

The Interaction of Electromagnetic Fields

Beyond Design

by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA
with Special Advisor Rick Hartley, R HARTLEY ENTERPRISES

When two (or more) electromagnetic fields overlap or meet, they add vectorially at each point in space. Fields have direction and polarity. At any point in space, there can be only one field, so at some spatial points, they will cancel each other, and at others, they will re-enforce each other. James Clerk Maxwell described electromagnetic fields as being linear. Linearity implies superposition, meaning that the fields do not merge with each other but rather add vectorially—distorting the signal. This applies to both static (DC) and time-varying (AC) fields.

When an electromagnetic field propagates along a transmission line, there are electric field lines between the signal and return paths and rings of magnetic field lines around the signal and return path conductors. These fields are

not confined to the immediate space between the signal and return paths but rather, fringe fields spread out into the surrounding volume.

In a stripline (inner) configuration (Figure 1a), the electromagnetic field propagates between the planes, and the return displacement current flows in close proximity (mainly) to the nearest plane. As the frequency increases, the current is forced into the outer surface of the copper, due to the skin effect, dramatically increasing loss. Figure 1b shows the electromagnetic field in a microstrip (outer layer) configuration. Electric fields terminate when they come into contact with a solid plane, whereas magnetic fields tend to radiate into the dielectric or air above or below the PCB. However, stripline (embedded signals) are rather confined by the adjacent planes and only emit radiation on the

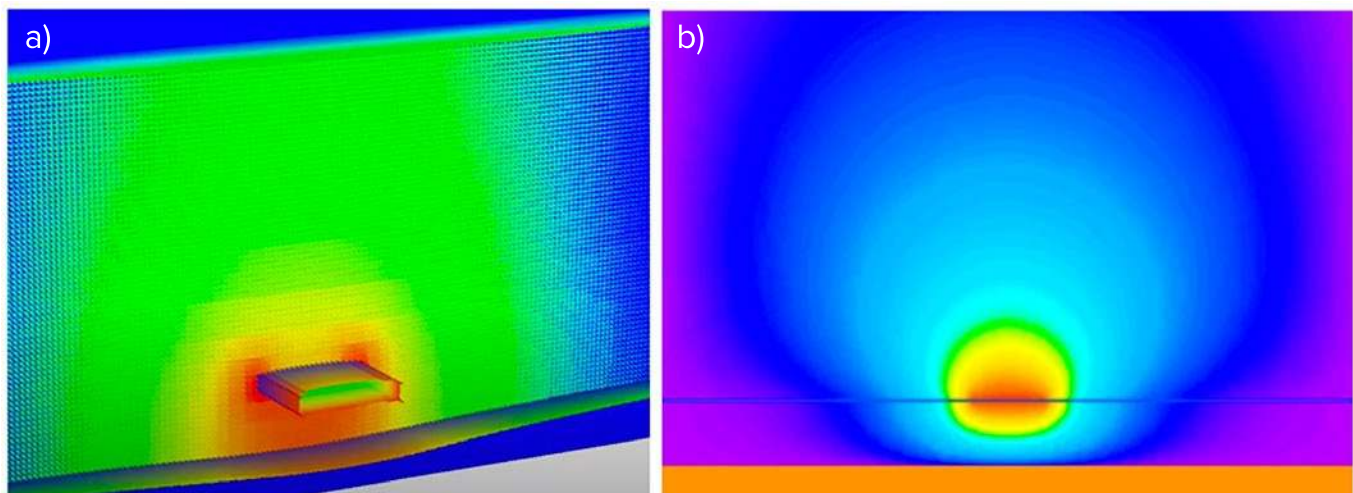


Figure 1: a) EM fields in stripline configuration.; b) Microstrip configuration.
(Source: Simboer & QuickField)

fringing fields of the PCB edge. This is a good reason to embed all critical signals.

Both electric and magnetic field coupling can occur on PCBs; however, crosstalk usually comes from electric field-based capacitive coupling. Trace segments running in parallel on the same layer or running vertically (broad-side) between two layers are extremely susceptible to crosstalk. Parallel traces act as capacitor plates separated by a dielectric. If there is a change in voltage, it will alter the electric field and generate a displacement current leading to crosstalk. Mutual capacitance between the two lines couples energy in both directions on the victim trace (Figure 2a).

Inductive coupling depends on the magnitude of the magnetic field surrounding the victim trace. The magnetic field change generated by the aggressor signal surrounding the victim trace results in a changing conduction current. Mutual inductance between the lines couples energy onto the victim trace segment in the backward direction (Figure 2b).

Though both mutual capacitance and mutual inductance play a role in crosstalk, there are two regimes to consider. When the return path is a wide, uniform plane, as is the case for most coupled transmission lines in a mul-

tilayer PCB, the capacitively coupled energy and inductively coupled energy are of the same order of magnitude. However, when the return path is a single lead in a package, or a single pin in a connector, the inductively coupled energy is much larger than the capacitively coupled energy. In this case, the noise behavior is dominated by the inductively coupled energy. The noise on the victim line is driven by a dI/dt in the aggressor net, which happens at the rising and falling edges of the signal when the driver switches.

Electromagnetic fields have direction and polarity. Forward (or far-end crosstalk, FEXT) and backward (or near-end crosstalk, NEXT) refer to the direction the crosstalk travels along a victim transmission line. Forward crosstalk propagates in the same direction as the aggressor signal, while backward crosstalk propagates in the opposite direction.

- FEXT = (Capacitive coupling – Inductive coupling)/2
- NEXT = (Capacitive coupling + Inductive coupling)/4

In an outer layer microstrip configuration, the mutual capacitive coupling between adjacent traces is generally weaker than the mutu-

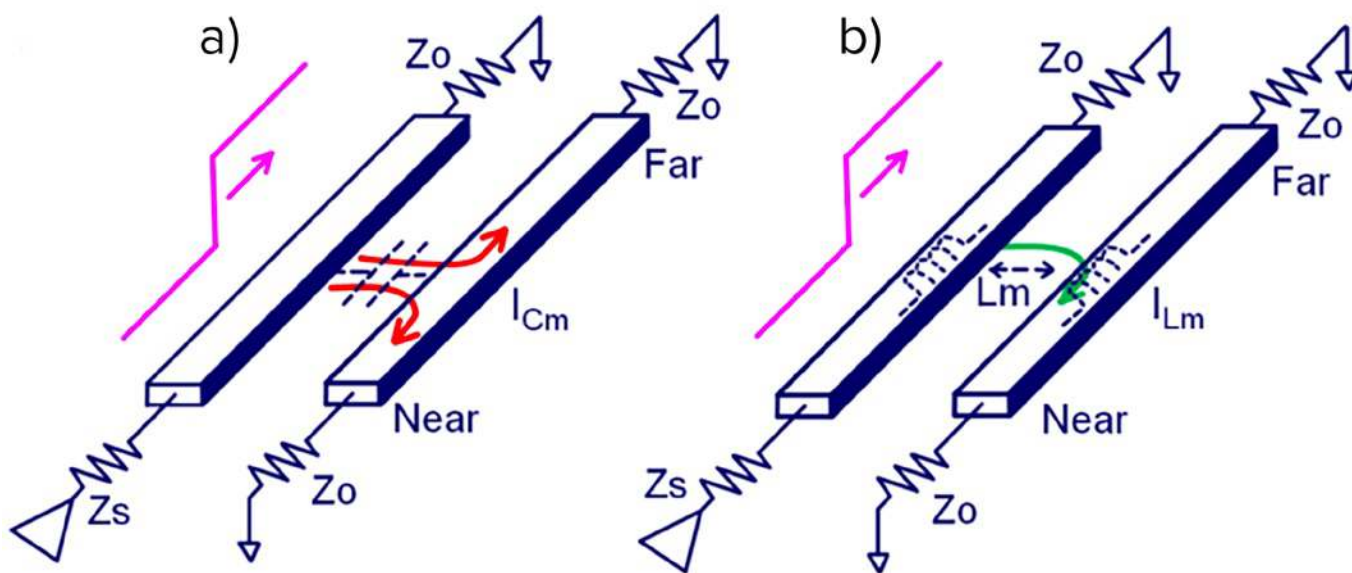


Figure 2: a) Capacitive coupling; b) Inductive coupling. (Source: Intel)

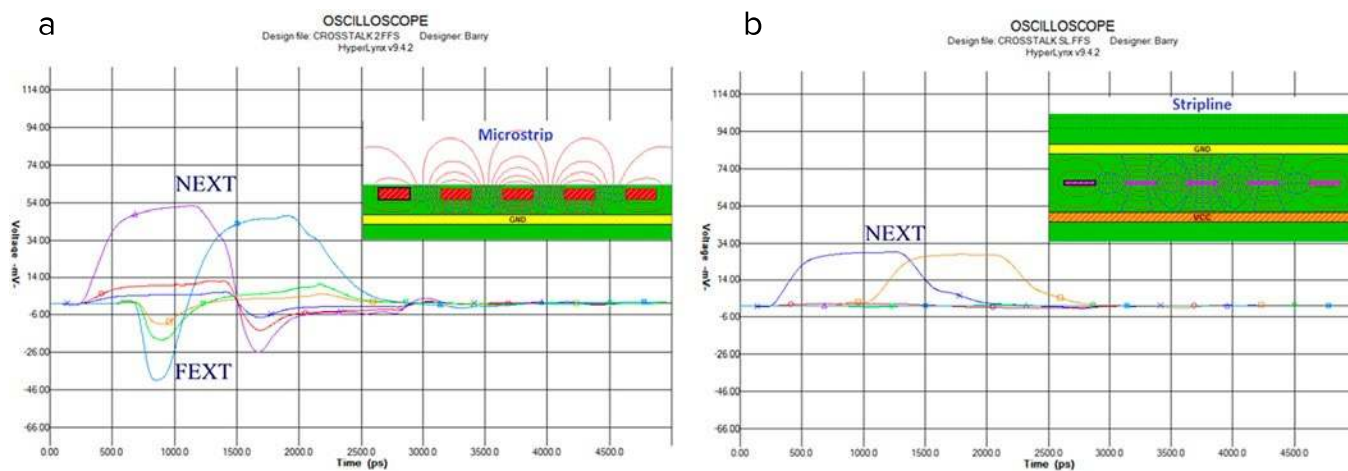


Figure 3: Forward FEXT; and backward NEXT crosstalk for a) microstrip and b) stripline configurations.

ally inductive coupling, driving the FEXT coefficient negative (Figure 3a). However, forward crosstalk does not exist in the stripline configuration. The fine balance between capacitive and inductive coupled crosstalk produces no observable forward crosstalk (Figure 3b). And, the NEXT component is halved. So, all other factors being equal, here is just another good reason why one should always route critical high-speed signals on the inner layers of a multilayer PCB.

Figure 4 illustrates the “backward crosstalk critical length.” This is a measure of how far an aggressor line and a victim line can route in parallel before the amplitude of backward crosstalk (NEXT) reaches its maximum level. Parallelism beyond this length will widen the coupled pulse in the victim line, but will not increase its amplitude. As rise time gets faster, “critical length” becomes shorter. In today’s systems, with extremely fast IC output switching times, almost any parallelism

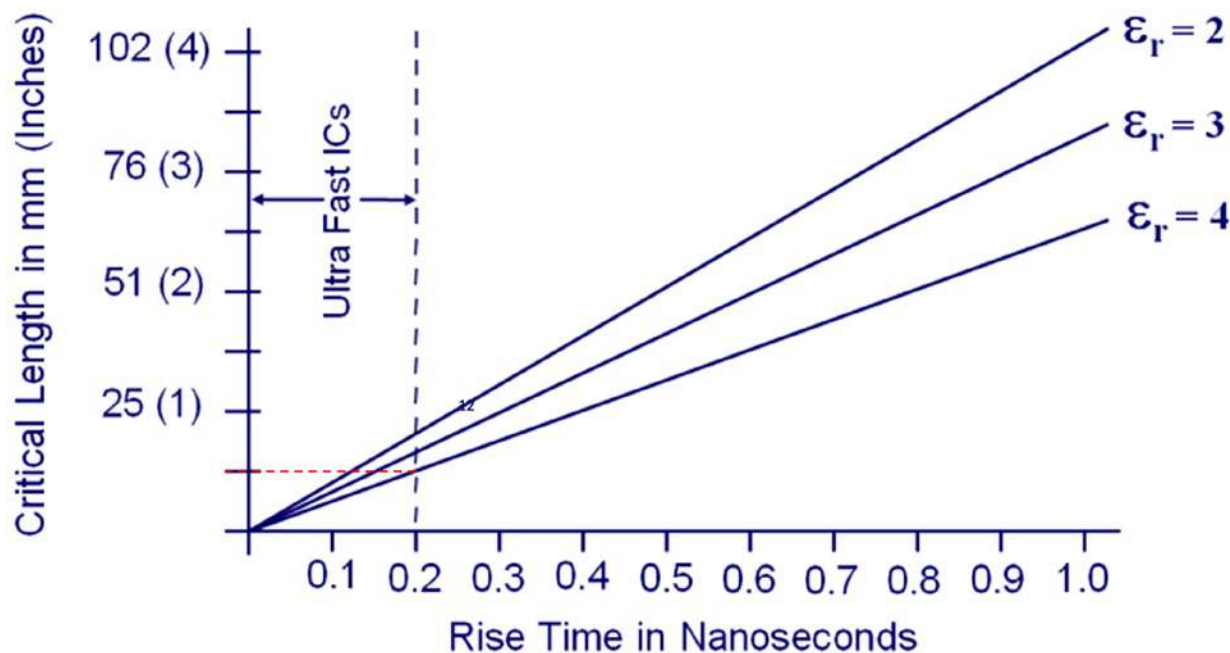


Figure 4: Backward crosstalk critical length.

(impedance of free space) and will often cause more problems than they solve.

To minimize the detrimental interaction of electromagnetic fields, it is good design practice to:

- Separate asynchronous signals, unrelated signals, and signals of different potentials.
- Avoid long parallel trace segments to minimize their coupling.
- Route all critical signals embedded between the planes to reduce crosstalk.
- Keep other signals away from I/O signals and connectors. Never allow anything to route into the I/O area of the board, except I/O lines.
- Use low dielectric constant material and closely couple the signal trace to the reference plane to decrease the radiation.

Key Points

- EM fields do not merge with each other but rather add vectorially—distorting the signal.
- Crosstalk usually comes from electric field-based capacitive coupling of trace segments running in parallel.
- Mutual capacitance between the two lines couples energy in both directions on the victim trace.
- A wide uniform plane as the return path is the configuration for the lowest radiation and crosstalk.
- Mutual inductance between the lines couples energy onto the victim trace segment in the backward direction.
- When multiple fields couple to a single reference line, in a connector or IC package, or into a single ground via, the return currents overlap, creating very high mutual inductance.
- Electromagnetic fields have direction and polarity.

- Forward crosstalk does not exist in the stripline configuration due to the fine balance between the capacitive and inductive coupling.
- For a rise time of 200 ps, backward crosstalk reaches its maximum amplitude in 12 mm of parallelism between lines.
- The amplitude of crosstalk is determined by the distance between the aggressor and victim lines relative to their height above the reference plane. It is extremely important to route all traces on a layer that is one dielectric distance above or below a ground plane.
- A guard trace that is grounded every 1/12th wavelength (bottom) prevents radiation and drastically lowers cross-coupling. **DESIGN007**

Resources

- Beyond Design columns by Barry Olney: “Return Path Discontinuities,” “The Dark Side—Return of the Signal,” “Stackup Configurations to Mitigate Crosstalk.”
 - “Noise, EMI & SI and Single-ended Crosstalk,” by Rick Hartley.
 - “Signal and Power Integrity Simplified,” by Eric Bogatin.



Rick Hartley is the principal engineer at R Hartley Enterprises and has been in the industry for over 50 years. He is one of the primary consultants for PCB manufacturing and design companies. Rick has

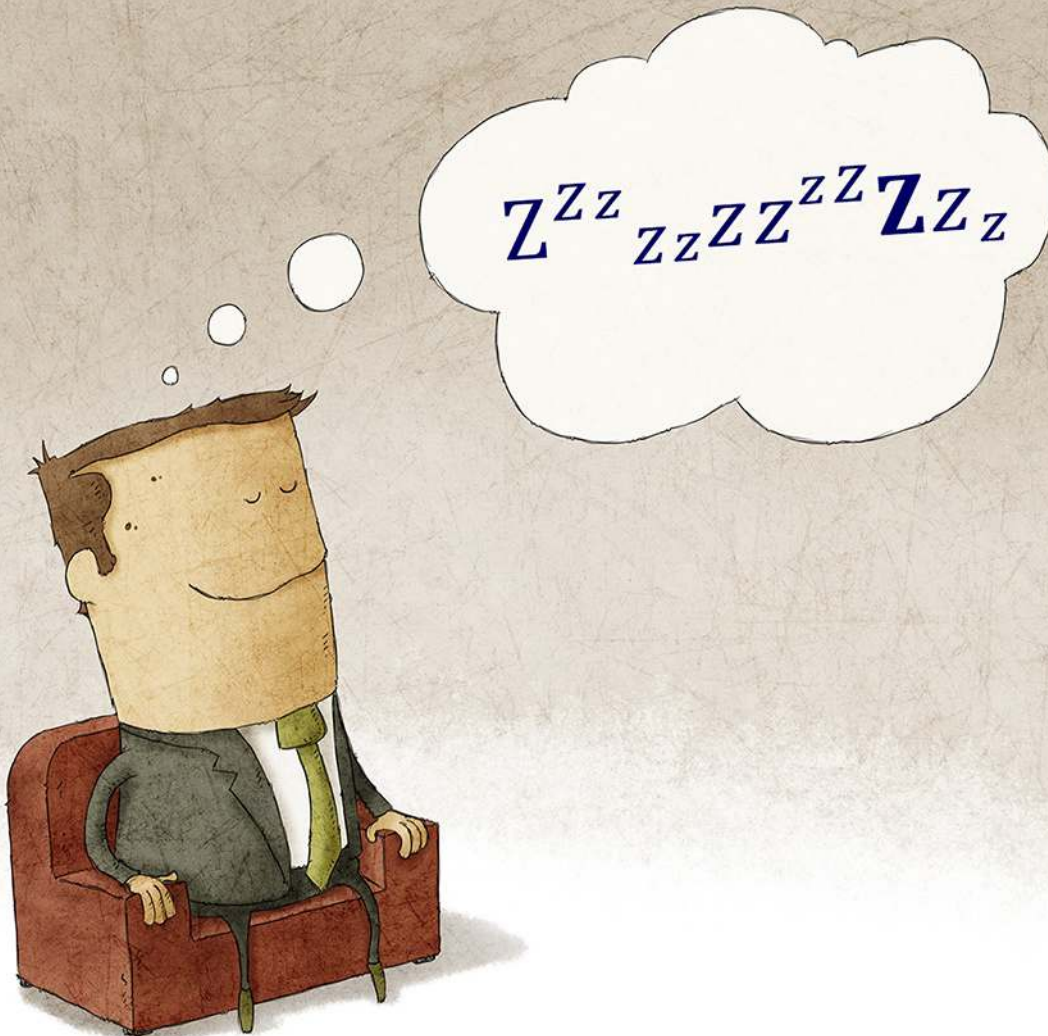
also conducted classes worldwide on EMI, signal integrity, and various other electrical topics for the last 30 years.



Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incor-

porating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns, [click here](#).

We **DREAM** Impedance!



Did you know that two seemingly unrelated concepts are the foundation of a product's performance and reliability?

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

iCD software quickly and accurately analyzes impedance so you can sleep at night.

iCD Design Integrity: Intuitive software for high-speed PCB design.

iCD
In-Circuit Design Pty Ltd

"iCD Design Integrity software features a myriad of functionality specifically developed for PCB designers."

– Barry Olney



www.icd.com.au