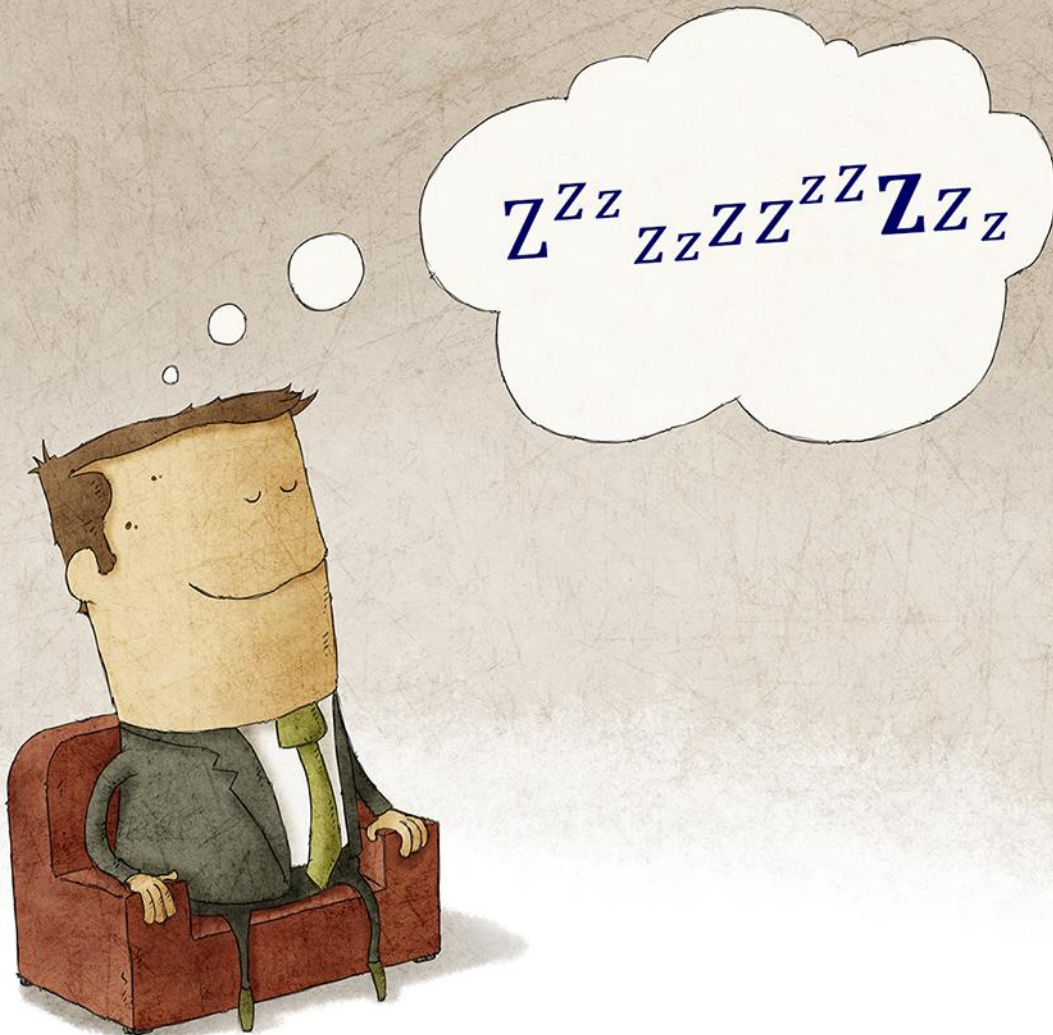


# We DREAM Impedance!



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

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

**DISCOVER MORE**

**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



# Common Symptoms of Common-Mode Radiation

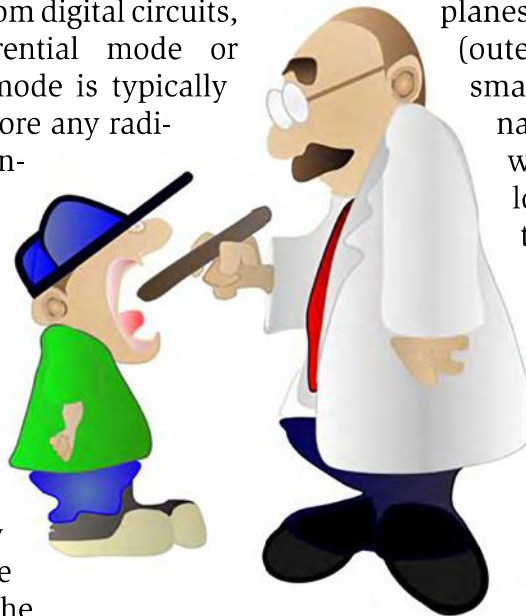
## Beyond Design

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

Electromagnetic radiation from digital circuits, can occur as either differential mode or common mode. Differential mode is typically equal and opposite and therefore any radiating fields will cancel. Conversely, common-mode radiation from two coupled conductors is identical. It does not cancel but rather reinforces.

Unfortunately, differential-mode propagation can be converted to common mode by parasitic capacitance or any imbalance caused by signal skew, rise/fall time mismatch or asymmetry in the channel. Also, return path discontinuities can create large common mode loop areas that increase series inductance and electromagnetic radiation. In this month's column, I will explore the common symptoms of, and present some cures for, common mode radiation.

Differential mode radiation accompanies normal circuit operation and is the result of current flowing in the return path loop formed by the PCB conductors (traces and reference



planes, as in Figure 1). Microstrip (outer layer) loops can act as small antennae that predominantly radiate magnetic fields, whereas stripline (inner layer) loops only emit radiation from the fringing fields at the edge of the PCB. Although these signal loops are necessary for circuit operation, their size and loop area must be controlled during the design process to minimize radiation.

Fortunately, it is not necessary to evaluate each loop individually. However, the most critical loops should be analyzed. The other loops can be controlled by good stackup design practices. Please refer to the structural guidelines in my *Beyond Design: Stackup Planning Parts 1-4* columns <sup>[1]</sup> for further information.

Generally, the most critical loops are the highest frequency where the signal is periodic. In a synchronous circuit, the clock is a sequence of repetitive pulses that generates the most emis-

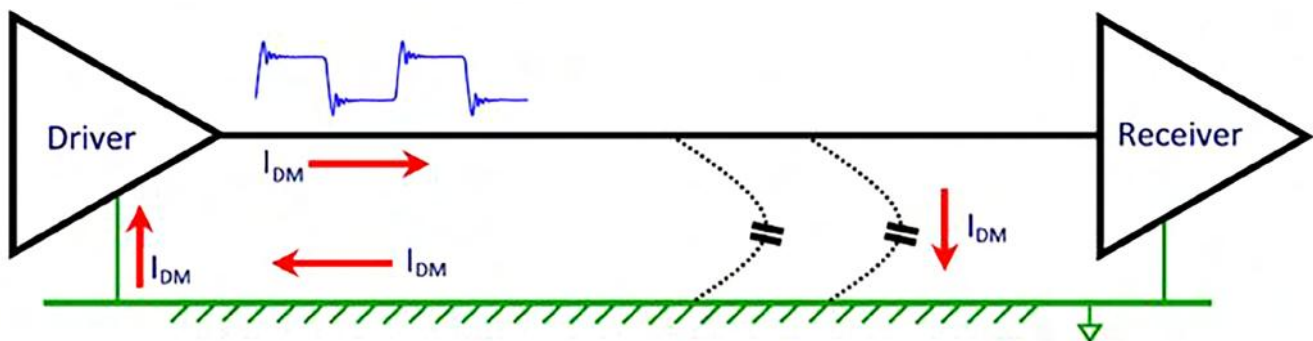


Figure 1: Normal (differential mode) signal return current.

sions. Clock signals should always be routed first, and every effort should be made to route them in the absolute minimum loop area possible. The length of the clock trace should be minimized as well as the number of layer transition vias. On a multilayer PCB, clocks should be routed on a stripline (inner layer) adjacent to a solid reference plane to reduce radiation. The spacing between the clock trace and the return plane should be as small as possible to increase coupling and reduce loop area. Also, to prevent the clocks from coupling to cables, that leave the PCB assembly, the clock circuitry should be located well away from I/O connectors and cables.

Data and address buses, together with their associated command and control signals, are second on the critical list. These buses are normally terminated and can carry large peak currents that radiate proportionally to the current transferred. Transient power supply currents can be another significant source of differential radiation. Although these loops can be quite small, they can carry large currents during switching.

Differential mode radiation is proportional to the frequency squared and can be controlled

by reducing the power distribution network (PDN) impedance to below the required target impedance, minimizing the loop area, canceling out the fields by using differential signals and by dithering the clocks. If the amplitude of the emission is spread out, in the frequency band, then the radiation is also reduced. Spread spectrum clocking can reduce the radiation by up to 15dB.

If a differential pair is well balanced, then tight coupling will achieve an effective degree of field cancellation. However, if they are not perfectly balanced (Figure 2), then the degree of cancellation is not determined by the spacing, but rather by the common mode balance of the differential pair. Most digital drivers have poor common mode balance and therefore differential pairs often radiate far more power in the common mode than in the differential mode. In such a case, one gains no radiation benefit from coupling the differential traces more closely together. Figure 3 highlights the simulated common-mode return current in the reference plane of the cross-section of an imbalanced differential pair.

Differential signals that are closely coupled will operate mainly in the differential mode

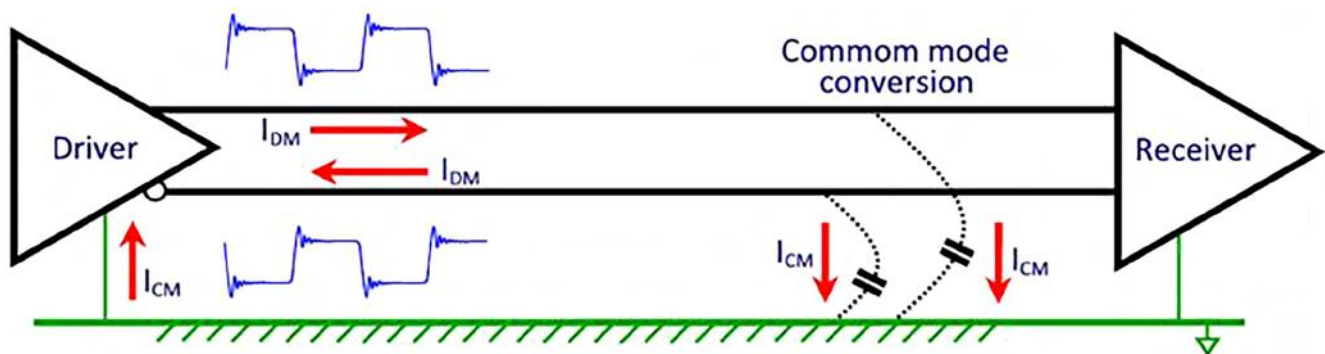


Figure 2: Differential-mode signals can be converted to common mode.

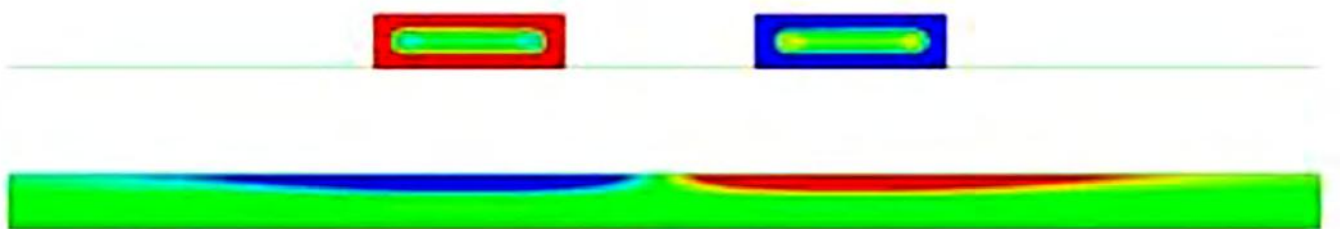


Figure 3: Coupled microstrip differential pair (courtesy Ansoft).

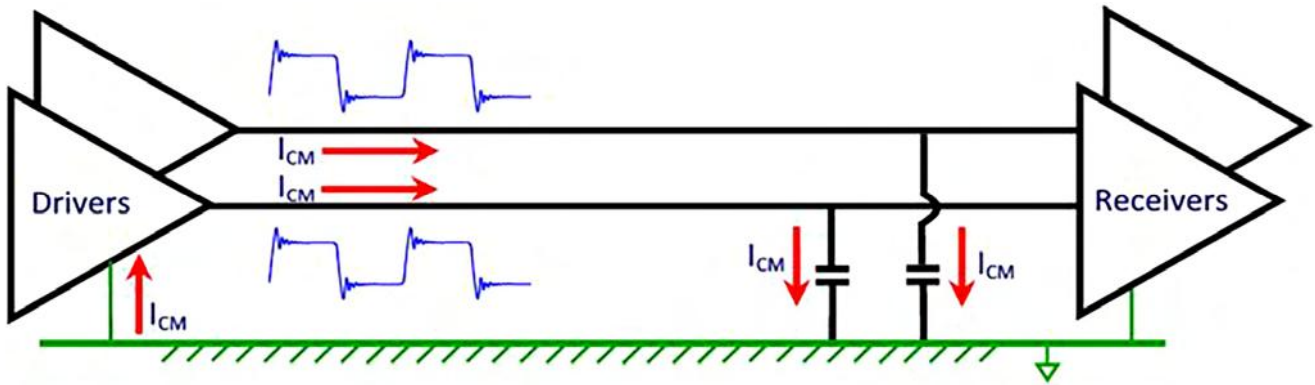


Figure 4: Common-mode signal return path.

with some common mode radiation from imbalances in the signals. If the two traces are separated enough to prevent coupling, then both act as single-ended signals. So a 100-ohm differential pair becomes two individual 50-ohm single-ended signals. This is fine, providing the loop area is small and the impedance does not change along the length of the signals.

In contrast, the “bad guy,” common-mode radiation, is the result of parasitics in the circuit which emanate from the unwanted voltage drops in the conductors. As the signal is driven down the transmission line, capacitive coupling between the trace and plane conductors completes the loop and displacement current flows through the capacitance which returns to the source (Figure 4). The common-mode current that flows through the ground impedance produces a voltage drop in the digital logic ground system and generates magnetic radiation.

To make things worse, when cables are connected to the PCB, they are driven by this common mode ground potential forming antennae. In fact, it only takes 3uA of common mode current, on a 1m long cable at 100MHz, to fail an FCC Class B EMC test. Differential-mode radiation can be controlled by stackup design and routing however, common mode radiation can be difficult to understand and control because it is unintentionally designed into the system. The schematics do not show the sometimes radical, current paths taken that are vital to our understanding of signal performance, crosstalk and electromagnetic emissions.

The most prevalent form of common mode radiation emanates from cables, of the system,

that act like dipole or monopole antennae. So, it is important to limit the common mode current by:

1. Reducing the magnitude of the current and source voltage.
2. Reducing the rise-time, frequency and harmonic content of the current.
3. Reducing the antennae length.
4. Provide a common mode impedance choke in series with the cable and isolate the cable from the PCB with a transformer or optical coupler if possible.
5. Shield the cable and shunt the current off the cable.

The PCB power planes should not be allowed to extend into the ground area of the I/O connectors. This is because the power plane will usually contain high frequency switching noise and if extended into the I/O area, can couple the noise to the I/O signals and ground. The key here is to have a very low impedance connection, at one point only, between the I/O ground and the enclosure/chassis ground.

Also, on the PCB, microstrip stubs can act as antennae once their length approaches one quarter wavelength. These short stubs become very efficient transmitters and so are best avoided. In multi-drop buses as with DDR3/4 fly-by architecture, stubs can be avoided by routing the signals directly through each memory device pad in succession.

To control common-mode radiation, it is important to minimize the common mode ground voltage that drives the antennae at

the source. PDN noise is the main driver of radiated emissions. Minimizing PDN noise is accomplished by blocking the escape of noise from the processor into the power and ground planes, and by designing a PDN that has AC impedance below the required target impedance up to the maximum bandwidth. The goal of a low impedance PDN is realized by minimizing the spacing between the power and ground planes and by using low impedance decoupling capacitors with low inductance mounting to the power/ground planes. The use of a PDN Planning tool is recommended to effectively analyze the complex requirements.

Also, good grounding minimizes noise sources by presenting common mode currents with a low impedance path to ground potential. The use of multiple ground planes, in the stackup, is a very effective way to do this. The importance of avoiding slots or splits, in the solid planes, cannot be overemphasized.

---

## The importance of avoiding slots or splits, in the solid planes, cannot be overemphasized.

---

If the return path of a common mode current is far from the signal path, then the common mode current will radiate. However, if you engineer the return path to be in proximity to the source current, then the loop area will be small and therefore the common mode current will not radiate. So not all microstrip traces will radiate—just the poorly designed paths.

### Key Points:

- Differential-mode radiation is equal and opposite and therefore any radiating fields will cancel.
- Common mode radiation, from two coupled conductors, is identical. It does not cancel but rather reinforces.

- Microstrip loops can act as small antennae that predominately radiate magnetic fields.
- Stripline loops only emit radiation from the fringing fields at the edge of the PCB.
- The most critical loops are the high-frequency clocks where the signal is periodic.
- Clocks should be routed on a stripline adjacent to a solid reference plane to reduce radiation.
- The spacing between the clock trace and the return plane should be as small as possible to increase coupling and reduce loop area.
- Differential mode radiation can be controlled by reducing the PDN impedance, minimizing the loop area, cancelling out the fields by using differential signals and by dithering the clocks.
- If a differential pair is not perfectly balanced, then the degree of cancellation is not determined by the spacing, but rather by the common mode balance of the differential pair.
- Differential signals, that are closely coupled, will operate mainly in the differential mode with some common mode radiation from imbalances in the signals.
- If the two differential signals are separated enough, to prevent coupling, then both act as single ended signals.
- Common mode radiation is the result of parasitics in the circuit which emanate from the undesired voltage drops in the conductors.
- When cables are connected to the PCB, they are driven by the common mode ground potential forming antennae.
- Power planes should not be allowed to extend into the ground area of the I/O connectors as noise can be coupled into the I/O signals and ground.
- Microstrip stubs can act as antennae once their length approaches one quarter wavelength.
- Stubs can be avoided by routing the signals directly through each memory device pad in succession.
- To control common mode radiation, it is important to minimize the common mode

ground voltage that drives the antennae at the source. PDN noise is the main driver of radiated emissions.

- The goal of a low impedance PDN is realized by minimizing the spacing between the power and ground planes and by using low impedance decoupling capacitors with low inductance mounting. **DESIGN007**

## References

1. Barry Olney's Beyond Design columns: [Return Path Discontinuities](#), [Uncommon Sense-Differential Pairs](#), [Stackup Planning Parts 1-4](#).
2. [Electromagnetic Compatibility Engineering](#), by Henry Ott.
3. [What is Differential and Common-Mode Current?](#) by Ron Brewer.

4. [Understanding Common-Mode Noise](#), by Pulse Electronics.

5. [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 from [www.icd.com.au](http://www.icd.com.au). To contact Olney, or read past columns, [click here](#).

---

# Researchers Develop Handheld 3D Skin Printer

U of T Engineering researchers have developed a handheld 3D skin printer that deposits even layers of skin tissues to cover and heal deep wounds. The team believes it to be the first device that forms tissue in situ, depositing and setting in place in two minutes or less.

Their research, led by Navid Hakimi (MIE PhD candidate) under the supervision of Professor Axel Guenther (MIE, IBBME), and in collaboration with Dr. Marc Jeschke, director of the Ross-Tilley Burn Centre at Sunnybrook Hospital, was recently published in the *Journal Lab on a Chip*.

For patients with deep skin wounds, all three skin layers—the epidermis, dermis and hypodermis—may be heav-

ily damaged. The current preferred treatment is called split-thickness skin grafting, where healthy donor skin is grafted into the surface epidermis and part of the underlying dermis.

Split-thickness grafting on large wounds requires enough healthy donor skin to traverse all three layers, and sufficient graft skin is rarely available. This leaves a portion of the wounded area ungrafted or uncovered, leading to poor healing outcomes.

Although a large number of tissue-engineered skin substitutes exist, they are not yet widely used in clinical settings.

The handheld skin printer resembles a white-out tape dispenser – except the tape roll is replaced by a microdevice that forms tissue sheets. Vertical stripes of “bio ink,” made up of protein-based biomaterials including collagen, the most abundant protein in the dermis, and fibrin, a protein involved in wound healing, run along the inside of each tissue sheet. The handheld device is the size of a small shoe box and weighs less than a kilogram.

The researchers hope that one day they can begin running clinical trials on humans, and eventually revolutionize burn care.

“Several steps are needed, but we are confident we will get there,” says Guenther.

