DEMYSTIFYING COMMON-MODE RADIATION

BY BARRY OLNEY, IN-CIRCUIT DESIGN PTY LTD

Common-mode radiation is a major contributor to unwanted electromagnetic interference (EMI). It arises when equal-phase currents flow on conductors without an opposing return current to cancel their fields. The resulting imbalance causes those conductors, especially attached cables, to behave as unintended antennas. Grasping how common-mode radiation is generated, the problems it creates, and the methods available to control it is essential for designing reliable electronic systems that meet regulatory requirements.

Electromagnetic radiation from digital circuits generally appears in either differential mode or common mode operations. In differential mode operation, currents flow in equal and opposite directions, and (at least in theory) their electromagnetic fields cancel, keeping radiation to a minimum. On the other hand, common mode currents are a very different beast. When two coupled conductors carry identical currents, the fields no longer oppose each other; they reinforce, thus turning cables, traces, and even chassis structures into surprisingly efficient antennas.

The real trouble begins when perfectly good differential mode signals are inadvertently converted into common mode currents. This mode conversion can arise from parasitic capacitance, geometric imbalances between traces, timing skew, inadequate return paths, or asymmetries anywhere in the channel. Even small discontinuities in the return path can create large common mode loop areas, increasing series inductance and dramatically boosting radiated emissions.

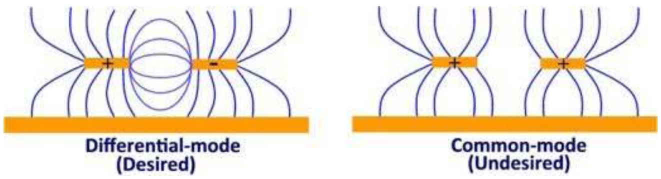


Figure 1: Differential and common-mode fields.

If I_1 and I_2 are the currents in the two conductors, then:

Differential Mode Current (I_{DM}) = $(I_1 - I_2)/2$

Common Mode Current (I_{CM}) = $(I_1 + I_2)/2$

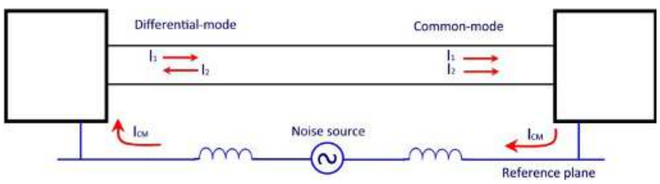


Figure 2: Differential and common-mode currents.

Differential-mode radiation is an inherent part of normal circuit operation, arising from currents flowing through the return-path loop formed by PCB conductors: both traces and their associated return paths (Figure 2). Microstrip loops on outer layers can behave like small antennas, primarily radiating magnetic fields, while stripline loops on inner layers radiate only through the fringing fields at the board edges. Although these signal loops are essential for proper functionality, their physical size and loop area must be carefully controlled during design to keep radiated emissions to a minimum.

The most critical radiation-producing loops are those carrying the high-frequency periodic signals. In synchronous systems, the clock, being a continuous stream of repetitive pulses, is typically the dominant emitter. Clock routing should therefore be prioritized, with every effort made to minimize loop area. This includes keeping the clock trace as short as possible and reducing the number of layer-transitions.

On multilayer PCBs, clock lines are best routed as striplines on inner layers adjacent to a solid reference plane, which helps contain fields and reduce radiation. Tight spacing between the clock trace and its return plane further increases coupling and shrinks the loop area. Additionally, to avoid coupling clock energy into external cables, clock circuitry should be placed well away from I/O connectors and cable interfaces.

Data and address buses, along with their command and control lines, are the next major contributors to emissions. Even with proper termination, these interconnects can carry significant bursts of electromagnetic energy, and the resulting radiation scales with the magnitude of that energy. Transient power-supply activity is another key source of differential-mode emissions. Although the associated loops may be physically small, they can concentrate intense switching electromagnetic energy, making them disproportionately strong radiators.

Differential-mode radiation increases with the square of frequency, and it can be managed through several design strategies: lowering the power distribution network (PDN) impedance below the target value, minimizing loop area, using differential signaling to cancel fields, and applying clock dithering. Spreading the emission energy across a wider frequency band also reduces peak radiation levels. In practice, spread-spectrum clocking can cut radiated emissions by as much as 15 dB.

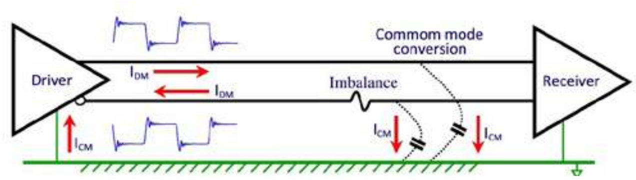


Figure 3: Differential mode signals can be converted to common mode by displacement current.

When a differential pair is well balanced, tight coupling helps achieve strong field cancellation. But if the pair is even slightly imbalanced (Figure 3), the effectiveness of cancellation is no longer governed by trace spacing. It is dictated by the pair's common-mode balance. Any imbalance in the routing must be corrected at the point where it occurs to preserve field cancellation.

Also, due to the inherently poor common-mode balance of most digital drivers, differential pairs frequently emit much stronger common-mode radiation than differential-mode radiation. In these cases, reducing the spacing between the traces provides little to no benefit in lowering radiation. Figure 4 illustrates the simulated common-mode return current in the reference plane beneath an imbalanced differential pair. Note that the return path of the non-inverting and inverting differential signal is not in the opposite pair, but rather the return path for both signals is in the reference plane.

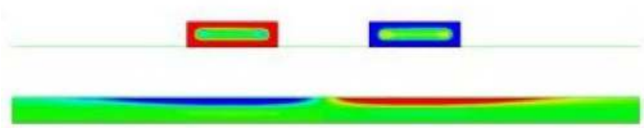


Figure 4: Coupled microstrip differential pair. (Source: Ansoft)

Closely coupled differential signals operate primarily in differential mode, with only minor common-mode radiation caused by any imbalance between the traces. However, if the traces are spaced far enough apart that coupling is lost, and they behave as two independent single-ended lines. In that case, a 100-ohm differential pair effectively becomes two 50 ohm single-ended signals, which is perfectly acceptable as long as the loop area remains small and the impedance stays consistent along the entire length of the routing.

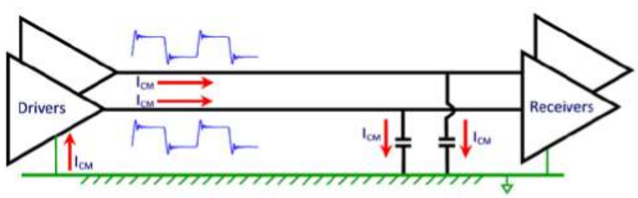


Figure 5: Common mode signal return path.

To make matters worse, once cables are connected to the PCB, they inherit this common-mode ground potential and effectively become monopole antennas. Remarkably, as little as 3 µA of common-mode current on a 1 m cable at 100 MHz is enough to fail an FCC Class B EMC test. While differential-mode radiation can be managed through careful stackup design and routing, common-mode radiation is far harder to predict and control because it is unintentionally engineered into the system. Unfortunately, schematics do not reveal the often-unexpected current return paths that are critical to understanding signal integrity, crosstalk, and radiated emissions.

Unrelated power planes should never extend into the ground region of the I/O connector area. These planes often carry high-frequency switching noise, and if they encroach on the I/O zone, that noise can readily couple into the I/O signals and their reference ground. To prevent this, the I/O ground must connect to the enclosure or chassis ground at a single, well-defined, low-impedance point. This minimizes noise injection, controls return-current paths, and maintains signal integrity at the interface boundary.

To effectively control common-mode radiation, the priority is to reduce the common-mode ground voltage that drives unintended antennas at the source. PDN noise is a dominant contributor to radiated emissions. Suppressing this noise requires preventing it from propagating out of the processor and into the power and ground planes, while designing a PDN whose AC impedance stays below the target impedance across the full bandwidth. Achieving a low-impedance PDN involves minimizing the spacing between power and ground planes, minimizing the loop area and using low-inductance, low-impedance decoupling capacitors. Plane cavity resonance can also generate standing waves within the cavity, which in turn amplify the overall resonance. Because these interactions are complex and highly frequency-dependent, using a PDN planning tool is strongly recommended.

Good grounding also helps suppress noise by giving common-mode currents a low-impedance path back to ground. Incorporating multiple ground planes in the stackup is one of the most effective ways to achieve this. Just as important is maintaining those planes as solid, uninterrupted surfaces—slots or splits can severely disrupt return paths and should be avoided.

When the return path for a common-mode current is physically separated from the signal path, the resulting large loop area leads to radiation. But if the return path is engineered to stay close to the source current, the loop area remains small, and radiation is minimized. In other words, microstrip traces don't inherently radiate; only the ones with poorly designed return paths do. The most prevalent form of common-mode radiation emanates from system cables, which readily behave like monopole antennas.

To limit the resulting common-mode current, it's important to:

- **Limit the driver:** Reduce the driver current strength.
- **Tighten the return path:** Keep the signal and return (reference plane) in very close proximity. Avoid plane splits, slots, or layer transitions that force long, wandering return paths.
- **Improve PDN and decoupling:** Design a low-impedance PDN (tight power-ground spacing, proper decoupling). Use low-inductance mounting and reduce the loop area of the decoupling capacitors.
- **Adopt an effective grounding strategy:** Use solid, continuous ground planes. Employ a single, low-impedance connection between I/O ground and chassis/enclosure ground. Avoid ground islands and poorly thought-out star-routed grounds in high-speed sections.
- **Tame edge rates and spectra:** Use slew-rate control where possible to slow unnecessarily fast edges. Consider spread-spectrum clocking to reduce peak energy at discrete frequencies.
- **Optimize differential pairs:** Maintain good balance with matched impedance, aligned delays, and symmetric routing. Avoid unnecessary asymmetry (stubs, single trace vias, or inconsistent reference conditions).
- **Treat cables as antennas:** Minimize common-mode voltage at connectors. Use common-mode chokes on I/O and data lines leaving the PCB. Add ferrites selected by impedance vs frequency. When possible, isolate using transformers or optocouplers. Keep noisy circuitry away from I/O connectors and the cable entry/exit points.
- **Apply shielding strategically:** Shield cables properly to the chassis (low-impedance connection). Provide clear return paths to shunt common-mode current off the cable/shield and into the chassis. The shield should be bonded at 360°, not by pigtails.

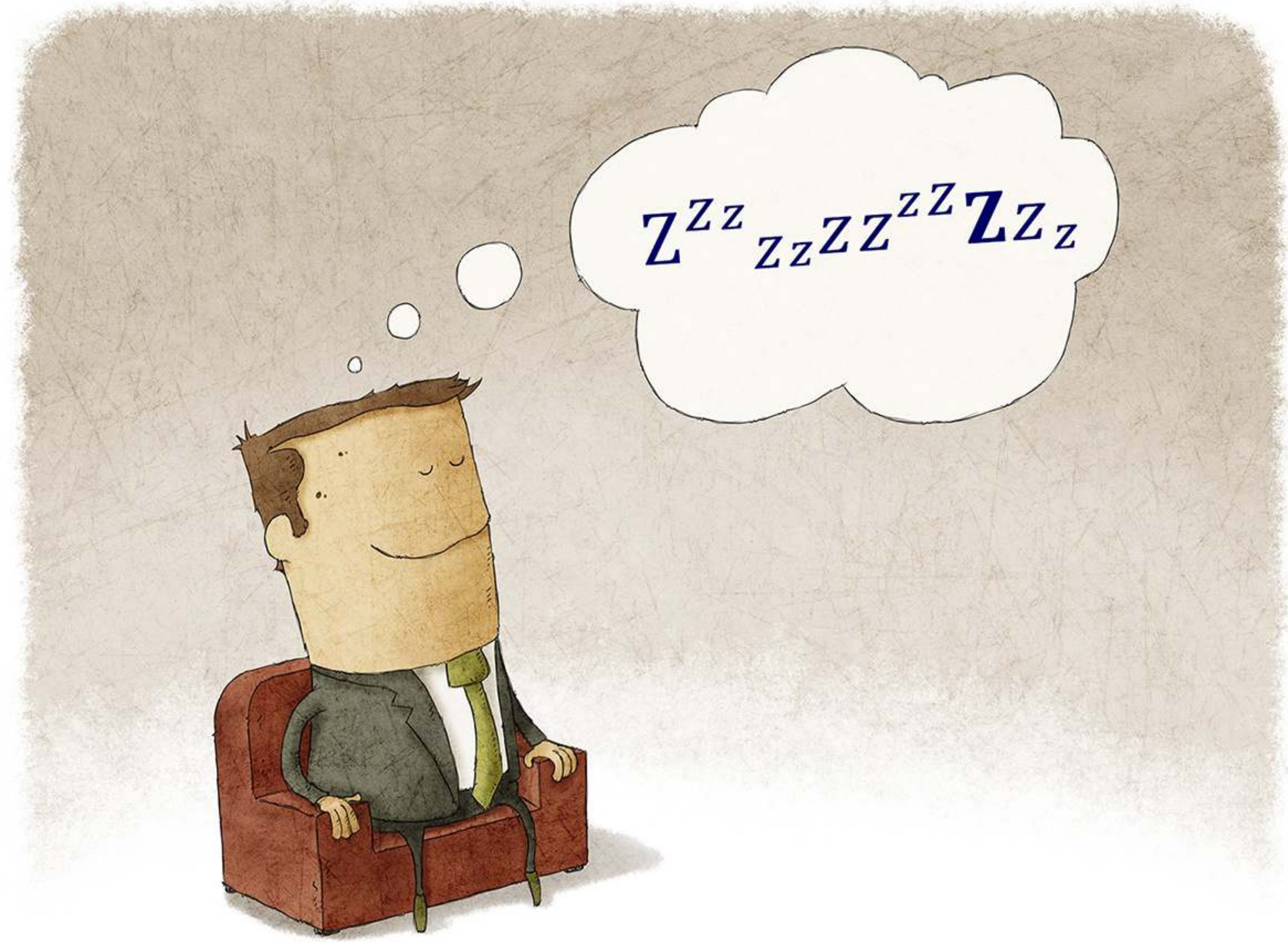
In any high-speed switching system, the active circuitry is the fundamental generator of electromagnetic interference: Its rapid edge transitions, return current discontinuities, and parasitic interactions create the spectral content that drives emissions. Yet it is the attached cabling that becomes the dominant radiator: an unintended, highly efficient antenna structure that couples to the common-mode currents and launches them into free space with surprising effectiveness. The circuit produces the noise energy; the cable provides the propagation mechanism, effectively transforming localized conducted noise into radiated energy. Like all unwanted noise, the only reliable strategy is to suppress it at its point of origin. **I-CONNECT007**

Resources

- Beyond Design: "Return Path Discontinuities," "Uncommon Sense-Differential Pairs," "Stackup Planning Parts 1–4," "Common Symptoms of Common Mode Radiation," all by Barry Olney
- "Electromagnetic Compatibility Engineering," by Henry Ott
- "Understanding Common mode noise," Pulse Electronics
- "High Speed Signal Propagation," by Howard Johnson

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**.

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 Design Integrity software features a myriad of functionality specifically developed for PCB designers."
– Barry Olney

iCD
In-Circuit Design Pty Ltd

www.icd.com.au