Dampening Plane Resonance with Termination

Beyond Design

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

Today's high-speed multilayer PCBs have multiple planes. The ground planes are used for shielding and to provide return current continuity. Whereas, closely coupled power/ground plane pairs provide low inductance power to the ICs and reduce the AC impedance and plane resonance of the power distribution network (PDN). However, these parallel-plate waveguides (plane pairs) also form a radial transmission line. That is, they form a transmission line that propagates electromagnetic (EM) energy within a plane cavity emanating from a feed point, within the plane, outward in all directions. And, like all transmission lines, it will reflect if not terminated.

The parallel-plate waveguide gets excited by currents flowing through the power/ground plane cavity which can lead to simultaneous switching noise. The current flowing through vias, connecting signal traces, can also cause a similar excitation at harmonic frequencies. In this month's column, I will look at how to dampen plane resonance, radiating from the fringing fields of the board, with RC termination to match the plane's characteristic impedance.

The first line of defense against power supply transients is filtering, bypassing and decoupling. However, in the critical 100 MHz-1 GHz band, the effectiveness of a typical decoupling capacitor is determined almost entirely by its series inductance. This is the frequency band now being used increasingly by digital logic. These strategies lower the AC impedance but do not suppress PDN transients in a system where a radial transmission line exists between the planes.

A termination that matches the transmission line's characteristic impedance reduces reflections. But, in the case of parallel-plate resonance, termination also dampens the standing wave of propagating EM energy that can be built-up within the cavity and radiate from the edge fringing fields (Figure 1). EM energy can be dissipated or absorbed by terminating the edges of the PCB, in its characteristic impedance.

As frequency and edge rates continue to increase, the impact of intrinsic electrical characteristics become more pronounced. AC switching currents in the power/ground planes can be very large. Under these circumstances,

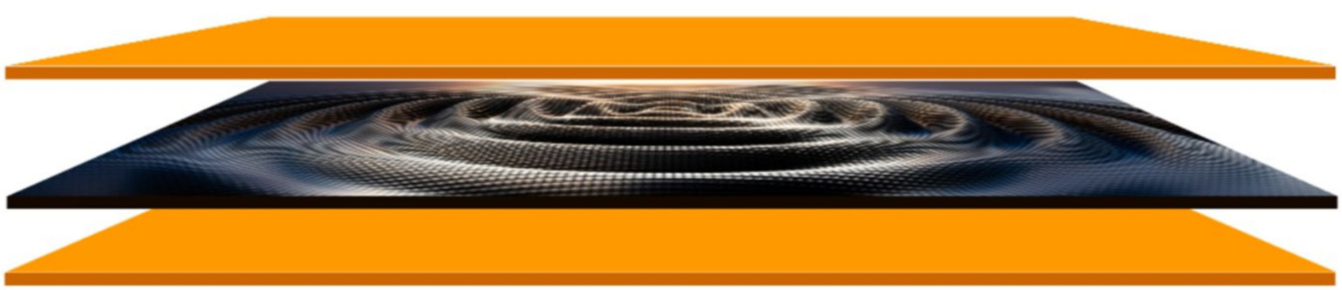

Figure 1: A standing wave built up by plane cavity resonance.

a plane pair acts more like a radial transmission line rather than a planar capacitor.

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. 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.

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. This is a good reason to make planes as large as possible.

How Do We Dampen the Plane Resonance?

Resistive termination along the board edges reduces the resonance peaks. 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 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 RC terminations (blue) dampen the plane resonance pushing the effective Zpdn down below the resonance, above 300MHz in Figure 2 (right).

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 there will be no reflections. This will not

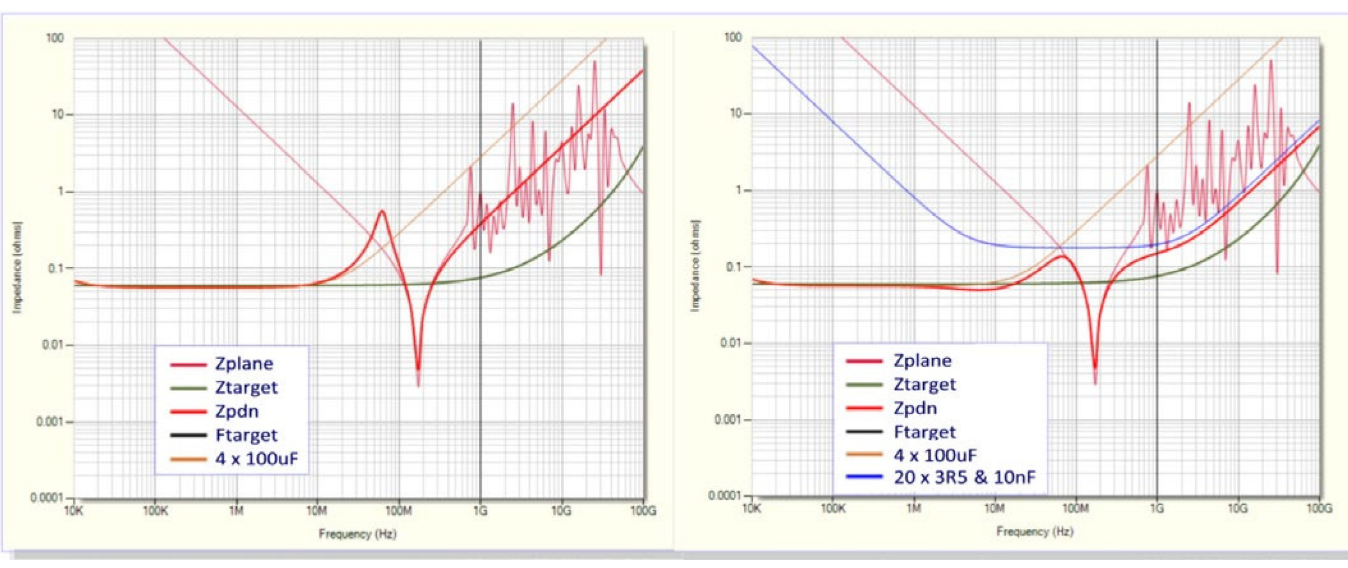

Figure 2: Zpdn with 4x100uF (left), Zpdn with added RC damping (right). (Source: iCD PDN Planner)

eliminate the initial transient but does, however, prevent it from being compounded. Edge effects can be particularly problematic since it is the board edges that are in such close proximity to the chassis and hence the radiating fields can induce currents into the chassis frame. Termination elements inhibit standing waves from developing between the planes and protect peripheral signals and electronics from radiation.

Where Do the Adjacent RC Terminations Need to be Placed?

That highest effective frequency of concern is typically the fifth harmonic of the fundamental frequency, corresponding to the rise time. Assuming that the fastest rise time is one nanosecond, then we are concerned with frequencies up to 5 GHz. The wavelength, in FR-4 at 5 GHz, is about 1.1 inches. So, in this case, AC-coupled terminations spaced one inch apart is a good approximation of a continuous termination. At 2.5 GHz bandwidth (500 MHz clock), two-inch spacing would suffice. The resistor, capacitor and power/ground vias should be placed closely together with wide connecting traces to minimize inductance as in Figure 3. The added losses from the RC terminations are helpful in the overall response of the PDN as they dampen impedance peaks arising from connecting the devices to the board.

The optimization of the PDN is a trial-and-error process that needs to be done in conjunction with the stackup configuration, dielectric materials, decoupling and AC-coupled terminations to fully exploit all avenues. Adding the terminations to the periphery of the board will result in new anti-resonance peaks that need to be dampened. The iCD PDN Planner shows the impact of RC terminations on the cavity resonance, allowing the designer to level out the peaks with a selection of decaps. Suppressing the plane resonance peaks to provide a low impedance profile at higher frequencies helps to minimize electromagnetic emissions.

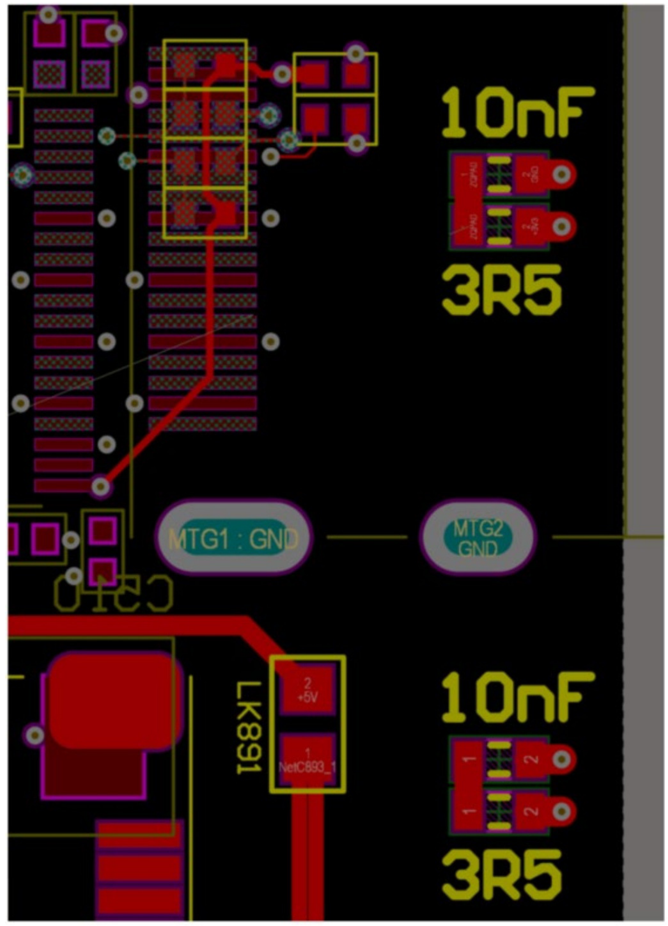

Figure 3: RC terminations (3R5 and 10nF) are spaced around the periphery of the PCB.

Key Points:

- Parallel-plate waveguides (plane pairs) form a radial transmission line that propagates electromagnetic (EM) energy within a plane cavity. The parallel-plate waveguide also gets excited by currents flowing through the power/ground plane cavity.
- Decoupling does not suppress PDN transients in a system where a radial transmission line exists.
- EM energy can be dissipated or absorbed by terminating the edges of the PCB in its characteristic impedance.
- 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.
- 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 larger the plane area, the lower the impedance.
- RC terminations of 3R5, in series with 10nF spaced along the board edges, are generally sufficient to reduce the resonance peaks.
- The addition of loss to dampen modal resonances is more important than the exact termination value and distribution.
- The added losses from the RC terminations are helpful in the overall response of the PDN as they dampen impedance peaks arising from connecting the devices to the board. **DESIGN007**

Resources

1. Beyond Design: PDN—Decoupling Capacitor Placement, Learning the Curve, Fringing Fields, by Barry Olney, January 2016.
2. Optimized Power Delivery Performance Using Plane Terminations by Istvan Novak, Samtec.
3. 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 or contact Olney, click here.