Standing Waves in Multilayer PCB Plane Cavities

Beyond Design

Feature Column by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

Plane cavities in multilayer PCBs are essentially unterminated radial transmission lines. They form a transmission line that propagates electromagnetic (EM) energy within a plane cavity emanating from a feed point within the plane and outward in all directions. Like all transmission lines, it will reflect if not terminated. This creates standing waves—ringing. The bigger the mismatch, the bigger the standing waves, and the more the impedance will be location-dependent. As frequency and edge rates continue to increase, the impact of intrinsic electrical characteristics becomes more pronounced. AC switching currents in the power/ground planes can be very large. Under these circumstances, a plane pair acts more like a radial transmission line rather than a distributed planar capacitor.

A parallel plate capacitor (or a planar pair) has two conductors separated by a dielectric layer. Most of the electromagnetic energy in the structure is concentrated directly between the plates. However, some of the energy radiates into the area outside the plates. The elec-

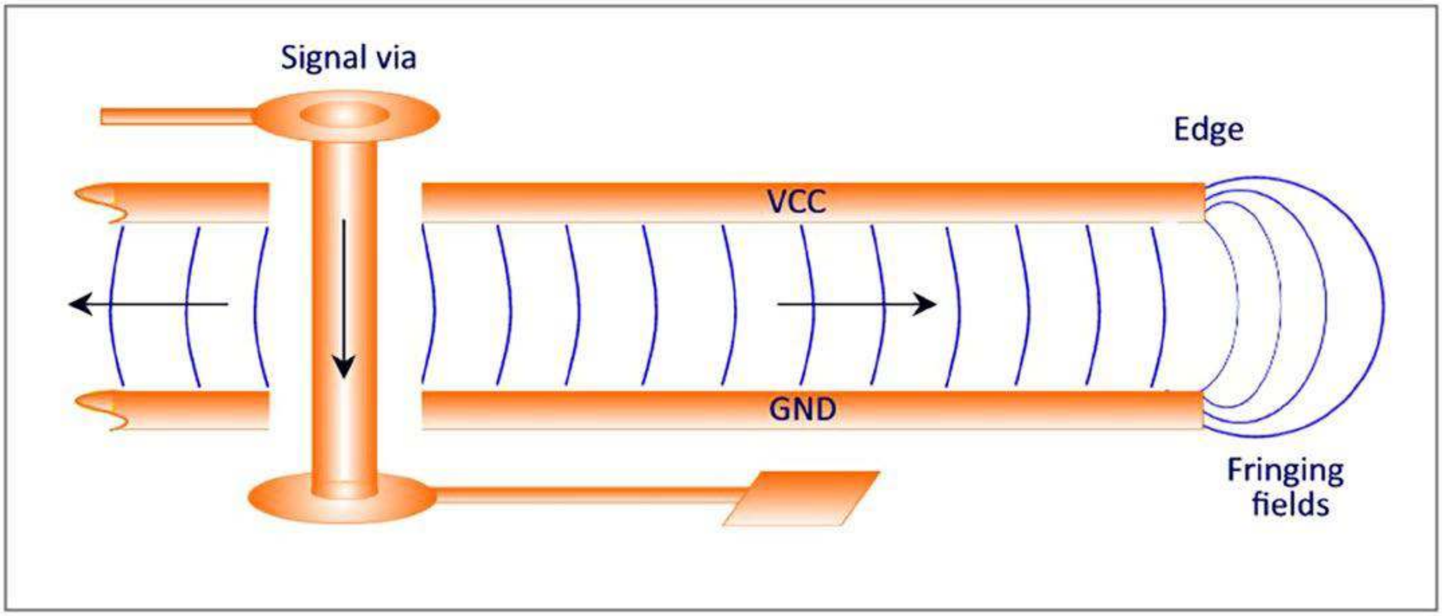


Figure 1: Signals passing through a plane cavity intensify fringing fields.

tric field lines associated with this effect are called fringing fields (Figure 1).

When displacement current flows through the impedance of a cavity between two planes, it generates voltage. Although quite small (typically in the order of 5 mV), the accumulated noise from simultaneous switching devices can become significant. This voltage, emanating from the vicinity of the signal via, injects a propagating wave into the cavity, which can excite the cavity resonances or any other parallel structure (for instance, between copper pours and planes). Other signal vias also passing through this cavity can pick up this transient voltage as crosstalk. When the wave meets the PCB edge, the two reference planes form a slot antenna that will radiate noise with the potential to generate electromagnetic interference (EMI) to nearby equipment.

The more switching signals that pass through the cavity, the more noise is induced into other signals. It impacts vias all over the cavity, not just the ones in proximity to the aggressor signal vias. This cavity noise propagates as standing waves spreading across the entire plane pair. This is the primary mechanism by which high-frequency noise is injected into cavities: by signals transitioning through cavities, using each plane successively as the signal return path.

Cavity resonance also affects the power/signal return layers at the edges of the PCB. Edge effects can be particularly problematic since it is the board edges that are in such proximity to the chassis and, hence, the radiation fields can induce currents into the chassis frame.

When the cavity has open-end boundary conditions, resonances arise when a multiple of half wavelengths can fit between the ends of the cavity. If the clock or data harmonics overlap with the cavity resonant frequencies, there is the potential for long-range coupling between any signals that run through the cavity. This is one reason why all return planes should be GND layers, so that stitching vias between GND planes can be placed adjacent to each signal via transition to minimize the possibility of exciting the cavity resonance. Figure 2 shows a standing wave produced by the superposition of two harmonic waves of equal amplitude, frequency, and wavelength moving in opposite directions.

A region under a large BGA densely populated with vias also appears as a discontinuity due to the large array of anti-pads eating a hole in the plane. A discontinuity reflects propagating energy because it represents a mismatch with the characteristic impedance of the transmission line.

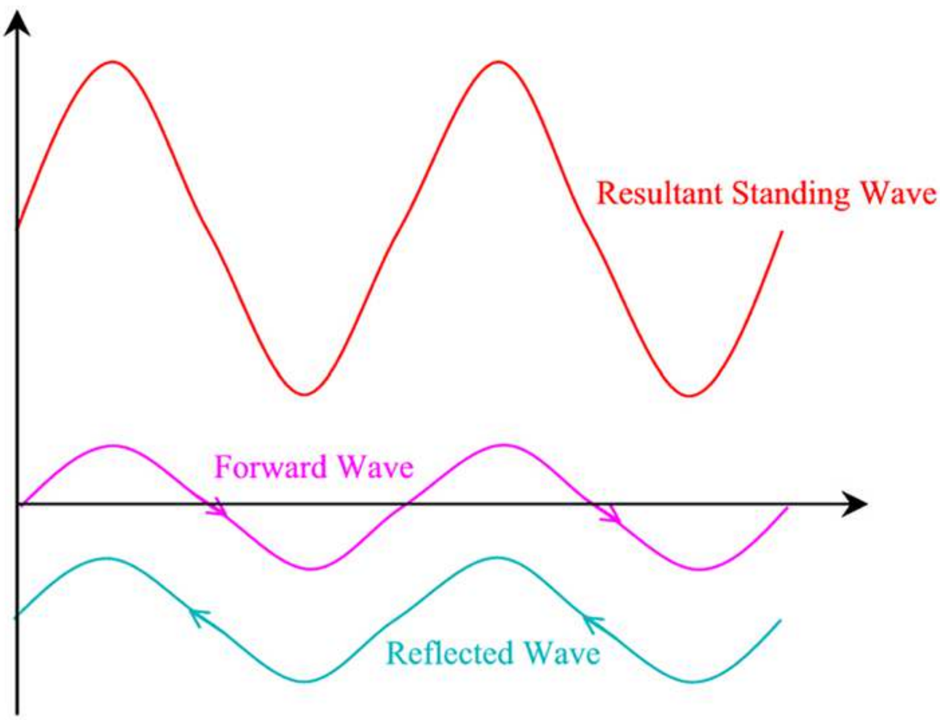


Figure 2: Standing wave is the superposition of two harmonic waves.

The edges of the board cause the greatest amount of reflection since an edge is a totally abrupt open circuit surrounding the board. Reflected energy is accompanied by phase reversals in its components, and combined reflections from the open circuit at the edge of the board can cause a phenomenon known as voltage doubling, creating a standing wave. The standing wave appears to be vibrating vertically without traveling horizontally.

There are various approaches to reducing radiation edge effects from the PCB. In many cases, energy can be reflected, possibly creating additional internal cavity resonance effects and coupling to internal vias, and resulting in increased radiation. When plane pairs resonate, their emissions come from the fringing fields at the board edges. With ground/power plane pairs, edge-fired emissions can be reduced by reducing the plane separation and lowering the AC impedance. Alternatively, make the power planes slightly smaller (~200 mil) than the GND plane. This modifies the pattern of the fringing fields, pulling them back from the edge, and may help reduce emissions to some extent.

Edge plating, as the name suggests, is the process of plating the edges around the PCB. This is an elegant (but expensive) solution to prevent emissions from extremely high-speed SerDes signals on terabit routers, etc., but is an overkill for a typical high-speed design.

Another way to mitigate this problem is to create a via fence, stitched to ground around the perimeter of the PCB. If the spacing between the stitching vias is less than or equal to 1/12th of a wavelength, the via fencing will appear as a short circuit, causing the propagating wave to be reflected back to the source rather than being launched from the PCB edge. Unfortunately, most of the above techniques create reflections and possibly exasperate the issue.

Parallel planes in multilayer PCBs exhibit multiple resonances, which increase the impedance and the EM radiation. A typical FR-4 laminate of 4-mil thickness produces a characteristic impedance of about 3 to 5Ω for adjacent planes. The larger the plane area, the lower the impedance.

The best solution to dampen the plane resonance is to terminate the transmission line with an impedance-matching resistive element along the board edges. But in practice, this means approximating a continuous structure with resistors spaced around the perimeter. Obviously, multiple low-value resistors cannot be placed directly between the power supply and ground as it would needlessly dissipate a huge amount of DC power. To prevent this, they should be AC coupled with a ceramic capacitor of sufficient capacitance to allow the resulting impedance to appear predominately resistive at and above the lowest frequency of

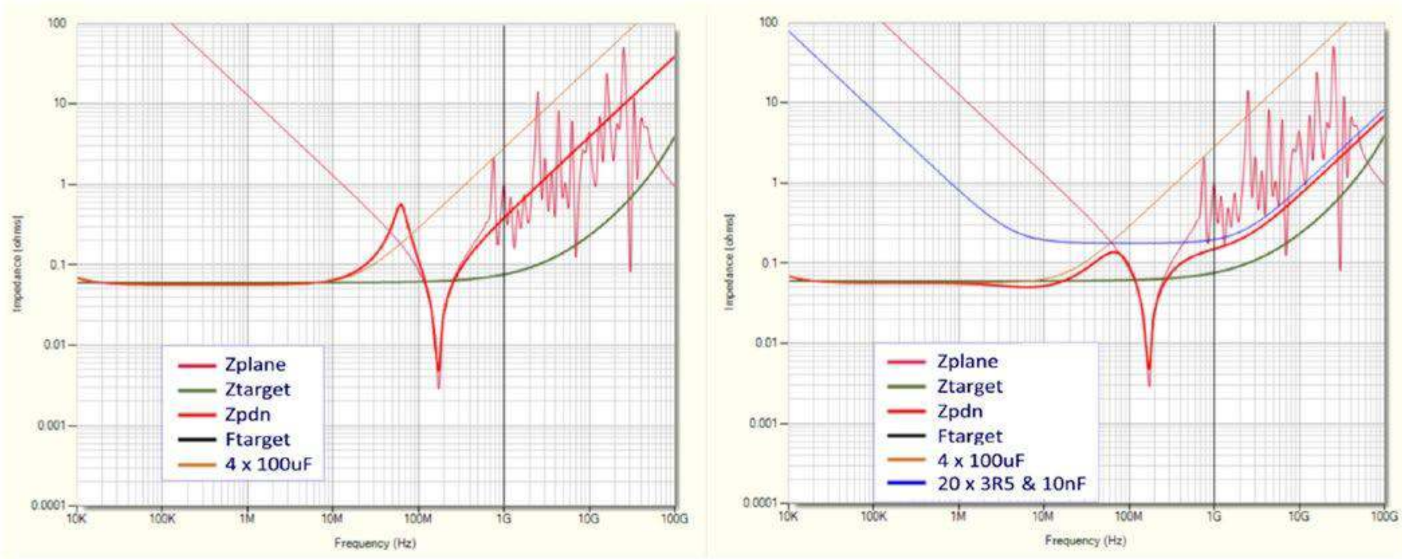


Figure 3: a) Zpdn with 4x100uF; b) Zpdn with added RC damping. (Source: iCD PDN Planner)

interest. A 10nF, X7R ceramic capacitor is a typical value. The addition of loss to dampen modal resonances is more important than the exact termination value and distribution. Notice how the 20-series RC terminations of 3R5Ω and 10nF (blue) dampen the plane resonance, pushing the effective PDN impedance (Zpdn) down below the resonance, above 200MHz (Figure 3b).

A standing wave can be generated at switching locations within the interior of the PCB. Therefore, there may be hot spots within that cavity that would benefit from the placement of an additional AC-coupled load at or near the source.

Reflections occur in a transmission line only when there is a discontinuity. If the edge of the board is terminated in the characteristic impedance of the radial transmission line, then the EM energy will be absorbed and there will be no reflections. This will not eliminate the initial transient but does, however, prevent it from being compounded. Termination elements inhibit standing waves from developing between the planes and protect peripheral signals and electronics from radiation.

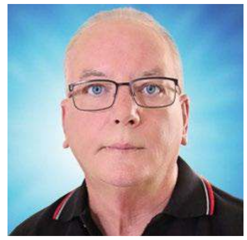
Key Points

- Plane cavities in multilayer PCBs are essentially unterminated radial transmission lines that will reflect if not terminated.
- The energy that radiates from the edges of the plane pair is called a fringing field.
- When displacement current flows through the impedance of a cavity between two planes it generates voltage.
- Signal vias passing through a plane cavity can pick up the transient voltage as crosstalk.
- A standing wave is produced by the superposition of two harmonic waves of equal amplitude, frequency, and wavelength moving in opposite directions.
- Standing waves impact vias all over the cavity, not just the ones in proximity to the aggressor signal vias.
- All return planes should be GND layers, so that stitching vias between GND planes can be placed adjacent to each signal via transition to minimize the possibility of exciting the cavity resonance.
- A region under a large BGA densely populated with vias also appears as a discontinuity due to the large array of anti-pads eating a hole in the plane.
- The edges of the board cause the greatest amount of reflection since an edge is a totally abrupt open circuit surrounding the board.
- The standing wave appears to be vibrating vertically without traveling horizontally.

- Parallel planes in multilayer PCBs exhibit multiple resonances, which increase the impedance and the EM radiation.
- The best solution to dampen the plane resonance is to terminate the transmission line with an impedance-matching resistive element, along the board edges. DESIGN007

References

1. Beyond Design by Barry Olney: "Dampening Plane Resonance with Termination," "Plane Cavity Resonance," "Fringing Fields."
2. Signal and Power Integrity—Simplified, 3rd ed., by Eric Bogatin.
3. "Optimized Power Delivery Performance Using Plane Terminations," by Istvan Novak, Samtec. Published in 2020 IEEE 24th Workshop on Signal and Power Integrity (SPI).
4. US5708400A—AC coupled termination of a printed circuit board power plane in its characteristic impedance, Google Patents, Terrel Morris, Hewlett-Packard.



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 incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns, click here.

Toward a European Chips Act

By Alison James, IPC senior director, European government relations

While the U.S. government has begun to implement its CHIPS and Science Act, the European Union is deliberatively moving to issue its own legislation. Rising geopolitical tensions and the supply chain vulnerabilities exposed during the height of the COVID pandemic, followed by the Russian invasion of Ukraine, accelerated a move in the European Union now happening throughout all global regions: taking stock of strategic assets and vulnerabilities.

Electronics and data, it seems, are the "oil" of the 21st century, and the high strategic importance of both elements focuses efforts to secure supply by building regional bases for high value-added activities and intensifying cooperation with strategic trading partners.

The European Union's proposal for legislation regarding semiconductors was issued in February 2022 against the backdrop of global chip shortages, a global "subsidy race" in the world's main producing regions, and a renewed EU industrial policy aiming to deliver on the bloc's ambitious digital and green transition. It is part of the region's evolving "strategic autonomy" agenda: reducing the continent's vulnerability to supply chain disruptions and geopolitical risks. At this time, the proposed legislation is in the final months of negotiation in the inter-institutional process under which European legislation is formed.

With announcements of confirmed and rumored investments by well-known chip companies, it's clear that a leading intention behind the European Chips Act is to attract high-end semiconductor manufacturing to produce the most advanced chips.

To read the rest of this article, which appeared in the Spring 2023 issue of *IPC Community,* click here.