Jan 15, 2004

Basic Switching-Regulator-Layout Techniques

This article addresses some of the fundamentals of laying out a switching-regulator board. Although it focuses on a step-up switching-regulator, the concepts it covers are also useful when working with other types of switching regulators. The article talks about the importance of ground schemes, component placement, reducing noise interference, and reducing stray capacitance and inductance.

When considering how best to lay out a switching regulator board, it's good to recall its purpose, which is to supply a steady voltage of a specific magnitude. Experienced layout designers pay close attention to the grounding scheme. They know that ground is never perfect. Ground is not "just ground," and what you do with it is crucial to the success of the circuit. They pay particular attention to the location of regulator components.

Ground

It is perhaps a mistake to let undergraduate engineers draw the three small lines that represent ground. That symbol tends to foster the fantasy that ground is ideal. When one draws long lines to connect circuit components to a power supply's or battery's negative terminal, one more readily intuits that ground is flawed. Those lines suggest that currents flow back to the power source through the resistance and inductance of a ground plane or trace, creating voltage drops that deviate from the perfectly steady voltage that we typically call zero volts.

The boost converter of Figure 1 can illustrate why it's necessary to account for imperfect grounds. This regulator relies on the reference within the controller IC and two feedback resistors to generate a specific voltage. To obtain accurate feedback and therefore an accurate output, the grounds of the reference, the resistor divider, and the output capacitor must reside at the same potential. More specifically, the voltage of the controller's analog ground pin (which is the reference's ground) and the voltage of the resistor divider's ground terminal must equal the voltage of the output capacitor's ground terminal. The output capacitor's ground-terminal voltage is important because the load, which is what requires the regulator's accurate output voltage, is usually placed next to the output capacitor—and thus we want the feedback to be referred to that particular part of ground.

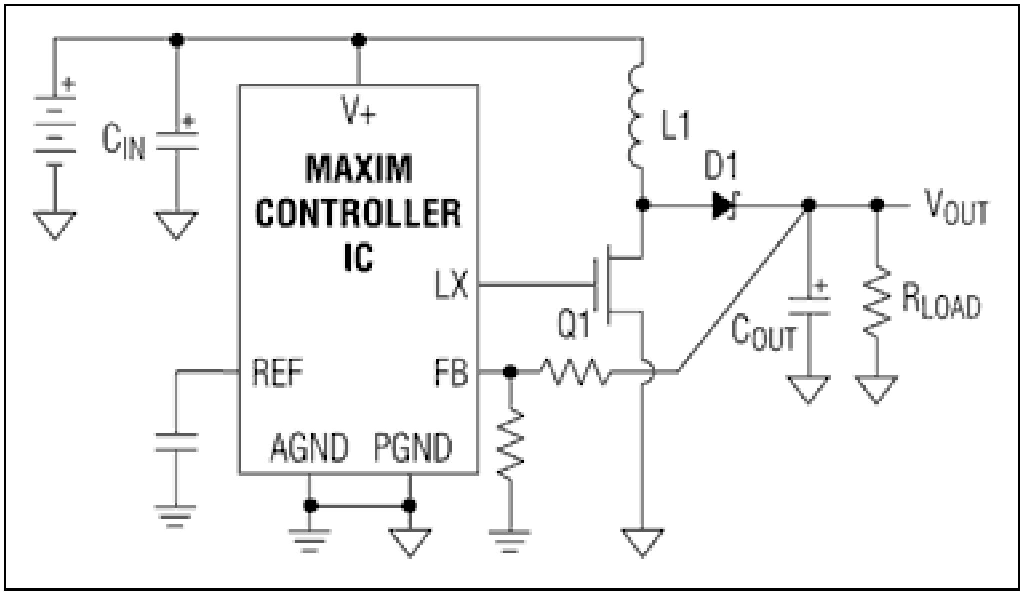

Figure 1. The ideas behind a successful board layout for this step-up switching regulator also apply to the layout of other switching regulator topologies.

The controller needs an accurate voltage fed back to it for another reason. To achieve jitter-free switching, the controller requires an accurate picture of any AC perturbations of the output voltage. It receives an accurate picture via the feedback.

Component Placement

Proper placement of the regulator's components is important. For example, the reference within the controller must be bypassed with a capacitor placed close to the REF pin, otherwise noise on the reference could affect the output voltage. This bypass capacitor's ground terminal must connect to a quiet ground, which is isolated from the noisier power ground. As previously discussed, the controller's analog ground pin and the resistor divider's ground terminal also need quiet, isolated grounds.

Why must we isolate the noisier ground from the quieter one? After all, we'll have to connect the grounds of the two sections together anyway. Such isolation is necessary to prevent high-level switching currents from returning to the battery or supply through the same ground-return path as the analog signals. If that happens, the ground path of those sensitive signals will be disturbed; the high-level switching currents flowing through the ground's resistance and inductance will cause the voltage along the return path to vary.

A look at the noisier power section can show how best to isolate it from the rest of the circuit. Figure 2 depicts the two current pathways of the regulator's power section. When the MOSFET is on, current flows through the input loop; when it's off, it flows through the output loop. By placing the components that make up each of the two loops close to each other, the high currents remain in the regulator's power section (and out of the ground return path of the quiet components). So C_{IN}, L1, and Q1 should be close to each other. Also, C_{IN}, L1, D1, and C_{OUT} should also be close. The two loops are drawn somewhat unusually in Figure 2 to clarify which components belong close together.

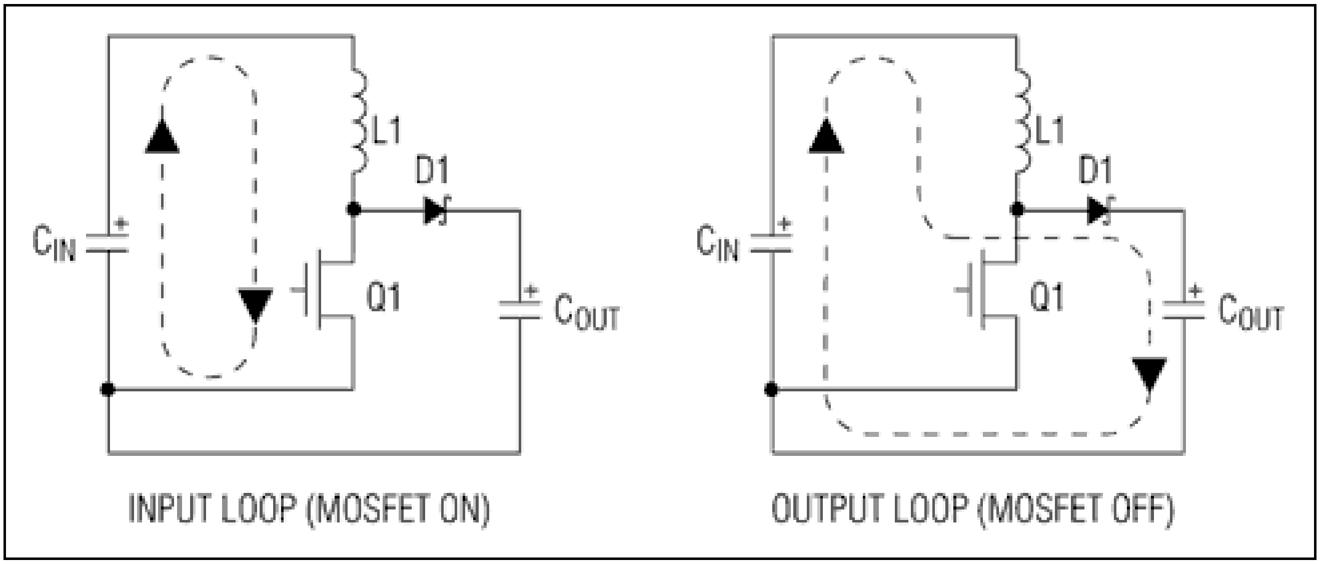

Figure 2. Special care must be taken to place the components of each of the two current loops pictured here close together. Using short, wide traces to achieve this tight layout improves efficiency, reduces ringing, and helps prevent interference to quieter parts of the circuit.

An actual layout usually involves some compromise. That could be the case when laying out the components of the two loops mentioned above. If choices need be made as to which components that should be placed close together actually are placed togethe, then determine which components in each loop have discontinuous current flowing through them. Those are the components that are most important to position close to each other so as to minimize stray inductance. See Minimizing Stray Capacitance and Inductance below.

Other Considerations

Regardless of whether a battery or a power supply powers the step-up switching regulator, the power source exhibits a non-zero resistance. That means that as the regulator draws quickly-changing current from the power source, the power source's voltage varies. To ameliorate that effect, board designers place the input bypass capacitor(s) near the two power loops described above (sometimes two capacitors are used—a ceramic and a polarized capacitor in parallel). This isn't done to steady the voltage fed to the power section; the power section will still function well if the voltage feeding it varies. Instead, placing the bypass capacitor near the power loops helps confine high ac currents to the power section, which helps keep those currents from interfering with quieter circuitry.

How might that interference occur? Three ways. First, as mentioned above, if the power section's ground-return current flows through part or all of the ground-return path of some sensitive portion of the regulator's analog circuitry, it would add switching noise to that ground path due to the resistance and inductance within it. That ground noise would degrade the accuracy of the regulator's output. It could also disturb other sensitive circuits that reside on the same board. Second, similar to concerns about the ground path, switching noise on the battery's or power supply's positive rail can be conducted to other components powered by that same rail. That includes the controller IC, whose reference could bounce. Adding an R/C filter at the controller's supply pin can help if the voltage across the input bypass capacitor varies. Third, the bigger the area over which ac currents flow, the bigger the magnetic field they create, and hence the greater the chance that those currents will cause interference. Placing the input bypass capacitor next to the power section minimizes that area and thus the potential interference.

Noise can also cause problems if the two divider resistors are placed improperly. Placing the two resistors next to the controller's FB pin ensures that a relatively noise-free voltage is fed back to the controller. Positioning the resistors that way minimizes the length of the trace leading from the mid-point of the resistor divider to the switching regulator's FB pin - a necessity because both the resistor divider and the input of the internal comparator at the FB pin are high impedances, and thus the trace connecting them is prone to picking up (primarily through capacitive coupling) the noise that switching regulators inevitably produce. You can, however, make the trace that runs from the regulator's output to the "top" of the resistor divider and the trace that runs from the "bottom" or ground side of the resistor divider to the ground side of the output capacitor relatively long; the low output impedance of the switching regulator reduces coupled noise on those traces.

Minimizing Stray Capacitance and Inductance

Identifying nodes in the Figure 1 circuit where voltage quickly changes indicates where to minimize capacitance, as a capacitor's voltage prefers not to change quickly. The node formed by the junction of the inductor, diode, and MOSFET is the only such point in the power portion of the circuit; this node is near ground when the switch is on and rises to a diode drop above the output voltage when the switch is off. Make sure to run the board traces in a manner that minimizes the stray capacitance at this node. If stray capacitance slows the voltage transitions of this node, the regulator's efficiency will suffer. Keeping the size of this node small not only helps reduce its stray capacitance, it also reduces the EMI that emanates from it. Don't make the area of the node small by using narrow traces, however. Instead, use wide, short traces.

Identifying circuit branches with quickly changing currents shows where to minimize inductance. Reminiscent of the voltage across a capacitor, the current through an inductor doesn't like to change quickly. When current through an inductance rapidly changes, it causes the voltage at that inductance to spike and ring, creating potential EMI problems. Also, the amplitude of that ringing voltage can be sufficiently high to cause damage to various circuit elements.

Figure 3 shows the current waveforms for the three branches of the circuit. The current I_1 doesn't present a problem because it changes in a relatively gradual manner; besides, a large inductance is already present there-i.e., L1 itself. However, inductance in series with the MOSFET can indeed cause a problem because current I_3 changes abruptly. This series inductance includes inductance from anything within I_3's return path-up to C_{IN}'s ground terminal: stray inductance from Q1's leads as well as inductance in the ground return path itself. Note that the current through C_{IN} doesn't undergo quick changes; it is equal to the ac portion of the inductor current (I1). (The battery supplies the dc portion.) A quickly changing current also flows through a portion of the loop formed when the MOSFET is off. This current (I_2) flows through both D1 and C_{OUT} as well as the copper in the ground return path, and thus the stray inductance of those components and of that ground-return path must be minimized.

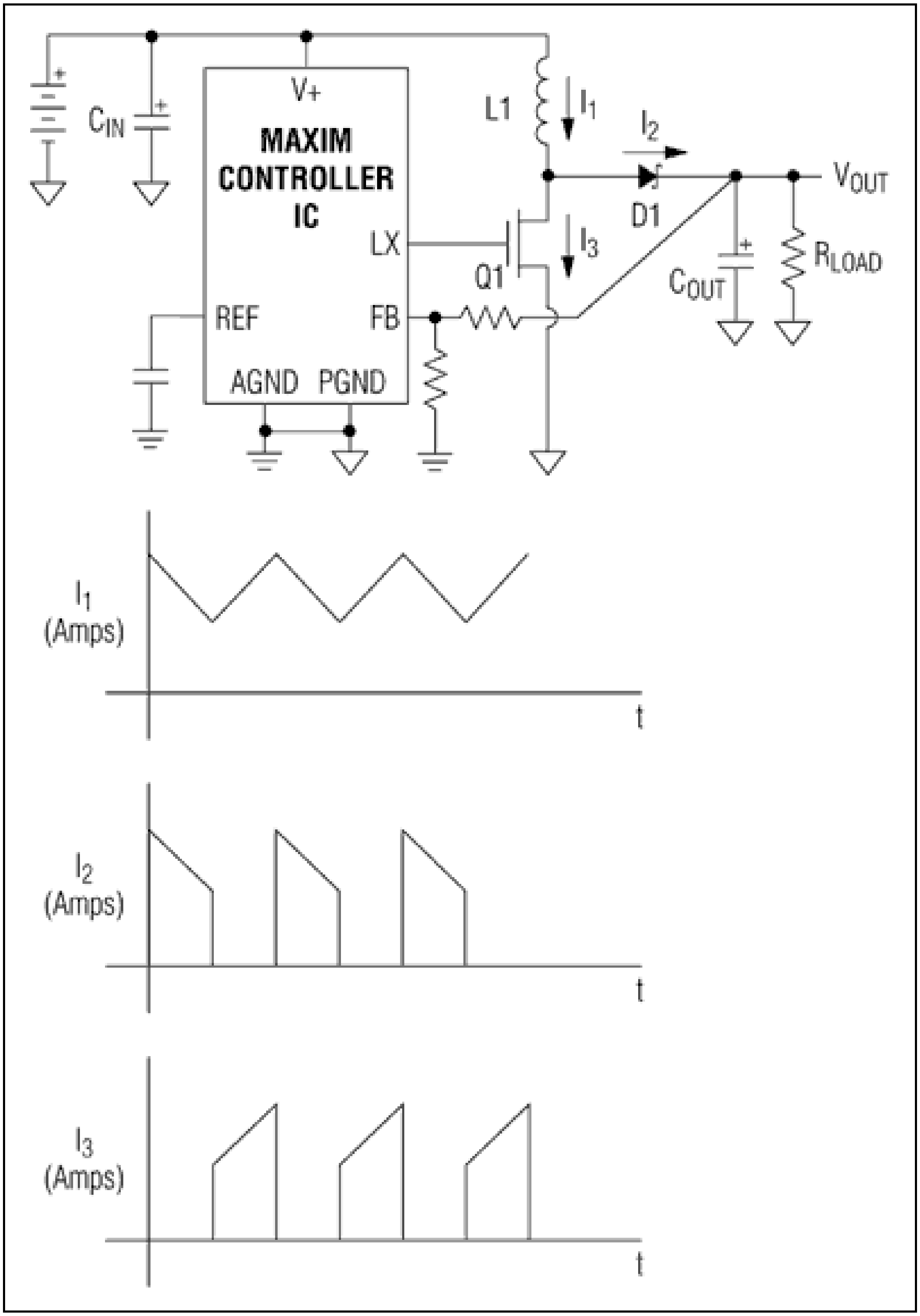

Figure 3. The current waveforms of the branches of the switching regulator circuit indicate where to minimize stray inductance. Quickly changing currents (e.g., I_2 and I_3) require that inductance in their paths be minimized.

When considering whether the inductance in the leads of the load also poses a problem, recall that if the output capacitor is large enough with a low enough ESR, its voltage remains relatively steady. That means that the current through the load resistor won't change much and therefore the inductance in series with it doesn't matter-unless the load itself changes dynamically.

Creating a Feasible Board Layout

There are a few ways to work with the ground portion of a switching regulator circuit. One is to use a single ground plane for all ground connections?a method that probably won't work very well. When that technique is used, ground currents from the power portion of the circuit could pass through the same ground path as the ground current of the resistor divider, the capacitors used to bypass certain of the controller's pins, the controller's analog ground, or all three, causing their grounds to bounce.

Probably the best approach is to create two separate ground sections-one for the power components and one for the more-quiet analog portion of the regulator. See Figure 4a. The ground portion of the power circuitry consists of the input- and output-capacitor ground terminals and the source of the MOSFET. Those connections should be made with short, wide traces. Maximizing the width and minimizing the length of the power circuitry's ground traces (and of the other power traces) improves efficiency by reducing resistance.

The analog ground section provides a ground-return path for the controller's analog ground pin, the resistor-divider ground terminal, and the ground terminals of any capacitors that bypass certain of the controller's pins (not the main input bypass capacitor C_{IN}, though). The analog ground need not be a plane. Instead, you can use long, spread- out traces because the currents are low level and relatively constant; trace resistance and inductance aren't big factors.

Connect the controller's AGND pin to the PGND pin as shown in Figure 4a. Connecting the two ground sections at these pins ensures that no switching current circulates within the analog ground. The connection between AGND and PGND can be relatively narrow, as virtually no current flows via that path. Although ideally the AGND pin would connect directly to C_{OUT}'s ground terminal, many controller ICs require that their two ground pins connect directly to each other (otherwise problems can occur if the voltage between the two pins becomes large enough to turn on the diodes that are connected between them). By making the trace from PGND to C_{OUT} short and wide, the feedback resistors and the reference inside the controller share essentially the same ground potential as the regulator's output. This fact is important because the output voltage is what these components are set up to control.

Figure 4 using separate analog- and power-ground areas, isolates the higher-amplitude power-ground currents from the quieter analog-ground currents, thus protecting the path through which those quieter currents flow.

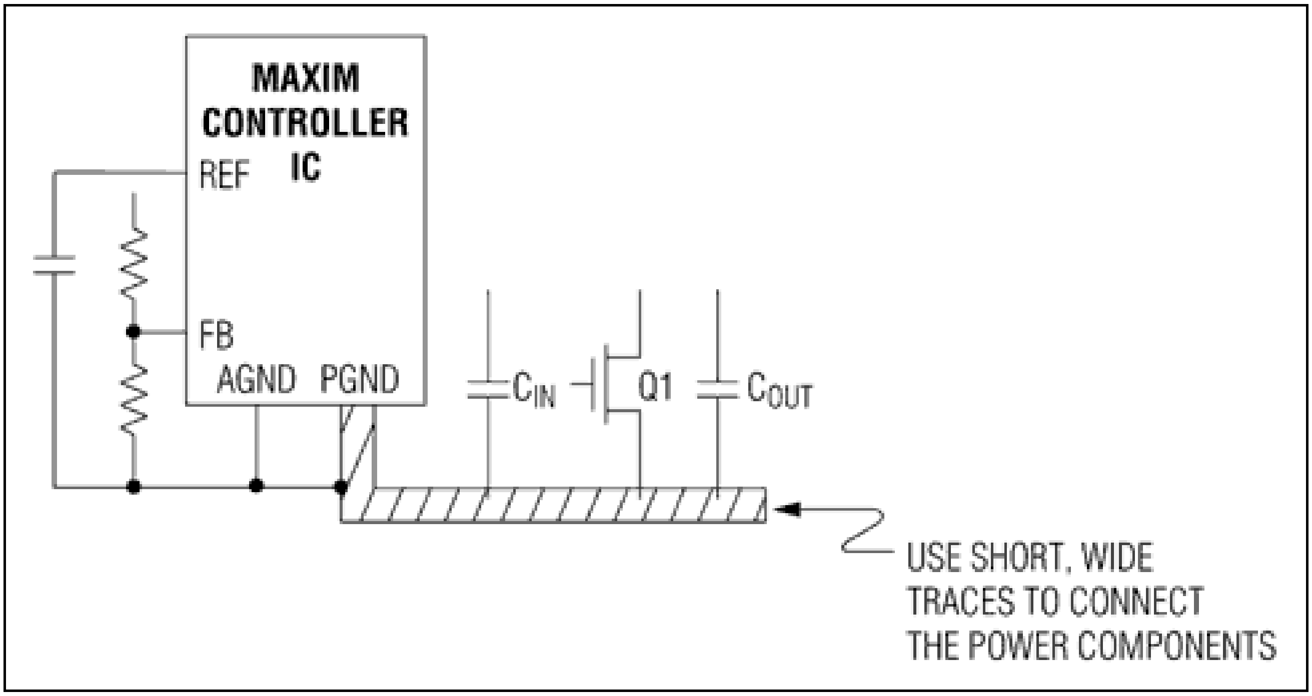

Figure 4a shows how to connect the two grounds when the controller IC includes both an AGND and a PGND pin.

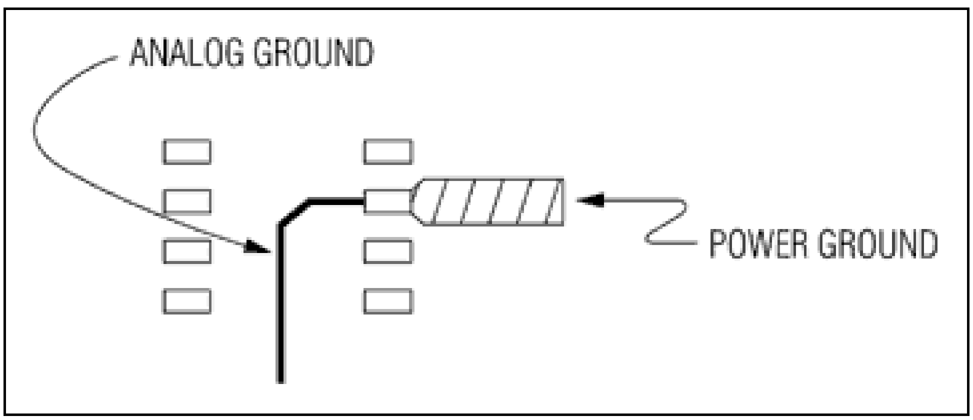

Figure 4b. When the controller includes a single GND pin, the traces can be routed in the manner shown here to prevent power current from mixing with sensitive analog current.

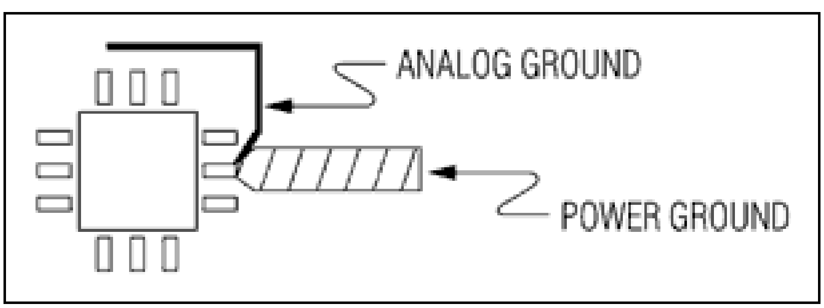

Figure 4c depicts the type of connection that might be necessary if the controller's package includes a grounded backside pad.

Sometimes there are capacitors that bypass the controller that should not be connected to the analog or the power part of ground. An R/C filter bypassing the step-up switching regulator's V+ pin (as mentioned above) is one example. In that situation, the capacitor's ground pin is too noisy for the analog ground; at the same time, the power ground is too noisy for the capacitor. You should return such a capacitor directly to the trace connecting the controller's AGND and PGND pins (or directly to the GND pin if the controller provides only one ground pin).

Finally, the number of board layers plays a part in a PCB's layout. On a multi-layer board, you can use one of the inner layers as a shield. A shield layer allows you to place components on the opposite side of the board from noisy components with little chance of interference. When incorporating a shield layer, it's generally not a good idea to connect the ground-side leads of the power components through the shield. Instead, connect them in an isolated, confined area so you know where those currents will flow and what effect they'll have.

Regardless of the number of layers, make those power-component ground connections on the top layer; doing so confines the currents to a known path that can't disturb other grounds. If that isn't possible, those connections can be made through other layers using isolated copper pieces and vias. For each connection, use multiple vias in parallel to reduce their resistance and inductance.

A similar version of this article appeared in the November 27, 2003 issue of *EDN* magazine.