Properly ground your circuits

Lee RitcheyKella Knack, - January 06, 2017

Engineers use the word "ground" in every electronic circuit to denote some part of a system or structure that is "neutral," or zero potential. Unfortunately, we often think of circuits and systems, especially those with both analog and digital signals, as having more than one ground. This concept gave rise to a recent discussion on a signal-integrity online community, which prompted us to write this article. Engineers and PCB designers often mention various types of grounds and methods for connecting them together. How those "grounds," which are really return paths, connect can significantly affect a system's performance.

The kinds of grounds mentioned in this discussion include:

- Logic ground
- Analog ground
- Chassis ground
- Safety ground
- Earth ground

Methods proposed for connecting these various "grounds" cover a broad range of options including:

- Connecting them at only one point.
- Cutting the ground plane under a mixed signal component
- Connecting them with capacitors.
- Segmenting the ground plane in a PCB such that there is only a narrow connection at one place between the analog and digital sides of the design.
- Separating the analog and digital grounds.

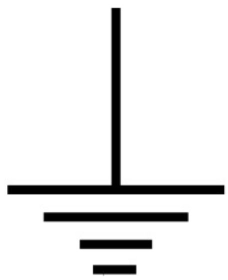

Figure 1. Digital logic ground symbol seen in most schematic diagrams.

These seemingly conflicting methods for dealing with ground can be a bit confusing. We'll start by clarifying what ground is, which should reduce the confusion.

The first question that you might ask is: How can all of the items above be ground? The answer is simple: None of them are. Ground is the one place in an electronic system that is a reference point from which we measure voltages.

If that's the only definition, then what are these other things called ground?

Digital logic ground is the "reference" terminal of a power supply for your digital logic. For most digital logic systems, it's the negative terminal of the logic power supply, usually shown with the symbol in **Figure 1**.

Analog ground is the reference terminal of the supply that powers an analog circuit. It is where one side of an analog signal source is tied. The other side of the signal source is tied to the analog input or output. Analog ground is usually designated with the symbol in **Figure 2**.

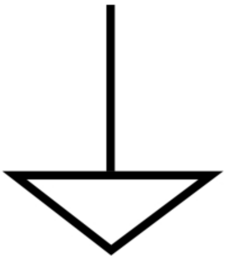

Figure 2. Analog ground schematic symbol.

Chassis ground is the name given to the connection of the safety wire from the AC mains to a product's case. It gets this name because the case of a product is often called the chassis. This wire is usually the green wire in an extension cord, in the three-wire mains connecting to a product, or the third pin on an AC connector (the round one). If you trace this green wire through a building, it will finally connect to a copper stake driven into the Earth. The purpose of this connection is to protect the operator of the product in the event that one of the mains wires accidentally makes a connection to the case or "chassis" of the product. Thus, it is a safety-only function.

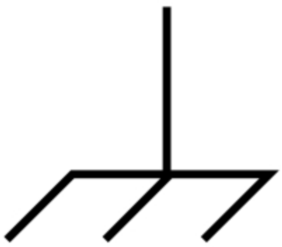

Figure 3. "Chassis" ground schematic symbol.

Sometimes, EMI engineers erroneously refer to this "Chassis ground" (**Figure 3**) as a place that has some function in the containment of EMI. This statement never has or never will be based on fact because it has no role in this part of an electronic design.

Safety ground is another name used to describe chassis ground. Earth ground is another name for the safety ground.

All of these names lead to the question of how to connect your "ground" circuits (return paths) together or whether they should be tied together in the first place, and, if so, why. This question usually arises either on how to protect sensitive analog signals from outside noise sources or how to contain EMI.

Handling Analog Signals

Taking the analog signal problem first, you need to protect analog signals from sources of outside noise that could degrade signal performance. **Figure 4** is an example of a typical mixed analog and digital IC showing the two sides of the circuit with both an analog ground pin and a digital ground pin. It's representative of most of the problems when designing mixed-signal electronics.

The red highlight indicates what is called the "analog decision-making loop." This is the circuit that must be protected from outside noise sources for the circuit to perform properly. The IC has an analog "ground" pin and a digital "ground" pin. You need to understood how to apply these pins to arrive at a proper PCB design. The digital side of this mixed-signal IC has transient currents flowing through its ground lead. These currents are associated with the internal digital processing of the analog signal and they drive the IC's output transmission lines. If this were an 8-bit A/D converter in a logic system with 2.5V logic levels, the current transients flowing in this path could be as large as 200 mA. This ?I or rapidly changing current flowing through the ground lead inductance can develop voltage transients as large as 100 mV between ground on the PCB and ground on the die. This is an acceptable transient for the logic circuit.

If the circuit under discussion is a 12-bit A/D converter, the analog side of the circuit is tasked with resolving voltage differences of 0.5 mV out of a total signal swing of 2 V. If there is only one ground path out of the IC, the 100 mV digital switching transient would be superimposed on the analog signal rendering the circuit useless. This is why the analog side of the IC has a separate ground path *out of the package*.

Figure 4 is typical of circuits that have application notes or other guidelines that specify an analog ground plane and a digital ground plane or splitting the ground plane under the component. Doing either of these things detracts from the actual engineering problem of protecting the analog signal loop from external noise. (Note: The direction of the current flow arrow in Fig. 4 is the flow of the electrons that make up the current flow.)

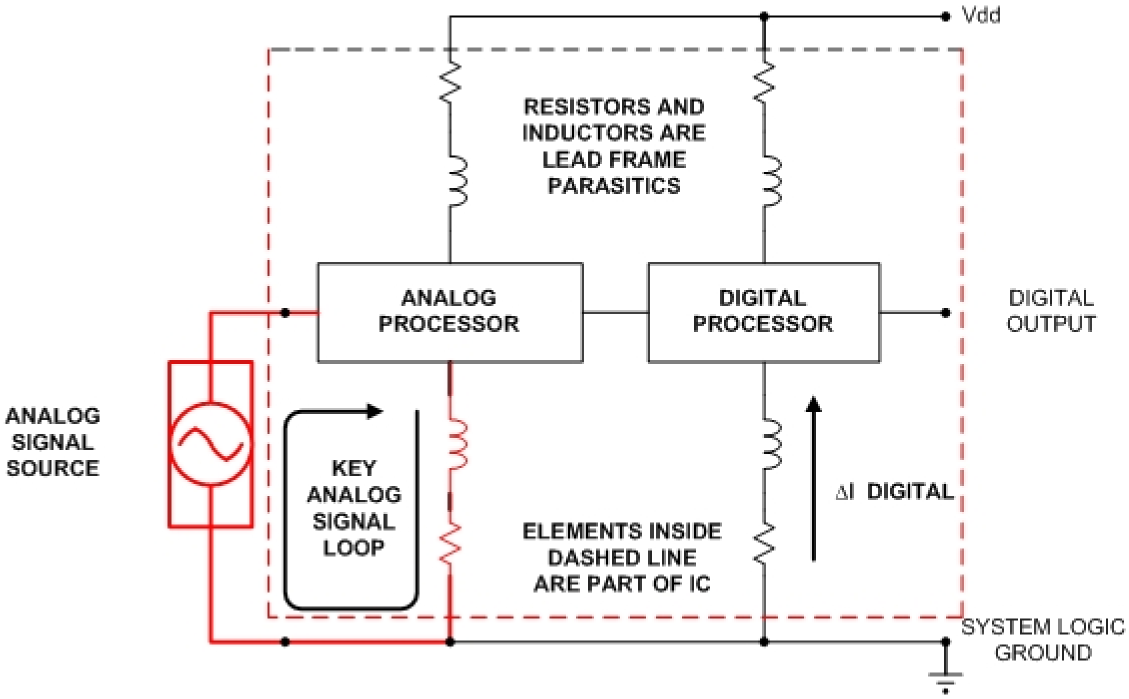

Figure 4. An Analog to Digital Converter typically has separate pins for analog and signal returns.

Splitting the ground plane under the component creates an unwanted side effect. Signals that must cross from one side of the cut to the other don't have a path for their return current. That current will find another way to return to its source, which can lead to signal integrity or EMI problems.

Noise enters the analog signal loop in two ways. The first is by coupling into either side of the loop by capacitive or magnetic coupling from an adjacent signal traveling too close (we usually call this crosstalk). Crosstalk can be generated by the electric component of an EM field (capacitive crosstalk) or the magnetic component of the EM field (inductive crosstalk). Which form exists depends on the configuration of the two conductors that are next to each other.)

The second way that noise can affect analog circuits is by allowing the "ground" side of the path to be shared by another signal. This usually happens when the connection between the analog source and the analog “ground” pin of the device is made to the ground plane some distance away from the part. In most cases, both of these problems are addressed by using a shielded cable that has its two connections made at the terminals of the IC, one connection being the shield that connects to the "analog ground" terminal of the device and the center conductor that connects to the input side of the analog device. Examples of this type of circuit are:

- The connection between the read head on a disc drive and the preamplifier
- The connection between a strain gage and the input amplifier
- The connection between a phonograph needle and the input preamplifier (only old folks know about this!)

The example in **Figure 4** deals with a system where the analog source is "off board." When both the source and load are on the same PCB, the proper way of dealing with the “analog loop” is to look at where it is and make layout choices that protect the loop from crosstalk and voltage gradients in the “ground” part of the circuit that would compromise performance. In almost all cases, this problem is dealt with by choosing to carefully place the components on the PCB surface so that no currents from other circuits flow through the region where the analog decision making loop is located. Examples of this type of circuit are the connections between amplifier stages in a radio or a stereo system

Handling EMI
Sometimes I find it useful to quote EMI engineer Bruce Archambeault when the subject of ground comes up in relation to EMI: "Ground is a place for potatoes and carrots."

The reason that both Bruce and I make this statement is that using the word "ground" in discussions about EMI has no value. In fact, it distracts us from the task at hand, which is containing RF energy that might escape our products and create and EMI failure.

Said another way, *none of the things listed as grounds at the beginning of this article have any bearing on EMI containment*. The items that are important to EMI containment are shields on cables and Faraday cages surrounding the products, but those are the subjects of another article.

There is a very large body of misinformation in the form of application notes and guidelines with respect to what ground is and how to use it. Some of these application notes indicate that the ground plane should be segmented into and analog side and a digital side and the two sides connected at only one point. Others suggest that there be two discretely different planes, one analog and the other digital. How these two planes are to be connected varies with each application note. My experience with these notes is that they are treating a problem that hasn't been proven to exist. The worst thing about such notes is that they don't address the actual problem: protecting the decision-making loop from outside noise sources.

Consider the following questions when choosing how to design a return network.

- Does a real problem exist?
- Does the proposed solution solve the problem?
- Does the proposed solution create a new problem such as an EMI problem?

If these three questions don't have valid answers, chances are the solution is simply made up and may well create a problem, such as an EMI problem, that might otherwise not exist. Many EMI problems I've solved had had their origins in split ground planes which I often encountered in small disc drives in the late 1990s and early 2000s.

Electronic systems have networks we call ground that have both AC and DC voltage gradients caused by the currents that flow in them. Thus, they can't be considered equipotential with magic properties with respect to EMI.

A PCB does not need an analog ground plane and a digital ground plane because having them doesn't guarantee proper operation of the analog section of the product. Instead, it should have only one ground plane that should be continuous throughout the PCB, followed by careful design of the decision-making loop.

Splitting a ground plane destroys its integrity as an ultralow impedance connection between all of the components in a circuit and you should never design it into a board. I've asked more than 9,000 students in my signal-integrity classes if they have examples where splitting a ground plane improved performance. To date, none have been able to produce one nor have any of my engineering colleagues. As Kenneth Wyatt noted, "The latest thinking, however (Todd Hubing, Clemson University), is that it's best to keep the return planes as a single plane and be careful about routing the signal traces (keeping in mind the corresponding return currents), so they don't cross the A/D boundary."

In those rare cases where the return plane must be split for high voltage isolation purposes, signals that must cross the split will need to do so in such a manner that the need for a continuous return current path is not required. Transformers, opto-isolators, and other types of isolators, are often used here.

As stated earlier, "Chassis" ground is a safety-only feature and has no role in the electronic function of the circuits or their EMI performance. Thus, there's no need to connect logic ground to the "Chassis" ground and, in some instances, it's not allowed.

Perhaps one of the toughest challenges a design engineer faces is sorting through all the misinformation in print and online that is inaccurate or, often, simply made up by someone who has not done the necessary research to ensure the advice given is technically valid.

I hope you found this article a good starting point for demystifying the things erroneously referred to as "ground" in PCB and system design.

—Authors Ritchey & Knack are with Speeding Edge

Also see:

- The myth called "ground"
- Crossing the river: The hazards of crossing a split-plane gap with a high-speed signal
- Return path discontinuities and EMI: Understand the relationship
- EMI and emissions: rules, regulations, and options
- Grounding and shielding: No size fits all
- EMI and emissions: rules, regulations, and options
- EMC questions answered (part 7)
- Successful PCB grounding with mixed-signal chips - Part 1: Principles of current flow
- Ten best practices of PCB design
- Questions on PC boards for EMC mitigation
- Understanding common-mode signals
- PCB signal coupling can be a problem
- Designer's Notebook: Signal Isolation