Circuit Board Layout for Automotive Electronics

Todd H. Hubing
Clemson University International Center for Automotive Electronics
Clemson, SC 29634 USA

***Abstract*— EMC and environmental requirements placed on automotive electronics present unique board layout challenges. This paper discusses component placement and trace routing strategies for circuit boards that will be used in automotive applications.**

Keywords- printed circuit board; design guidelines; board layout; electromagnetic compatibility

I. Introduction

A typical automobile today has dozens of microcontrollers and hundreds of circuits. With each new model year cars and trucks rely more heavily on electronics for everything from engine and stability control to navigation and entertainment. Automotive components must operate reliably in a relatively harsh electromagnetic environment where noisy high-power circuits share tight spaces with sensitive high-speed circuits. In a typical mid-sized sedan, digital, analog and power circuits share less than 20 cubic meters of space with a variety of intentional transmitters and sensitive receivers. All of these systems are expected to operate without error in a variety of configurations, conditions and climates.

Printed circuit boards for automotive applications are generally subject to unique requirements that affect the electromagnetic compatibility of the final design. For example, the choice of components is limited by requirements related to the temperature range, mechanical strength, power consumption and cost. The board's form factor and external connections are generally pre-determined making it difficult to utilize optimum component placement. Also, most boards must be connected to an unshielded wiring harness and are required to isolate the power return (battery negative) from any chassis ground.

Engineers and board layout specialists often rely on EMC design rules [1] to help make decisions regarding component placement and trace routing. Unfortunately, the uninformed application of design rules can result in very poor circuit board layouts and tends to cause more EMC problems than it prevents. A far more effective board layout strategy is to be aware of the noise sources and susceptible components in a given design and pay attention to the current paths associated with ALL signals and noise on the board. With a little care and attention, this is not as hard a task as it may at first seem.

This paper describes some of the primary sources and susceptible components often found in automotive designs, and discusses the process of identifying current paths. It then addresses the issues of component placement, grounding and power distribution in typical automotive circuit board layouts.

II. Sources and Susceptible Components

Many potential sources of interference are readily apparent to the engineer designing automotive electronics. For example, a MOSFET dumping several amps of current into the same copper plane or trace used to return digital signal currents should be viewed with particular scrutiny. Likewise, it is clear that a trace connected to a microprocessor output switching at data rates of megabits per second must be routed with care to avoid coupling high-frequency content to the wiring harness.

However, the sources that result in the most difficult EMC problems are often unintentional. A primary example of this in the automotive world is microcontrollers and other VLSI devices that generate high-frequency noise on many of their input and output pins (including pins that are nominally low-speed). Figure 1 shows a map of the currents flowing in the lead-frame of a microprocessor programmed on a field programmable gate array (FPGA) at the third harmonic of the clock frequency. Note that many of the nominally low-speed pins have as much current flowing in them as the clock input. Several commercial automotive microcontrollers that have been mapped over the past few years have been shown to generate clock-frequency harmonics on virtually all of their inputs and outputs. This means that we cannot assume that a nominally low-speed trace (e.g. a processor reset) does not carry high-frequency currents. It is important to route all traces as if they were carrying high-frequency signals unless we can perform the appropriate measurements to determine that this is, in fact, not the case.

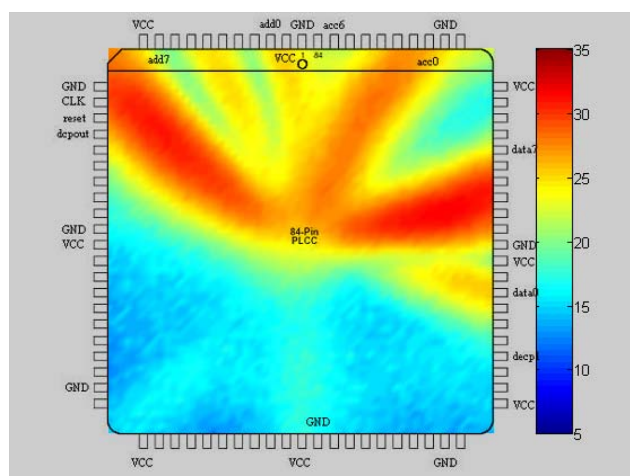

Figure 1. Currents flowing in a microprocessor package at the 3rd harmonic of the clock freq2uency.

Unintentional high-frequency noise on VLSI components has become such a significant problem that many automotive

1-4244-1350-8/07/$25.00 ©2007 IEEE

electronics companies are insisting that their suppliers provide TEM-cell measurement data [2] or similar evidence that their components are sufficiently "quiet" at RF frequencies. Near-field scans such as the one shown in Figure 1 can help to identify potentially noisy inputs and outputs.

III. Identifying Current Paths

Perhaps the most important skill that any board layout engineer can have is the ability to recognize where the currents associated with signals, power and noise sources will flow in any given situation. Generally speaking, the rules for describing the current flow are simple:

1. Current always flows in loops (i.e. current returns to its source)
2. Current takes the path(s) of least impedance.

Although most engineers are familiar with these concepts, they are often ignored when working with digital circuits. Clock and data signal traces are often routed with great care while the equally important path that returns these trace currents back to the source is completely overlooked.

At frequencies above 100 kHz, the path of least impedance is generally the path of least inductance. This means that high-frequency currents want to return to their source along a path as near to the signal trace as possible. This is good news for EMC engineers, because it means that as long as we provide a nearby return path, we can count on virtually of the current returning on that path. In fact, if a trace is routed a millimeter or less above an unbroken "ground" plane, virtually all of the high-frequency current will return on this plane directly beneath the trace.

At low frequencies (e.g. below a few kHz), the paths of least impedance are the lowest resistance paths and return currents will divide and take all possible paths in proportion to their relative resistance. This can present problems for the EMC engineer, because the lowest resistance paths may not be readily apparent and simply providing a nearby current return path will not guarantee that all the current will take this path.

In automotive designs, the most significant high-frequency currents are generally associated with digital signals and noise from digital devices. By ensuring that every high-frequency current path has a nearby, unbroken return path; many of the most difficult high-frequency emissions and susceptibility problems can be avoided. The most significant low-frequency currents are generally associated with sensors, actuators and power circuitry. The best way to control the paths of these currents is to provide each particularly sensitive or noisy component with its own isolated current return conductor. While it is not usually practical or cost effective to provide every device with its own return, it is generally necessary to isolate the return paths of high-current and low-current devices at low frequencies.

IV. Grounding Structure

The function of a circuit ground is to provide a known (usually 0-volt) reference potential. Many poor board layout decisions are the result of confusing the concepts of "circuit ground" and "current return". In order to avoid radiated emission and susceptibility problems, it is very important that automotive printed circuit board designs have 1 (and only 1) high-frequency ground and that all wire-harness wires and all chassis connections are referenced to that high-frequency ground. At the same time, it is important to control (and sometimes isolate) the current paths of potential low-frequency noise sources. When these current return paths are given names like "digital ground", "analog ground", and "chassis ground"; there appears to be a conflict with the 1-and-only-1 ground rule. This apparent conflict would disappear if engineers would learn to name conductors according to their primary function. For example, if our 1-and-only-1 high-frequency ground were the "chassis ground"; then the plane in our board whose purpose was to return digital signal currents to their source could be called "digital return". Traces or planes serving as the low-frequency return paths for sensitive sensors or high-current actuators could be named "sensor return" or "actuator return", respectively. All of these return conductors would have to be referenced (i.e. electrically tied) to the chassis ground at high frequencies along with every other conductor of significant length. However, it not necessary to violate the 1-and-only-1 high-frequency ground rule in order to provide isolated low-frequency current return paths for the devices that require them.

An example of an automotive circuit board with both high-frequency and low-frequency components is illustrated in Figure 2. It complies with the requirement that "battery negative" is isolated from chassis ground, but does not violate the 1-and-only-1 high-frequency ground rule. All wires in the wiring harness must be referenced to chassis ground (the 1-and-only-1 ground). However, their signals currents return on the digital return plane, which must be isolated at low-frequencies from chassis ground. Therefore, it is important to provide a low-impedance, high-frequency connection between digital return and chassis ground while maintaining low-frequency isolation. This is accomplished by establishing a chassis ground plane on the board and providing low-inductance capacitive connections between the two planes.

V. Component Placement

The most obvious way for noise to couple into or out-of automotive electronics is through the wiring harness (or harnesses). Therefore it is important to place components and route traces in a manner that minimizes the coupling to the harness. If there is more than one harness connector, the layout should avoid placing high-frequency components between the connectors because small voltages induced between two harness connections can easily drive high-frequency common-mode currents onto these harnesses. Low-frequency interference problems can be avoided by grouping sensitive sensor circuitry away from high-current circuitry to allow return paths to be more effectively isolated.

VI. Power Distribution and Decoupling

A good power distribution strategy can be as important as a good grounding strategy. The power bus on a printed circuit board is potentially a good coupling path for both high-frequency and low-frequency noise. Adequate decoupling (capacitance between power and power-return conductors) is essential. However, it is not always necessary or desirable to provide solid planes for power distribution.

As a general rule, power planes should only be used when the number of power connections to be made is large. Devices that are sensitive to noise on the power bus can employ power traces that are isolated from traces supplying power to high-current devices. These power traces may be filtered if necessary.

For boards with more than two layers, it is a good idea to route all power traces and/or planes on the same layer. This helps to reduce high-frequency coupling between different power buses and helps to prevent power (and power bus noise) from being routed to areas of the board where it is not needed.

Optimum decoupling strategies depend on the whether or not a power plane is utilized and what the spacing is between power and power-return planes [3]. In general, if the board does not have a solid power plane, then local decoupling capacitors should be connected between the power and power-return pins of each active component taking care to minimize the inductance of the connection by keeping loop areas small and connecting traces wide. On boards with power planes spaced 0.5 mm or more apart, decoupling capacitors should be located near the power or power-return pin of each active device and connected directly to the planes with the lowest possible connection inductance. For boards with power and power-return planes spaced less than 0.5 mm apart, the exact location of the decoupling capacitors is not particularly important since all capacitors tend to be shared by all active devices connected to the same planes. However, in this case it is particularly critical to minimize the connection inductance between the capacitors and the planes since there is a direct relation between this inductance and the effectiveness of the decoupling at high frequencies.

VII. Design Review

Every circuit board layout should be subjected to a design review before the first boards are built. Having a board layout independently reviewed by someone who is knowledgeable in EMC design practices and familiar with the function and application of the design can significantly improve the likelihood that the first prototype will not have significant problems. Even when the person laying out a board is an expert in EMC, it is possible to miss potential coupling paths or problems. Involving others in a design review significantly improves the odds that any potentially significant problems will be identified.

VIII. Conclusions

Circuit board layout for automotive electronics poses unique challenges. The best layout strategies involve identifying possible noise sources and susceptible circuits; and paying attention to the high-frequency and low-frequency current paths associated with these circuits. It is important to keep the difference between "ground" and "current return" in mind when laying out the board; and remember that all boards should have 1 (and only 1) high-frequency ground. Careful attention to component selection, placement, trace routing, grounding and power distribution early in the design process will help to ensure that your layout meets all of its EMC requirements when the first boards are built. It should also help to ensure that relatively expensive "fixes" such as add-on filters or shields are not required once your design is installed in a vehicle.

References

[1] "EMC Design Guideline Collection," Clemson University Vehicular Electronics Laboratory Web Site, http://www.cvel.clemson.edu/emc/tutorials/guidelines.html.

[2] IEC 61967-2:2005, "Integrated circuits – Measurement of electromagnetic emissions, 150 kHz to 1 GHz – Part 2: Measurement of radiated emissions, TEM-cell and wideband TEM-cell method", First edition, Sep. 2005.

[3] "Printed Circuit Board Power Bus Decoupling Questions: What is the best strategy for connecting decoupling capacitors to a printed circuit board?" Clemson University Vehicular Electronics Laboratory Web Site, http://www.cvel.clemson.edu/emc/FAQ/question-pcb01.html.

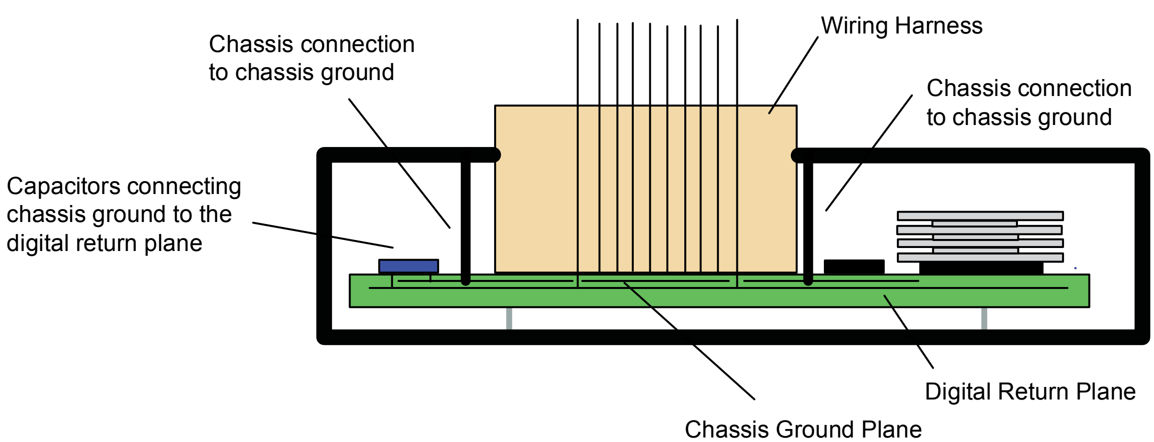

Figure 2. Chassis ground plane on an automotive printed circuit board.