



Missouri University of Science and Technology
Scholars' Mine

Electrical and Computer Engineering Faculty
Research & Creative Works

Electrical and Computer Engineering

01 Jan 2003

Printed Circuit Board EMI Source Mechanisms

Todd H. Hubing
Missouri University of Science and Technology

Follow this and additional works at: https://scholarsmine.mst.edu/ele_comeng_facwork

 Part of the [Electrical and Computer Engineering Commons](#)

Recommended Citation

T. H. Hubing, "Printed Circuit Board EMI Source Mechanisms," *Proceedings of the IEEE International Symposium on Electromagnetic Compatibility, 2003*, Institute of Electrical and Electronics Engineers (IEEE), Jan 2003.

The definitive version is available at <https://doi.org/10.1109/ISEMC.2003.1236553>

This Article - Conference proceedings is brought to you for free and open access by Scholars' Mine. It has been accepted for inclusion in Electrical and Computer Engineering Faculty Research & Creative Works by an authorized administrator of Scholars' Mine. This work is protected by U. S. Copyright Law. Unauthorized use including reproduction for redistribution requires the permission of the copyright holder. For more information, please contact scholarsmine@mst.edu.

Printed Circuit Board EMI Source Mechanisms

Todd H. Hubing

University of Missouri-Rolla
Rolla, MO 65409
hubing@umr.edu

Abstract

This tutorial paper reviews the basic mechanisms by which signal voltages and currents on a printed circuit board produce unintentional radiated emissions.

Keywords

voltage-driven, current-driven, common-mode emissions

INTRODUCTION

To the novice EMC engineer, printed circuit board (PCB) EMI issues can be overwhelmingly complex. The mechanisms by which signal currents and voltages on a PCB become radiated electromagnetic fields are elegantly described by field theory and Maxwell's equations. Unfortunately, these descriptions are not particularly helpful to the engineer who is laying out a complex PCB or tracking down the source of a radiated emissions problem. EMC engineers generally rely on their intuition, experience and a little luck when they are attempting to trace unwanted radiated emissions back to their source.

However, PCB EMI is not quite as complex as many people think. Typical PCB structures (microstrip or stripline traces, SMT components, etc.) are not efficient antennas and their mere presence in a PCB design does not necessarily result in significant radiated emissions. Emission problems generally result when designers unintentionally build unnecessary antenna structures into their designs that are driven by signals on the board. In this paper, we review the basic mechanisms by which signal voltages and currents on a PCB drive the antennas that create radiation problems.

CIRCUIT BOARD ANTENNAS

You don't need to be an expert in antenna theory to be a good EMC engineer. However, there are two basic antenna concepts that every EMC engineer should be aware of:

1. Electrically small antennas (antennas with a maximum dimension that is much smaller than a half-wavelength) are not efficient;
2. To drive an antenna efficiently, the source must be able to pull current from one antenna part of electrically significant size and push it on to another antenna part of electrically significant size.

In other words, an efficient EMI antenna must have two parts (one part driven relative to the other by the EMI source) and both parts must not be too small relative to a wavelength. For example, in order to radiate significantly at 100 MHz ($\lambda = 3$ meters), a printed circuit board must have two metallic structures (each approximately 15 cm in size or larger) and there must be a 100-MHz source that drives one structure relative to the other.

With a little practice, spotting the possible antennas on a typical PCB is not difficult. At frequencies below a few hundred megahertz, the antennas must be fairly large. Connectors with attached cables are the most common EMI antenna parts. A connection to the chassis or enclosure can also be an effective antenna part. The key to a good PCB design is not to allow any sources on the board to drive one antenna part relative to another. Below a few hundred megahertz, the best design strategies ensure that high-speed circuitry is not located between the readily identifiable antenna parts.

Above a few hundred megahertz, smaller metallic structures (e.g. heatsinks, area-fills, etc.) can become effective antenna parts. For example, at 1 GHz ($\lambda = 30$ cm), two metallic structures with a maximum dimension of a few centimeters can become an efficient antenna. Although structures of this size are more common on printed circuit boards, sources with significant energy at these frequencies tend to be well contained and relatively easy to identify. At frequencies above a few hundred megahertz, the best design strategies involve identifying the possible sources and then making sure that there are no two antenna parts that can be driven relative to each other by these sources.

In summary, below a few hundred megahertz, identify the antenna parts and try to keep sources from driving them. Above a few hundred megahertz, identify the sources and try to keep possible antenna parts from being driven by them.

VOLTAGE-DRIVEN SOURCES

Now that we know what to look for in terms of the antenna, how do we identify the sources that drive these antennas? Although there are many possible circuit configurations typically found on PCBs that can drive antennas effectively, it is helpful to divide them into two basic categories, voltage-driven sources and current-driven sources [1]. A voltage-driven source occurs when a signal voltage appears

across two possible antenna parts resulting in radiated emissions. For example, a signal trace that couples to a heatsink may drive the heatsink relative to the return plane as illustrated in Figure 1. This mechanism is referred to as *voltage-driven*, because the level of radiated emissions is directly proportional to the signal voltage. Voltage-driven source problems are best avoided by ensuring that signal traces and components cannot readily couple to antenna “parts” of significant electrical size. The signal trace itself (even if it is electrically long) will not be an efficient antenna part as long as it is located just above or below a solid signal return plane.

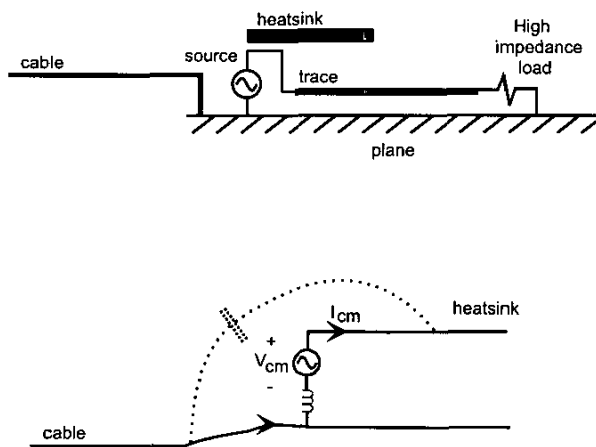


Figure 1: Voltage-driven source mechanism

CURRENT-DRIVEN SOURCES

Voltage-driven sources are relatively intuitive and easy to spot. For this reason, they are not as likely to be the major sources of radiated EMI in well-designed PCBs. However, there is another source mechanism that is often overlooked by PCB layout designers. Current-driven sources occur when high-speed circuitry is located anywhere between two antenna parts on a PCB.

Figure 2 illustrates a trace-above-plane structure with two cables attached to the plane at opposite ends. Signal current flowing in the trace-plane loop generates a magnetic flux that wraps the trace and (to a lesser extent) the plane [2]. Because there is flux wrapping the plane, a voltage is developed between any two cables attached to opposite sides of the plane. This voltage is small relative to the signal voltage, however only a few millivolts between two cables is usually sufficient to cause a product to fail to meet radiated emissions requirements.

This source mechanism is referred to as *current-driven*, because the amount of radiation is proportional to the magnetic field wrapping the plane, which is proportional to the

signal current. This source mechanism is particularly troublesome because there is often no low-cost way to correct it once it occurs. Basically, the designer has three options:

1. Reduce the signal current and/or the signal loop area
2. Relocate the cables or the circuit so that the circuit is no longer between the cables;
3. Enclose the board in a metallic structure and bond the cable shields to this structure.

The first two options involve making a new board layout. The third option is expensive and wouldn't have been necessary if the original layout had addressed this problem. The predominance of current-driven EMI sources is the main reason that designers should attempt to locate all cable connectors on one edge or corner of a PCB.

ENCLOSURE RESONANCES

Enclosing the PCB in a shielded enclosure can effectively mitigate either of the source mechanisms described above. It is important to ensure that the enclosure is well connected at the seams and bonded to the cable shields however. Otherwise, a poor enclosure becomes just another antenna “part” that can be driven relative to the cables and other metallic structures.

If there are still EMI problems with a product in a well-designed enclosure, they tend to be due to resonances that can be set up inside the enclosure. At high frequencies, it is possible for signal harmonics to coincide with cavity resonances inside the enclosure [3, 4]. When this happens, large fields can develop inside the enclosure and power can be radiated through the enclosure seams and apertures. At the PCB level, the best way to avoid this type of EMI problem is to attempt to contain the fields associated with signals that have significant harmonic content above a few hundred megahertz.

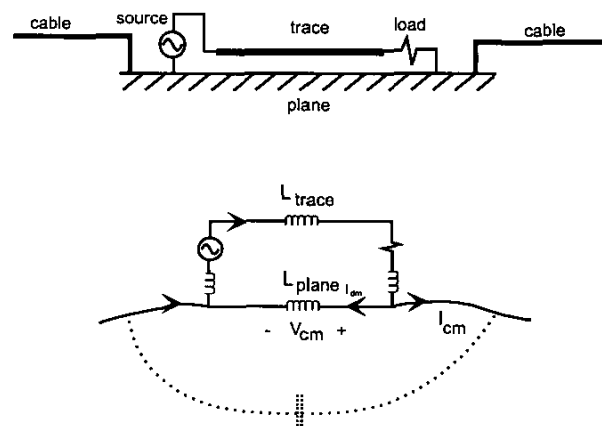


Figure 2: Current-driven source mechanism

SIGNAL COUPLING TO I/O

The voltage-driven and current-driven mechanisms described above are both mechanisms for coupling common-mode current to antenna parts such as cables. However, it is also possible for signal currents to become antenna currents by coupling directly to the traces that connect to the wires in the cable. This is most likely to be a problem when the cable is not designed to carry high-frequency signals. For example, the keyboard and mouse cables on a personal computer are not usually well shielded. If high-speed signal traces couple energy to the keyboard or mouse traces near the connector, this energy can be carried off the board. Depending on whether this coupling was due to capacitive or inductive crosstalk, this radiation mechanism could be viewed as a form of voltage-driven or current-driven EMI. However, this problem occurs frequently enough that it is worth mentioning independently. Traces connecting to unshielded cable connectors should be relatively short and kept away from high-frequency sources.

SUMMARY

Many radiated EMI problems with PCBs can be avoided by being aware of the basic mechanisms that allow signal voltages and currents to couple to EMI "antennas". At frequencies below a few hundred megahertz, the EMI antenna parts must be fairly large and therefore relatively easy to identify. In this situation, the best design strategy is to avoid locating any sources between any two antenna parts. At frequencies above a few hundred megahertz, the sources

are less numerous and relatively easy to identify. In this case, the best design strategy is to prevent possible antenna parts from being driven relative to one another by these sources.

With practice and experience, this simple strategy allows engineers to develop low-EMI PCB designs without resorting to complex models, Maxwell's equations or trial-and-error techniques.

REFERENCES

- [1] D. M. Hockanson, et. al., "Investigation of fundamental EMI source mechanisms driving common-mode radiation from printed circuit boards with attached cables," *IEEE Transactions on Electromagnetic Compatibility*, vol. EMC-38, no. 4, Nov. 1996, pp. 557-566.
- [2] D. M. Hockanson, et. al., "Quantifying EMI resulting from finite-impedance reference planes," *IEEE Transactions on Electromagnetic Compatibility*, vol. EMC-39, no. 4, Nov. 1997, pp. 286-297.
- [3] Y. Huang, et. al., "EMI considerations in selecting heat-sink-thermal-gasket materials," *IEEE Transactions on Electromagnetic Compatibility*, vol. EMC-43, no. 3, Aug. 2001, pp. 254-260.
- [4] M. Li, et. al., "An EMI estimate for shielding-enclosure evaluation," *IEEE Transactions on Electromagnetic Compatibility*, vol. EMC-43, no. 3, Aug. 2001, pp. 295-304.