National Edge

The problem of ADC and mixed-signal grounding and layout for dynamic performance while minimizing RFI/EMI

Nicholas Gray, Staff Applications Engineer

Obtaining the best performance from Analog-to-Digital Converters (ADCs) has long eluded many circuit designers. Many designers have concluded that getting the performance claimed in data sheets is not possible in practical circuits. Previous articles[1,2,3] describe methods that help improve the performance of mixed-signal circuits, particularly those using ADCs.

Inadequate grounding techniques in highspeed ADC and mixed-signal circuits and systems can lead to excessive noise when digital return (ground) currents find their way into analog circuit areas.

There are many ways to configure grounds for ADCs and other mixed-signal products and there are many opinions as to which is the best method. These opinions are generally based upon what we have found to be successful. An earlier article[1] discussed the limitations of a popular ADC-grounding technique and suggested a split ground plane, which has always yielded excellent first time noise performance. However, as good as this method is for obtaining good noise performance from highspeed ADCs, a split ground plane is not good for RFI/EMI (Radio Frequency Interference/Electro-Magnetic Interference) performance.

This article discusses how we handle grounding for a combination of good dynamic performance and minimization of RFI/EMI. The skin effect and the proximity effect are our guidelines.

The skin effect

We know that electrical current always seeks the path of lowest impedance. At relatively low frequencies this is the same as the path of least *resistance*. The path of least resistance is the path that causes the DC and low frequency current to fill the entire volume of the conductor. In a plane, the current would spread out to fill the entire volume of the plane and the resistance of that plane is dependent upon the cross-sectional area of that plane.

Once the frequency reaches a few MHz, however, the path of least impedance becomes the path of least *inductance*. As AC current flows through a conductor, the expanding and collapsing magnetic field around it is strongest at the center of the conductor and weakens with distance from that center. The result is a higher inductance at the center of the conductor that decreases with distance from the center, causing AC current to flow primarily near the outside surface of the conductor, or on its skin. This is known as the skin effect[4]. The skin effect tells us that, at high frequencies, the effective cross-sectional area of the conductor is reduced below what we would presume so that the AC resistance part of the conductor impedance is greater than we would measure with an ohmmeter. The frequency-dependent AC resistance of a flat conductor (such as a PCB trace) is defined as:

$$R_{ac} = \frac{2.61 \times 10^{-7} \sqrt{f \times \rho_r}}{2 \times (w + h)}$$

where:
R_{AC} = AC resistance, ? /inch
f = frequency, Hz

?r = conductor relative resistivity, compared to copper = 1.00
w = flat trace width in inches
h = flat trace height or thickness in inches

From this formula we see that, at 40 MHz, a typical PCB trace of 0.006 inches wide by 0.0015 inches high would have an AC resistance of 0.11? per inch. Doubling the trace thickness (height) to 0.003 inches would double the volume (cross-sectional area) of the conductor while reducing its AC resistance by only 17%. If the trace width is doubled to 0.012 inches, doubling the surface area and again increasing the trace cross-sectional area by a factor of 2, the AC resistance is reduced by 44%. So we see that the total surface area of a conductor is more important at higher frequencies than is its cross-sectional area.

The skin effect is more prominent as frequency increases, where the conductor may as well be a hollow, thin-walled conductor as far as the current is concerned (Figure 1).

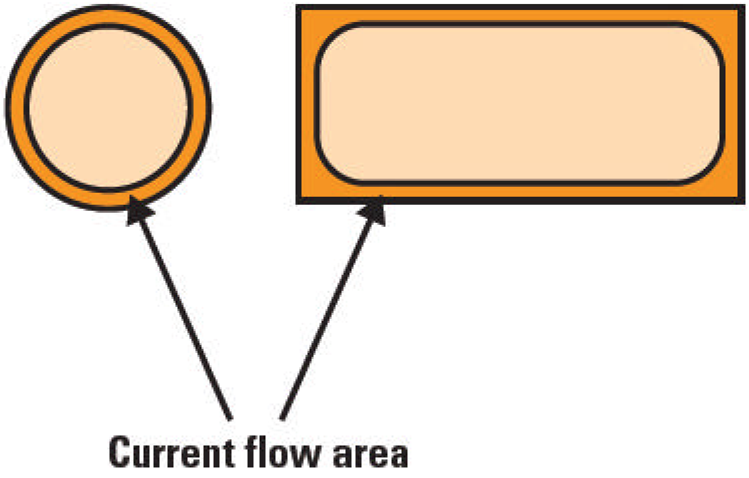

Fig. 1: Skin effect restricts current flow

The inductance of the wire or trace forces the current through it to flow only in its surface, so the effective crosssectional area of that trace is drastically reduced, raising the conductor resistance as frequency increases. This is not the only effect upon the AC resistance of a conductor.

The proximity effect

The magnetic field surrounding outgoing and return currents causes those currents to want to flow in close proximity to Current flow area each other to minimize the amount of energy required to establish and maintain that magnetic field. This causes the return current in a reference plane to flow through that plane in a path that is as close as possible to its corresponding outgoing current. The result is a flow of return current in the reference plane that is restricted to a narrow area below or above the trace carrying the corresponding outgoing current.

The reference plane current density at any point horizontally displaced from the trace carrying the outgoing current is defined[5] as:

$$I_{RP} = \frac{i}{H \times \pi \times (1 + (D/H)^2)}$$

where:
I_{RP} = the reference plane current density at horizontal distance D from the outgoing signal trace
i = the signal current
H = the height of the signal trace above the reference plane
D = the horizontal distance from the edge of the trace.

A plot of this formula reveals that the reference plane current falls off rapidly with distance from the edge of the outgoing current trace. See Figure 2. The fact that return current in a reference plane tends to flow very close to the outgoing current is known as the proximity effect[5].

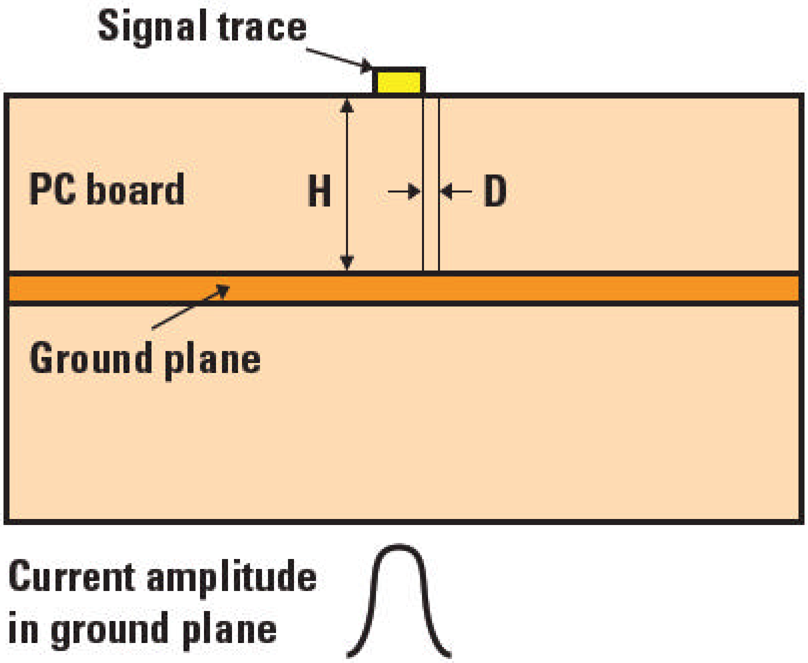

Fig. 2: Proximity effect reduces current flow

The proximity effect also causes outgoing current to flow mostly on the side of the conductor closest to the return current and the return current to flow mainly on the side of its conductor or plane closest to the outgoing conductor.

The skin effect and the two cases of the proximity effect combine to limit the current-carrying cross-sectional area of a conductor to a very small part of the full cross-sectional area of that conductor (Figure 3). Since current flows in a small portion of the conductor, the high-frequency AC resistance seen by the current is again much higher than we might expect it to be.

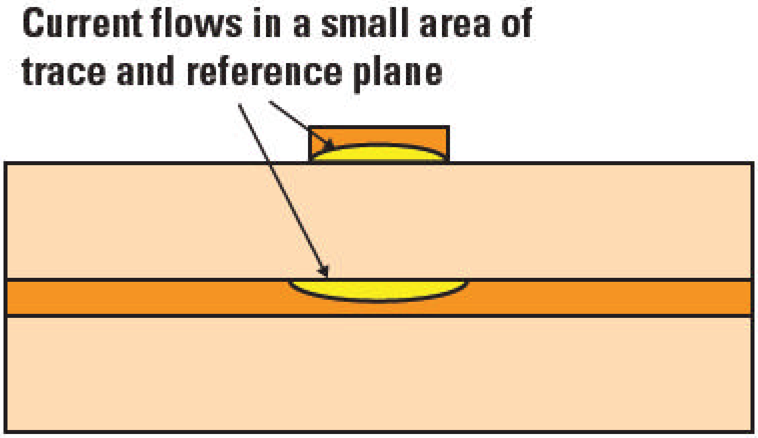

Fig. 3: Combined skin and proximity effect

Radiation

Whenever there are two conductors carrying outgoing and return currents, the amount of radiation from those conductors is a function of their length and separation. Therefore, radiation is a function of the loop area outlined by the current path. It does not matter whether one or both of these conductors is a piece of wire, a PCB trace, or a plane. Radiation increases with loop area because of the resulting increase in electromagnetic field fringing around the conductors. This loop area forms a loop antenna that radiates.

An area of copper, or other board plating metal, that is grounded at one point can also form an antenna that will radiate energy that is in the ground plane at that point. Figure 4 shows such a problem. It is best to either include a grounded via at the other end of this grounded copper or not include this area of copper at all.

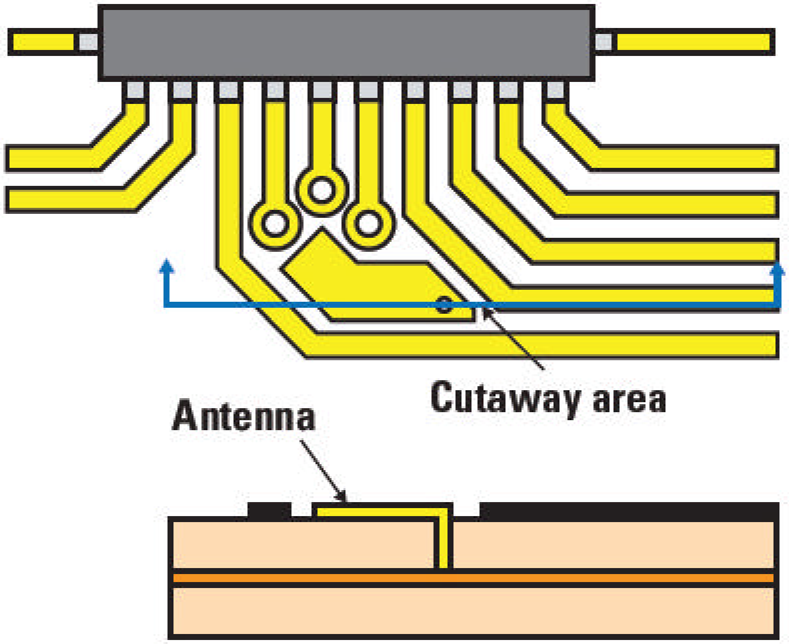

Fig. 4: Beware of ungrounded antennae

Many concepts underlie effective PCB layout design. These concepts are our guiding principles when designing the layout and grounding scheme. Our goal is to achieve the best noise performance possible while minimizing RFI/EMI.

Footnotes:

1. "Attack The Noise Gremlins That Plague High-Speed ADCs", ELECTRONIC DESIGN, December 17, 1999, p107ff
2. "Maintaining Signal Integrity Enhances ADC Circuit Performance", ELECTRONIC DESIGN, May 1, 2000, p115ff
3. "Pay Attention To The Clock And Output Bus To Improve High-Speed ADC Designs", ELECTRONIC DESIGN, June 26, 2000, p137ff
4. "High-Speed Digital Design: A Handbook of Black Magic", Howard Johnson & Martin Graham Prentis Hall, 1993, pg 151ff
5. "High-Speed Digital Design: A Handbook of Black Magic", Howard Johnson & Martin Graham Prentis Hall, 1993, pg 158

For more information visit the A/D Converters Featured Community

---