Grounding in mixed-signal systems demystified, Part 1

By Sanjay Pithadia, *Analog Applications Engineer,*
and Shridhar More, *Senior Analog Applications Engineer*

Introduction

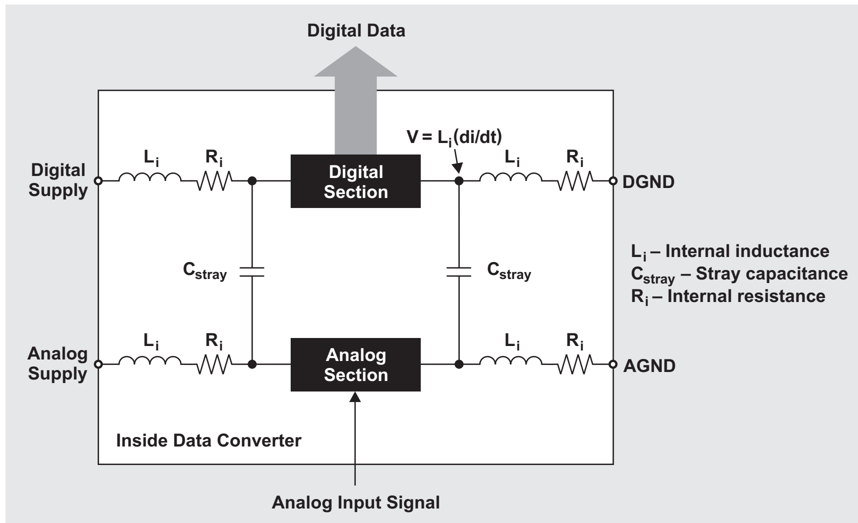
Every signal-processing system requires mixed-signal devices, such as analog-to-digital converters (ADCs) and/or digital-to-analog converters (DACs). The need for processing analog signals with a wide dynamic range imposes the requirement to use high-performance ADCs and DACs. Maintaining performance in a noisy digital environment is dependent upon using good circuit-design techniques like proper signal routing, decoupling, and grounding.

Undoubtedly, grounding is one of the most discussed subjects in system design. Though the basic concepts are relatively simple, the implementation is difficult. For linear systems, the ground is the reference against which the signal is based; and, unfortunately, it also becomes the return path for the power-supply current in unipolar supply systems. An improper application of grounding strategies can degrade the performance in high-accuracy linear systems. There is no "cookbook" that guarantees good results, but there are a few things that, if not done properly, can cause issues.

This article is the first of a two-part series that looks closely at the grounding techniques used in mixed-signal systems. Part 1 explains typical terminologies and ground planes and introduces partitioning methods. Part 2 explores techniques for splitting the ground planes, including pros and cons. It also explains grounding in systems with multiple converters and multiple boards. Part 2 will appear in a future issue of *Analog Applications Journal*.

A term often used in system design is *star ground*. This term builds on the theory that all voltages in a circuit are referred to as a single ground point, or star ground point. The key feature is that all voltages are measured with respect to a particular point in the ground network, not just to an undefined ground wherever one can clip a probe. Practically, it is difficult to implement. For example, in a star ground system, drawing out all signal paths to minimize signal interaction and the effects of high-impedance signal or ground paths causes implementation problems to arise. When power supplies are added to the circuit, either they add unwanted ground paths or their supply currents flowing in the existing ground paths are large enough or noisy enough to corrupt the signal transmission.

Figure 1. AGND and DGND pins in a data converter



Interpretation of AGND and DGND pins in mixed-signal devices

Digital- and analog-design engineers tend to view mixed-signal devices from different perspectives, but every engineer who uses a mixed-signal device is aware of analog ground (AGND) and digital ground (DGND). Many are confused about how to deal with these grounds; and, yes, much of the confusion comes from how the ADC ground pins are labeled. Note that the pin names, AGND and DGND, refer to what's going on inside the component and do not necessarily imply what one should do with the grounds externally. Data-converter datasheets usually suggest tying the analog and digital grounds together at the device. However, the designer may or may not want the data converter to become the system's star ground point. What should be done?

As illustrated in Figure 1, the grounds inside a mixed-signal IC are typically kept separate to avoid coupling digital signals into the analog circuits. An IC designer cannot do anything about the internal inductance and resistance (negligible compared to the inductance) associated with connecting the pads on the chip to the package pins. The rapidly changing digital currents produce a voltage (di/dt) in digital circuits, which inevitably couples into the analog circuits through the stray capacitance.

The IC works well in spite of such coupling. However, in order to prevent further coupling, the AGND and DGND pins should be joined together externally to the same low-impedance ground plane with minimum lead lengths. Any extra external impedance in the DGND connection can cause more digital noise and, in turn, can couple more digital noise into the analog circuit through the stray capacitance.

Analog or digital ground plane, or both?

Why is a ground plane needed? If a bus wire is used as a ground instead of a plane, calculations must be done to determine the bus wire's voltage drop because of its impedance at the equivalent frequency of most logic transitions. This voltage drop creates an error in the final accuracy of the system. To implement a ground plane, one side of a double-sided PCB is made of continuous copper and is used as a ground. The large amount of metal has the lowest possible resistance and lowest possible inductance because of the large, flattened conductor pattern.

The ground plane acts as a low-impedance return path for decoupling high-frequency currents caused by fast digital logic. It also minimizes emissions from electromagnetic interference/radio-frequency interference (EMI/RFI). Because of the ground plane's shielding action, the circuit's susceptibility to external EMI/RFI is reduced. Ground planes also permit high-speed digital or analog signals to be transmitted via transmission-line (microstrip or stripline) techniques, where controlled impedances are required.

As mentioned earlier, the AGND and DGND pins must be joined together at the device. If the analog and digital grounds have to be separated, should both be tied to the analog ground plane, the digital ground plane, or both?

Remember that a data converter is *analog*! Thus, the AGND and DGND pins should be connected to the analog ground plane. If they are connected to the digital ground plane, the analog input signal is going to have digital noise summed with it, because it is probably single-ended and referenced to the analog ground plane. Connecting the pins to a quiet analog ground plane can inject a small amount of digital noise into it and degrade the noise margin of the output logic. This is because the output logic is now referenced to the analog ground plane and all the other logic is referenced to the digital ground plane. However, these currents should be quite small and can be minimized by ensuring that the converter output does not drive a large fan-out.

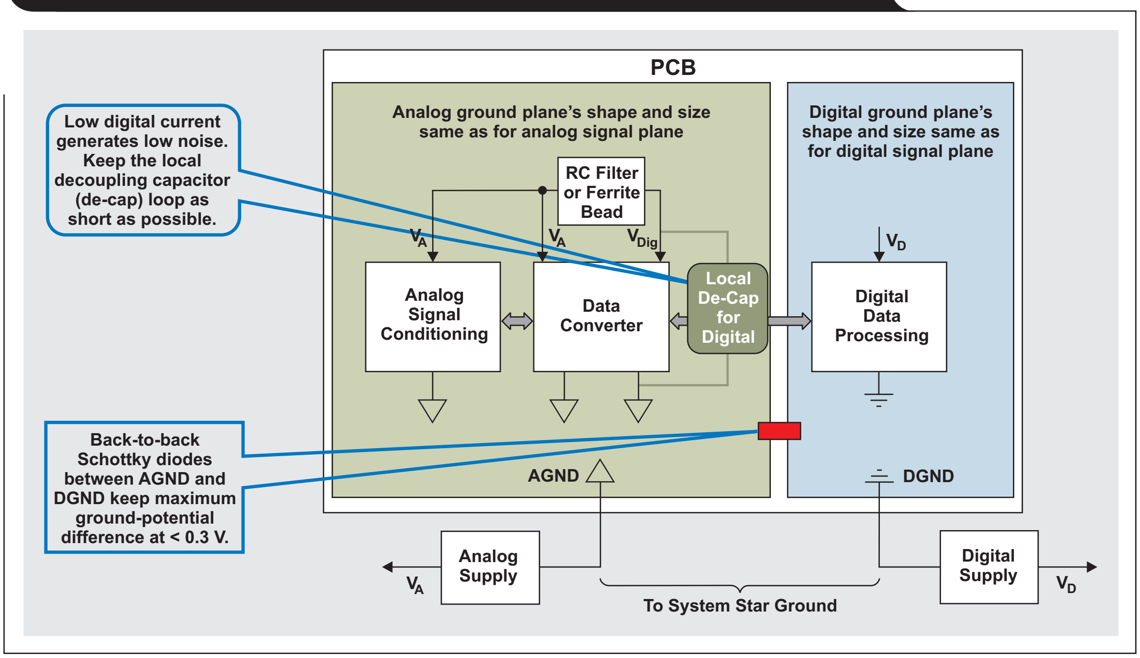
It is possible that the devices used in a design have either low digital currents or high digital currents. The grounding scheme is different for both cases. Traditionally, data converters may be thought of as low-current devices (such as flash ADC). But today's data converters with on-chip analog functions are becoming more and more digitally intensive. Along with the additional digital circuitry come larger digital currents and noise. For example, a sigma-delta ADC contains a complex digital filter that adds considerably to the digital current in the device.

Grounding data converters with low digital currents

As mentioned, a data converter (or any mixed-signal device) is analog. In any system, the analog signal plane is where all the analog circuitry and mixed-signal devices are placed. Similarly, the digital signal plane has all the digital data-processing circuits. The analog and digital ground planes should have the same size and shape as the respective signal planes.

Figure 2 summarizes the approach for grounding a mixed-signal device with low digital currents. The analog ground plane is not corrupted because the small digital

Figure 2. Grounding data converters with low internal digital currents



transient currents flow in the small loop between V_{Dig}, the local decoupling capacitor, and DGND (the green line). Figure 2 also shows a filter between the analog and digital power supplies. There are two types of ferrite beads: high-Q resonant beads and low-Q nonresonant beads. Low-Q beads are commonly used for power-supply filtering in series with the power connection.

Grounding data converters with high digital currents

The circuit in Figure 2 depends on the decoupling capacitor between V_{Dig} and DGND to keep the digital transient currents isolated in a small loop. However, if the digital currents are significant enough and have components at DC or low frequencies, the decoupling capacitor may have to be so large that it is impractical. Any digital current that flows outside the loop between V_{Dig} and DGND must flow through the analog ground plane. This may degrade performance, especially in high-resolution systems. An alternative grounding method for a mixed-signal device with high levels of digital currents is shown in Figure 3. The AGND pin of the data converter is connected to the analog ground plane, and the DGND pin is connected to the digital ground plane. The digital currents are isolated from the analog ground plane, but the noise between the two ground planes is applied directly between the device's AGND and DGND pins. The analog and digital circuits must be well isolated. The noise between AGND and DGND pins must not be large enough to reduce internal noise margins or cause corruption of the internal analog circuits.

Connecting analog and digital ground planes

Figures 2 and 3 show optional back-to-back Schottky diodes connecting the analog and digital ground planes. The Schottky diodes prevent large DC voltages or low-frequency voltage spikes from developing across the two planes. These voltages can potentially damage the mixed-signal IC if they exceed 0.3 V, because they appear directly between the AGND and DGND pins.

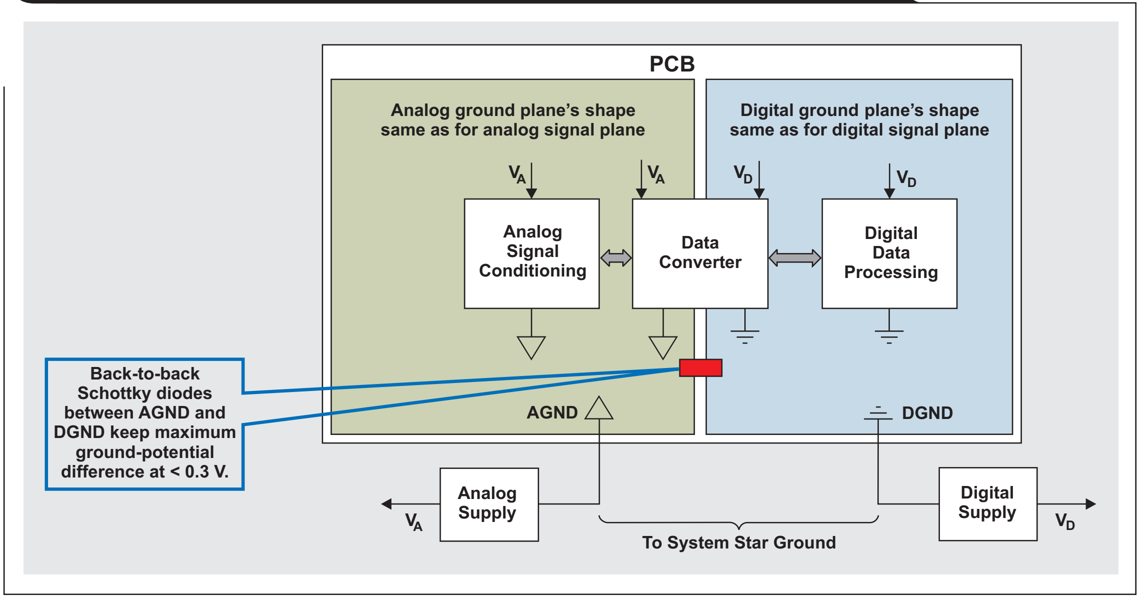
As an alternative to the back-to-back Schottky diodes, a ferrite bead can provide a DC connection between the two planes but isolate them at frequencies above a few megahertz where the ferrite bead becomes resistive. This protects the IC from DC voltages between AGND and DGND, but the DC connection provided by the ferrite bead can introduce unwanted DC ground loops and may not be suitable for high-resolution systems. Whenever AGND and DGND pins are separated in the special case of ICs with high digital currents, provisions should be made to connect them together if necessary.

Jumpers and/or strap options allow both methods to be tried to verify which gives the best overall system performance.

Isolation or partitioning: Which is important for ground planes?

A common concern is how to isolate the grounds so that the analog circuit does not interfere with the digital circuit. It is a well-known fact that digital circuitry is noisy. Saturating logic draws large, fast current spikes from its supply during switching. Conversely, analog circuitry is quite vulnerable to noise. It is not that the analog circuit might interfere with the digital logic. Rather, it is possible that the high-speed digital logic might interfere with the low-level analog circuits. So the concern should be how to prevent digital-logic ground currents from contaminating the low-level analog circuitry on a mixed-signal PCB. The first thought might be to split the ground planes to isolate DGND from AGND. Although the split-plane approach can

Figure 3. Grounding data converters with high internal digital currents



be made to work, it has many problems--especially in large, complex systems.

There are two basic principles of electromagnetic compatibility (EMC):

1. Currents should be returned to their sources locally and as compactly as possible. If not, a loop antenna should be created.
2. A system should have only one reference plane, as two references create a dipole antenna.

During EMC tests, most problems are observed when traces are routed across a slot or a split in a ground or power plane. Since this routing causes both radiation and crosstalk issues, it is not recommended.

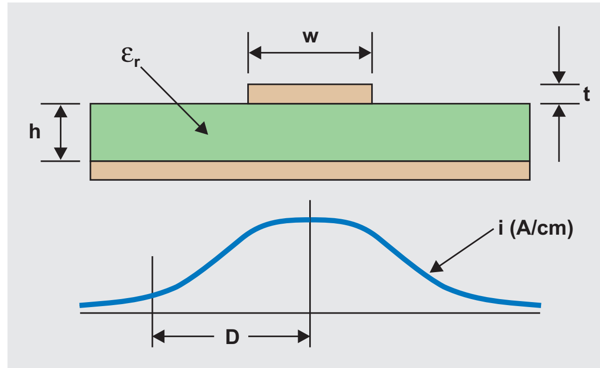
It is important to understand how and where the ground currents in a split plane actually flow. Most designers think only about where the signal current flows and ignore the path taken by the return current. The high-frequency signals have a characteristic of following the path of least impedance (inductance). The path's inductance is determined by the loop area that the path encloses. The larger the area that the current has to travel to return to the source, the larger the inductance will be. The smallest inductance path is directly next to the trace. So, regardless of the plane--power or ground--the return current flows on the plane adjacent to the trace. The current spreads out slightly in the plane but otherwise stays under the trace. The actual distribution is similar to a Gaussian curve in nature. Figure 4 illustrates that the return-current flow is directly below the signal trace. This creates the path of least impedance.

The current-distribution curve for the return path is defined by

$$i\ (\text{A/cm}) = \frac{I_O}{\pi h} \times \frac{1}{1+\left(\frac{D}{h}\right)^2},$$

where I_O is the total signal current (A), h is the height of the trace (cm), and D is the distance from the trace (cm). From this equation it can be concluded that digital ground currents resist flowing through the analog portion of the ground plane and so will not corrupt the analog signal.

Figure 4. Distribution of return current



For reference planes, it is important that the clearance sections of vias do not interfere with the return current's path. In the case of an obstacle, the return current finds a way around it, as shown in Figure 5. However, this rerouting will most likely cause the current's electromagnetic fields to interfere with the fields of other signal traces, introducing crosstalk. Moreover, this obstacle adversely affects the impedance of the traces passing over it, leading to discontinuities and increased EMI.

Part 2 of this two-part article series will discuss the pros and cons involved in splitting the ground planes and will also explain grounding in systems with multiple converters and multiple boards.

References

1. H. W. Ott, "Partitioning and layout of a mixed-signal PCB," *Printed Circuit Design*, pp. 8–11, June 2001.
2. "Analog-to-digital converter grounding practices affect system performance," Application Report. Available: www.ti.com/sbaa052-aaj

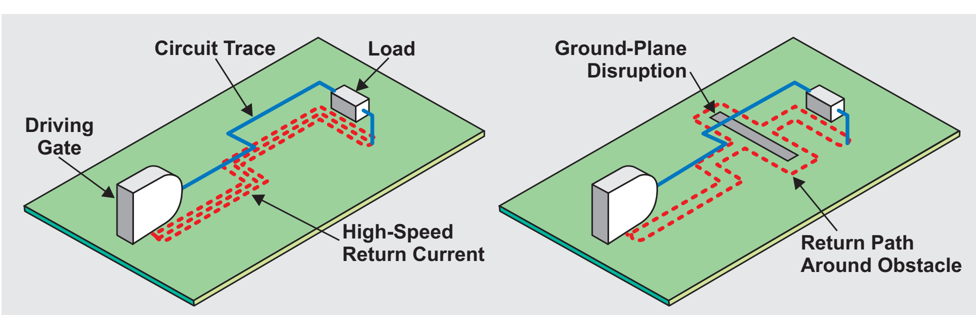
Related Web sites

Data Converters:
www.ti.com/dc-aaj

For examples of grounding for precision data converters, visit: **www.ti.com/e2egrounding-aaj**

Subscribe to the AAJ:
www.ti.com/subscribe-aaj

Figure 5. Return current with and without slot



TI Worldwide Technical Support

Internet

TI Semiconductor Product Information Center Home Page
support.ti.com

TI E2E™ Community Home Page
e2e.ti.com

Product Information Centers

Americas Phone +1(512) 434-1560

Brazil Phone 0800-891-2616

Mexico Phone 0800-670-7544

Fax +1(972) 927-6377
Internet/Email support.ti.com/sc/pic/americas.htm

Europe, Middle East, and Africa

Phone

European Free Call	00800-ASK-TEXAS (00800 275 83927)
International	+49 (0) 8161 80 2121
Russian Support	+7 (4) 95 98 10 701

Note: The European Free Call (Toll Free) number is not active in all countries. If you have technical difficulty calling the free call number, please use the international number above.

Fax	+(49) (0) 8161 80 2045
Internet	www.ti.com/asktexas
Direct Email	asktexas@ti.com

Japan

Phone	Domestic	0120-92-3326
Fax	International	+81-3-3344-5317
	Domestic	0120-81-0036
Internet/Email	International	support.ti.com/sc/pic/japan.htm
	Domestic	www.tij.co.jp/pic

Asia

Phone

International	+91-80-41381665
Domestic	Toll-Free Number

Note: Toll-free numbers do not support mobile and IP phones.

Australia	1-800-999-084
China	800-820-8682
Hong Kong	800-96-5941
India	1-800-425-7888
Indonesia	001-803-8861-1006
Korea	080-551-2804
Malaysia	1-800-80-3973
New Zealand	0800-446-934
Philippines	1-800-765-7404
Singapore	800-886-1028
Taiwan	0800-006800
Thailand	001-800-886-0010

Fax	+8621-23073686
Email	tiasia@ti.com or ti-china@ti.com
Internet	support.ti.com/sc/pic/asia.htm

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, enhancements, improvements and other changes to its semiconductor products and services per JESD46, latest issue, and to discontinue any product or service per JESD48, latest issue. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All semiconductor products (also referred to herein as “components”) are sold subject to TI’s terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its components to the specifications applicable at the time of sale, in accordance with the warranty in TI’s terms and conditions of sale of semiconductor products. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by applicable law, testing of all parameters of each component is not necessarily performed.

TI assumes no liability for applications assistance or the design of Buyers’ products. Buyers are responsible for their products and applications using TI components. To minimize the risks associated with Buyers’ products and applications, Buyers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI components or services are used. Information published by TI regarding third-party products or services does not constitute a license to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of significant portions of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI components or services with statements different from or beyond the parameters stated by TI for that component or service voids all express and any implied warranties for the associated TI component or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Buyer acknowledges and agrees that it is solely responsible for compliance with all legal, regulatory and safety-related requirements concerning its products, and any use of TI components in its applications, notwithstanding any applications-related information or support that may be provided by TI. Buyer represents and agrees that it has all the necessary expertise to create and implement safeguards which anticipate dangerous consequences of failures, monitor failures and their consequences, lessen the likelihood of failures that might cause harm and take appropriate remedial actions. Buyer will fully indemnify TI and its representatives against any damages arising out of the use of any TI components in safety-critical applications.

In some cases, TI components may be promoted specifically to facilitate safety-related applications. With such components, TI’s goal is to help enable customers to design and create their own end-product solutions that meet applicable functional safety standards and requirements. Nonetheless, such components are subject to these terms.

No TI components are authorized for use in FDA Class III (or similar life-critical medical equipment) unless authorized officers of the parties have executed a special agreement specifically governing such use.

Only those TI components which TI has specifically designated as military grade or “enhanced plastic” are designed and intended for use in military/aerospace applications or environments. Buyer acknowledges and agrees that any military or aerospace use of TI components which have ***not*** been so designated is solely at the Buyer's risk, and that Buyer is solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI has specifically designated certain components as meeting ISO/TS16949 requirements, mainly for automotive use. In any case of use of non-designated products, TI will not be responsible for any failure to meet ISO/TS16949.

Products		**Applications**	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Applications Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com
Wireless Connectivity	www.ti.com/wirelessconnectivity		

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265