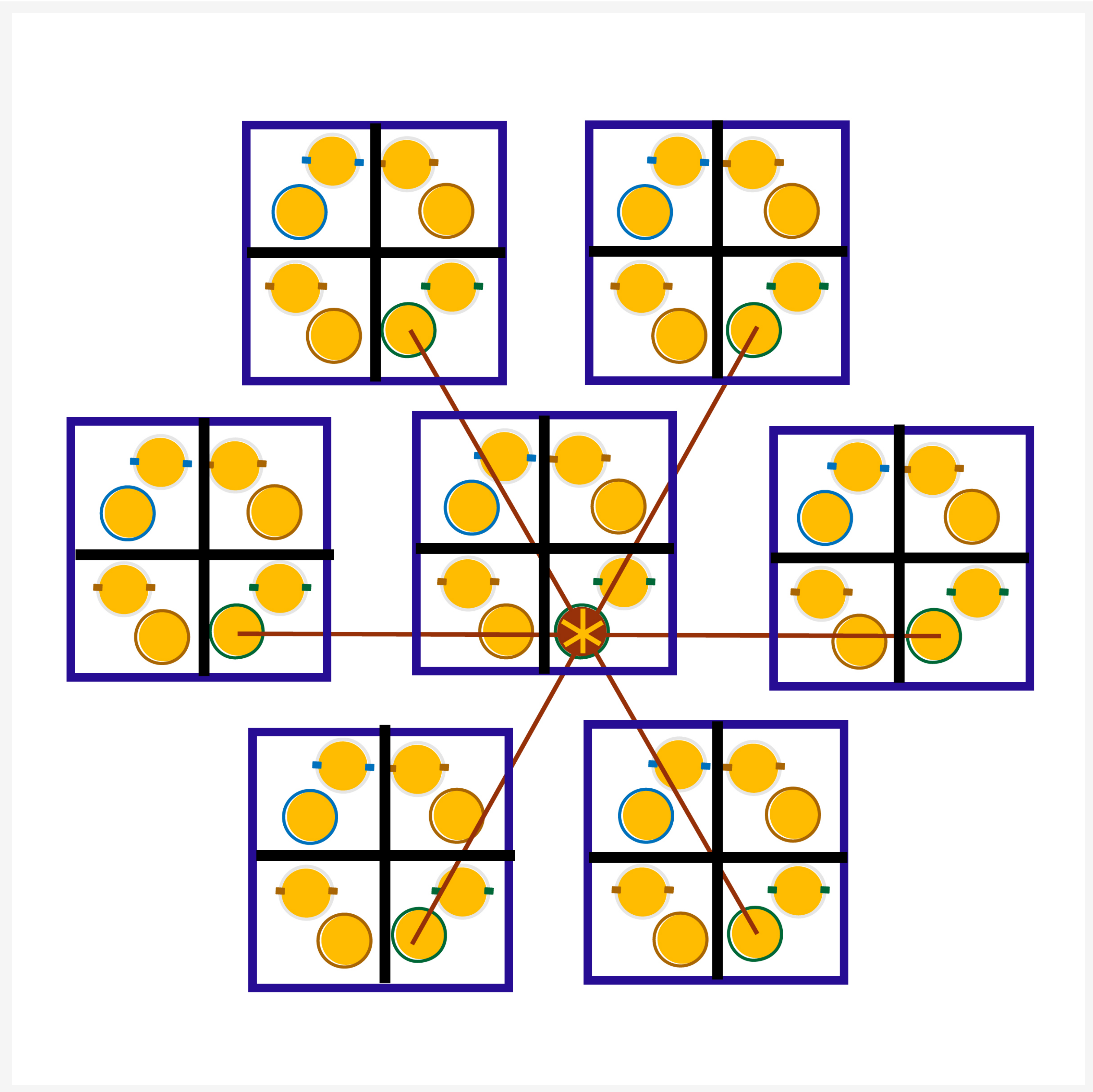
Differential NEXT and FEXT measurement

Note: The NEXT value is expressed in decibels (dB) and varies with the frequency of transmission. A higher dB of NEXT means less interference.

Based on quantification

- **Power-sum-NEXT(PS-NEXT):** Absolute or relative power of near-end crosstalk. PSNEXT gives total crosstalks from all the adjacent pairs and involves measuring all pair-to-pair groupings relative to power.
- **Power-sum-FEXT (PS-FEXT):** Absolute or relative power of far-end crosstalk.
- **Power-sum-equal-level-crosstalk (PS-ELFEXT):** Sum of PS-NEXT and PS-FEXT.

Alien crosstalk is very complex and cannot be eliminated easily by phase cancellation. Here, multiple signals of different frequencies mix up with the victim signal, and the situation gets worse when bandwidth has to be increased for a faster response. Shielding is not sufficient to prevent this from occurring. Crosstalk can also be measured using a TDR. For more details, read our post on how TDR impedance measurements work.



Alien crosstalk

S-parameters

S-parameters describe the microwave properties of interconnects in PCBs. Each S-parameter element is basically the ratio of sine wave coming out of the terminating end of an interconnect to the sine wave entering the starting terminal of another interconnect. As they are ratios of sine waves, each element (S31 and S41) is complex. Standard impedance is always 50Ω.

S-parameters are used for the basic analysis of crosstalk. They automatically detect the port into which the signal enters and the port from which the signal comes out. It is crucial to determine how much crosstalk occurs in the lines as well as the direction of crosstalk is measured from the farthest port of the victim with respect to the aggressor signal. It propagates in the forward direction.

Le croy VNA result. Image credit: EDN

Even though S31 and S41 measure noise on the same victim caused by the same aggressor, their values differ. Here it can be shown by Le croy signal integrity network analyzer.

In both cases, the vertical scale is 40db full scale whereas the horizontal scale for the left one is 1 GHz full scale and for the right it is 20 GHz. The difference between S31 and S41 can be clearly seen here. We can also probe s-parameters using a vector network analyzer (VNA). To learn more about this, see s-parameters measurement using a vector network analyzer.

How is crosstalk measured?

Crosstalk is generally specified as a percentage of the signal that appears on the victim line, relative to the aggressor line. It can also be expressed in terms of dB below the driven line level. NEXT varies with the frequency of the transmission since higher frequencies create more interference. The higher the dB value, the less crosstalk is received by the disturbed link/channel. FEXT is calculated from the crosstalk elements of the system S-parameters.

The formula for crosstalk is given by:

$$\frac{K\ (H^2)}{(H^2 +\ D^2)}$$

Where:

K is a constant whose value always remains less than 1 and depends upon the rise time of the circuit and the length of the traces experiencing crosstalk.

H2 is the product of the height of the parallel traces.

D2 is the product of the direct distance between the centerline of the traces.

The above equation clearly shows that crosstalk can be minimized by reducing H and maximizing D.

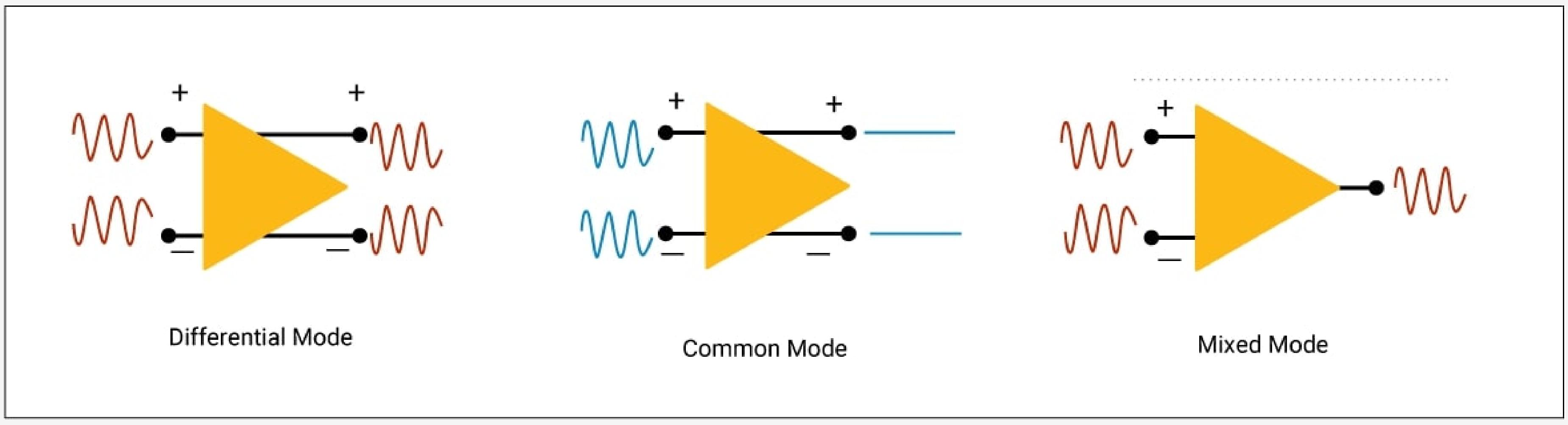
Crosstalk in dB is given by:

$$20 \log \frac{V_{victim}}{V_{aggressor}}$$

Where, **V**victim is the voltage on the victim line and **V**aggressor is the voltage on the aggressor line.

- Degree of coupling between aggressor and victim lines
- The distance up to which coupling occurs
- Effectiveness of the type of termination used

How is crosstalk induced in a differential pair?



Crosstalk in a differential pair

Whenever there is an imbalance in a differential system, the fields no longer completely cancel, which causes them to radiate in proportion to the imbalance. Similarly, external fields can induce currents in a differential pair that are not equal in amplitude and opposite in phase, so they no longer cancel. The resultant current is called common-mode current. Common mode crosstalk has more adverse effects on the system performance than the differential mode.

Comparison between common mode and differential mode crosstalk effects with respect to frequency. Image credit: Intel

What are the causes for crosstalk?

Capacitive and inductive coupling: Capacitive coupling is due to parasitic capacitance and inductive coupling occurs due to mutual inductance.

- Difference in propagation velocity: It can be avoided by trace length matching and propagation delay matching.

- Increased data rates: With increased data rate, the rise time increases as well. According to Faraday's law, with an increase in rise time, the crosstalk will also increase. One way to reduce crosstalk between such signals is to increase the spacing between the traces.
- Board size: As the board size increases, the trace lengths also increase, and these traces behave as antennas. So, it is important to keep the trace lengths as minimum as possible.

Design guidelines to reduce crosstalk

Crosstalk is inevitable in high-frequency boards. What we can do is mitigate it in such a way that it becomes insignificant. Here are some quick popular techniques to avert crosstalk:

Maintain adequate separation between the traces

Provide adequate separation between traces (adopt 3W rule). If adequate separation is not maintained, then it will increase the mutual capacitance (Cm). The 3W rule reduces the crosstalk by 70%. To achieve 98% crosstalk reduction, go for 10W.

Use of solid reference plane

The use of a solid reference plane or return path is to absorb the fringe electric and magnetic fields. Therefore, they can not spread out and make noise on other signals.

Make use of segregated transmission lines

Crosstalk is induced by the aggressor trace onto the victim trace, so it is obvious that a higher aggressor voltage will induce more crosstalk. Therefore, it is best to segregate groups of nets according to their signal amplitude. This strategy prevents larger voltage nets (3.3V) from affecting smaller voltage nets (1.5V).

Implement back drilled vias

Via stubs decrease signal integrity, hence an increase in crosstalk. This can be reduced by implementing back drilling.

Reduce parallel trace runs

Longer trace runs (more than 500mils) increase the mutual inductance hence crosstalk.

Adopt orthogonal routing

Route adjacent signal layers orthogonally to minimize capacitive coupling between them.

Do not decrease the signal rise time

Decreased signal rise time increases the crosstalk.

Opt for differential pair routing

Tightly coupled differential routing eliminates crosstalk because the noise from the aggressor is coupled equally into both branches of the differential pair, producing common-mode noise. Differential pairs reject the common mode noise that helps with crosstalk reduction.

Use trace of short length

In practical design, it is not always possible to use signals perpendicular to each other. Then keeping trace length as short as possible is the feasible technique to reduce the chance of coupling.

Isolate high frequency signals

Isolate asynchronous signals

Asynchronous signals must be kept away from high-speed signals. As they come into use very infrequently in normal circuit operation, we can place them near power lines.

Use guard traces

Guard traces are used to control capacitive crosstalk between transmission lines. Such traces should be used wisely as they make routing difficult.

Terminate even and odd mode transmission properly

A three resistor network (T termination) can be used to terminate the odd and even mode.

T termination for even and odd mode transmission

Limit of crosstalk

Make sure that the overall system crosstalk does not exceed 150mV.

How does crosstalk affect transmission line parameters?

Transmission line can be called a two-conductor system where two separate traces affect the propagation of the signals through them. Two propagation modes can be considered: even mode (both line in-phase) and odd mode (lines 180 degrees out of phase).

In odd mode transmission, a considerable potential difference will exist between the two lines. This potential difference will increase the effective capacitance equal to the value of mutual capacitance.

Since currents in both the lines are flowing in opposite directions, it will decrease the total inductance by mutual inductance (Lm) value.

Current flow during odd mode transmission

Transmission line impedance for odd mode is given by:

Note that Z differential = 2Zodd

Transmission line propagation delay for odd mode is given by:

In even mode transmission, the two lines (victim and aggressor) will always be of equal potential. This will reduce the effective capacitance by mutual capacitance value.

Field lines during even mode transmission

Since currents in both the lines are flowing in the same direction, it will increase the total inductance by mutual inductance (Lm) value.

Current flow during even mode transmission

Transmission line impedance for even mode is given by:

Transmission line propagation delay for even mode is given by:

Crosstalk in different transmission lines

The capacitively coupled current propagates both clockwise and counterclockwise directions. The burst of current in the forward direction incidentally concurs with the aggressor signal. Thus, the far-end crosstalk due to capacitive coupling increases. The current in the backward direction dribbles back repeatedly, making the near-end crosstalk the same in magnitude but long in duration. Here in both directions, the voltage pulses are positive.

Inductively coupled current circulates clockwise. The phenomenon in forward and backward is the same as capacitive coupling except for the fact that here voltage pulse resulting in far-end crosstalk in the forward direction is negative.

Capacitively and inductively coupled current

In a stripline, the environment above and below the signal line is homogeneous. Thus, the resulting far-end crosstalk from both coupling will cancel out each other. But the scenario mismatches if we introduce the microstrip line as a transmission line. There is air above the microstrip whereas below it is dielectric. The difference in medium causes far-end crosstalk to elevate. Basically dielectric is the reason for electrical coupling. So, we can conclude in the case of a microstrip transmission line, we can decrease the capacitive coupling but far-end crosstalk increases. For details, read what is the difference between microstrip and stripline?

Crosstalk cannot be reduced at the system level. Integrated modeling and characterization cycles can be used to mitigate crosstalk on device or package levels. If not controlled properly, it can turn your board into non-functional. Even though PCB designers ensure minimum separation between the traces, it may not be enough for associated issues.

For HDI boards, check out how to avoid crosstalk in HDI substrate.

Let us know in the comments section if you want to learn something specific to signal integrity and transmission lines. We will be happy to resolve your queries.

Steve Sandler on Power Integrity in PDN Design and High-Speed Simulations

Next-Gen Memory DDR5 and Signal Integrity in High-Speed PCBs with Stephen Slater

VNA or TDR: The Right Time-Domain Measurement by Mike Resso

Subscribe

Be the First to Comment!

B I U S

0 COMMENTS

Reduce Re-Spins with our **Zero Defect Guarantee.**

Our platform sources the whole package, creating a thread from design to delivery. Fabricating PCBs. Procuring parts. Assembling your boards.

Once you upload your BOM, Our best-in-class software automatically checks for errors and cleans your file for you. After you place an order, your dedicated CAM engineer will run a full DFM and DFA check to ensure your prototype achieves Zero Defect every time—all at no additional cost to you.

Get to market faster and spend less time troubleshooting.

Start Quote →

Talk to a Sierra Circuits PCB Expert today

24 hours a day, 7 days a week.

Call us: +1 (800) 763-7503
Email us: through our Customer Care form

AS9100D | IPC-A-600 Inspectors | IPC 6012 CLASS 2 | IPC 6012 CLASS 3/ES | UL Certification - Flex & Rigid Flex PCBs | UL Certification - Rigid PCBs | RoHS / REACH Compliance | ISO 9001:2015 | ISO 13485:2016 | MIL-PRF-31032 | MIL-PRF-55110 | ITAR Registered | Defense Logistics Agency (DLA) Approved

Sierra Circuits is a minority-owned business, certified by the National Minority Supplier Development Council.

Phone: (800) 763-7503
Fax: (408) 735-1408

Sierra Circuits, Inc.
1108 West Evelyn Avenue
Sunnyvale, CA 94086, USA