

Partitioning and Layout of a Mixed-Signal PCB

By Henry W. Ott

Component placement and partitioning, combined with routing discipline, are the keys to success in laying out a mixed-signal PCB—not the isolation of the ground planes.

A question that I often hear is: How do I prevent digital logic ground currents from contaminating my low-level analog circuitry on a mixed-signal PCB? This is a good question, without a simple answer.

In our attempt to answer this question, let's keep in mind two basic principles of electromagnetic compatibility (EMC). One is that currents should be returned to their source as locally and compactly as possible—that is, through the smallest possible loop area. The second is that a system should have only one reference plane. If we provide two references for a system, we create a dipole antenna¹. If we do not return current locally and compactly, we create a loop antenna². Both are undesirable results.

Some people suggest splitting the ground

plane on mixed-signal boards in order to isolate the digital ground from the analog ground. Although the split plane approach can be made to work, it has many potential problems—especially in large complex systems. One major problem is that you cannot route a trace over the split in the plane. If you do, you will discover that both radiation and crosstalk will increase dramatically. Traces routed across a slot or split in a ground or

power plane are one of the most common problems that I encounter as an EMC consultant.

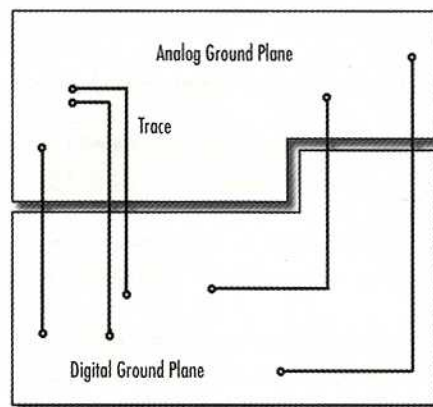


Figure 1 - Signal traces crossing over a split in the ground plane. What is the return path for the current?

If you do split the ground plane and run traces across the split as shown in **Figure 1**, what will be the current return path? Assuming that the two planes are connected together somewhere, usually at a single point, the return current will have to flow in a large loop. High-fre-

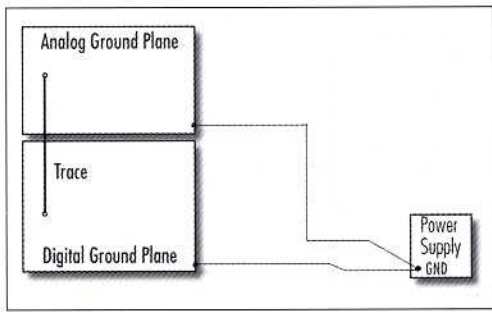


Figure 2 - A trace crossing over the split between the isolated analog and digital ground planes, which are connected together only at the power supply.

quency currents flowing in large loops produce radiation and high ground inductance. Low-level analog currents flowing in large loops are susceptible to interference. An exceptionally bad configuration is where the ground plane is split and the two planes are connected together only at the power supply as shown in **Figure 2**. In this case you will force the return current to flow all the way back to the power supply ground—a really big loop! In addition, you have made a very effective dipole antenna, consisting of the analog ground and the digital ground planes (which are at different RF potentials) connected together with long wires.

The key to determining the optimum mixed-signal board layout is to understand how and where the ground return currents actually flow. Most designers only think about where the signal current flows (obviously on the signal trace), and ignore the path taken by the return current. (Of course, the fact that many designers think this way helps to keep EMC engineers employed.)

If you must split the ground plane and

run traces across the split, you should first connect the planes together at one location, thus forming a bridge, as shown in **Figure 3**. Then, by routing all the traces so that they cross at this bridge, you will have provided a current return path directly underneath each of the traces, thereby producing a very small loop area.

Other acceptable ways of passing a signal over a split plane are with optoisolators or with transformers. In the first case, the only thing crossing the split in the plane is light; in the second case, a magnetic field. Another possibility is with a true differential signal, where the signal flows down one trace and returns on the other trace—in which case ground is not needed for the return current.

Before continuing, let's first define the basic problem that we are trying to solve. It is not that the analog circuits might interfere with the digital logic. Rather, it is the possibility that the high-speed digital logic might interfere with the low-level analog circuits. This is a legitimate concern, so we want to make sure that the digital ground currents do not flow in the analog section of the ground plane. Hence, we often hear the recommendation to isolate the analog ground plane from the digital ground plane.

To address the above concern we have to understand a little more about the characteristics of high-frequency currents. High-frequency currents want to return on a plane directly underneath the signal trace, since this is the lowest impedance (lowest inductance) path. The return current, therefore, will flow on the plane adjacent to the trace regardless of what the plane is—power or ground. The current will spread out

slightly in the plane, but will otherwise stay under the trace.

Table 1 lists the percentage of the return current contained within a distance of $\pm x/h$ from the center of the trace, where x is the horizontal distance from the center of the trace and h is the height of the trace above the plane. The actual distribution of the ground current under a signal trace is shown in **Figure 4**. From this we can conclude that the digital ground currents have no desire to flow through the analog portions of the ground plane and corrupt the

Table 1 – Percentage of ground plane current contained within a distance of $\pm x/h$ from the center of the trace	
$x/h = 2$	70 % of current
$x/h = 5$	87 % of current
$x/h = 10$	94 % of current
$x/h = 20$	97 % of current

analog signal. **Figure 5** shows a digital logic signal trace and a representation of its associated ground return current path.

Why then do we need to split the ground plane to prevent the digital current from doing something that it does not want to do in the first place? The answer is we don't! Therefore, I prefer the approach of using only one ground plane and partitioning the PCB into digital and analog sections. Analog signals must be routed only in the analog section of the board (on all layers). Digital signals must be routed only in the digital section of the board (on all layers). Under these conditions the digital return currents will not flow in the analog section of the ground plane but will remain under the digital signal trace as shown in **Figure 6**. The

A/D converters can then be positioned to straddle the partition.

You will notice from comparing **Figures 5** and **6** that the digital logic ground current follows the same path whether or not the ground plane is split. What causes problems is when a digital signal is routed in

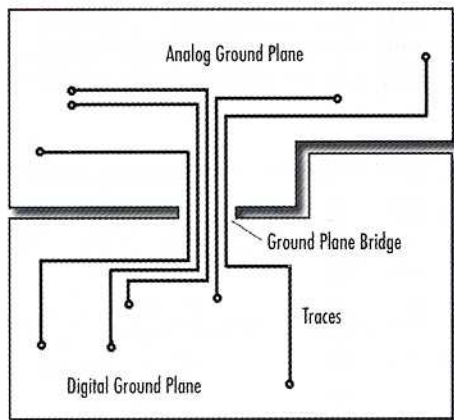


Figure 3 - Analog and digital ground planes connected together at a single location, thereby forming a bridge for the signal traces to cross over. The bridge provides the return current path.

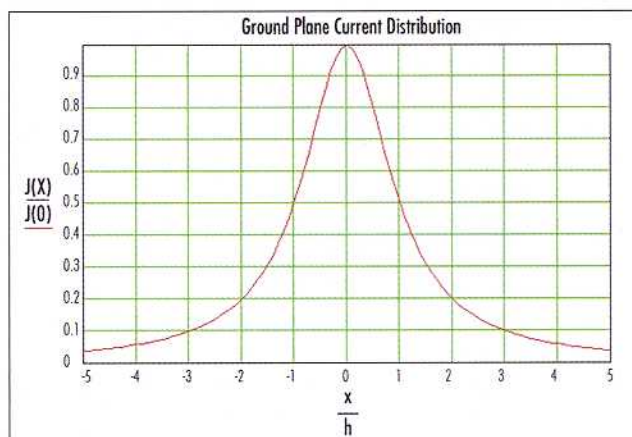


Figure 4 - Ground current distribution underneath a digital logic trace.

Component placement and partitioning are, therefore, critical to a good layout. If the layout is done properly, the digital ground currents will remain in the digital section of the board and will not interfere with the analog signals.

the analog section of the board, or vice versa. This is shown in **Figure 7**. The digital ground currents now do flow in the analog section of the board. Note, however, that the problem was not caused by the fact that we did not split the ground plane; rather, the problem was caused by the improper routing of the digital logic trace.

A PCB with a single ground plane, partitioned into analog and digital sections, and discipline in routing the signals can usually solve an otherwise difficult layout problem, without creating any additional

problems caused by a split ground plane. Component placement and partitioning are, therefore, critical to a good layout. If the layout is done properly, the digital ground currents will remain in the digital section of the board and will not interfere with the analog signals. The routing, however, must be checked carefully to assure that the above mentioned routing restrictions are adhered to one hundred percent! It only takes one improperly routed trace to destroy an otherwise perfectly good layout.

Another problem occurs when we consider where and how to connect the analog and digital ground pins of the A/D converter. Most A/D converter manufacturers, while suggesting the use of split ground planes, state something similar to the following in their data sheets or application notes: "The AGND and DGND pins must be

connected together externally to the same low impedance ground plane with minimum lead length³. Any extra external impedance in the DGND connection will couple more digital noise into the analog circuit through the stray capacitance internal to the IC." Their recommenda-

tion is then to connect both the AGND and the DGND pins of the A/D converter to the analog ground plane. This approach has the potential of creating a number of additional problems. One of these is where do you

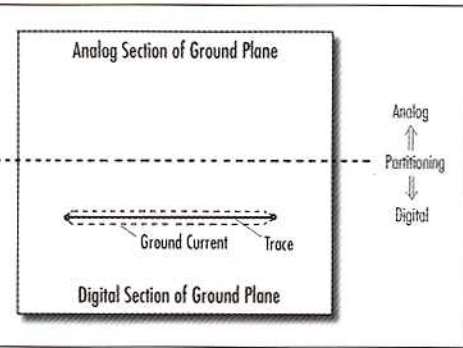


Figure 6 - A digital logic trace and its associated ground current—on a partitioned, single ground plane, mixed-signal board.

connect the ground side of the digital decoupling capacitor—the analog plane or the digital plane?

The above requirement can easily be satisfied if the system contains only one A/D converter. You could split the ground plane and connect the analog and digital sections together at one place, under the A/D converter as

shown in **Figure 8**. If you use this approach, you should make the bridge between the two ground planes as wide as the IC, and remember that no traces can be routed across the split in the plane.

What do you do if instead of one A/D converter, your system has 10 A/D converters? If you connect the analog and digital ground planes together under each converter, the planes are connected together at multiple points and are no longer isolated. If you don't connect the planes together under each

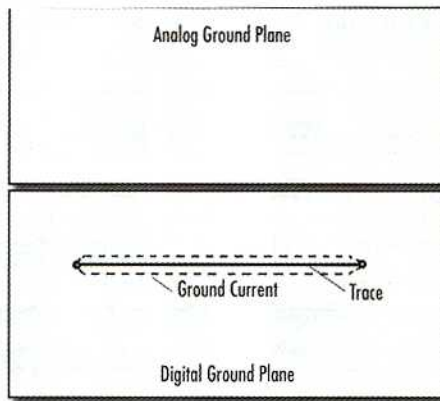


Figure 5 - A digital logic trace and its associated ground current—on a split ground plane, mixed-signal board.

ing AGND and DGND pins together, and not create additional problems in the process, is to use only one ground plane to begin with. The ground plane can then be partitioned into analog and digital sections as shown in **Figure 9**. This layout satisfies the IC manufacturer's requirement of connecting the analog and digital ground pins together through a low impedance, as well as meets the EMC concerns of not creating any unintentional loop or dipole antennas.

If you are still skeptical about using a single ground plane on your mixed-signal PCBs, I suggest you do the following experiment. Lay out the board with a split ground plane, but provide means for connecting the two planes together at intervals of 1/2-inch with jumpers or zero Ohm resistors. Partition and route the board properly, with no digital traces (on any layer) over the analog plane and no analog traces (on any layer) over the digital plane. In addition, no traces can be routed across the split in the ground or power planes. Build the board and test its functionality and EMC performance. Then connect the planes together with zero Ohm resistors or jumpers, and test the board again for functionality and EMC performance. I think you will find that in almost all cases, both the functional perfor-

converter, the device manufacturer says that you must connect both the analog and digital ground pins to the analog plane. Are you sure that is what you want to do? I'm not!

A much better way to satisfy the above requirement of connect-

ing AGND and DGND pins together, and not create additional problems in the process, is to use only one ground plane to begin with. The ground plane can then be partitioned into analog and digital sections as shown in **Figure 9**. This layout satisfies the IC manufacturer's requirement of connecting the analog and digital ground pins together through a low impedance, as well as meets the EMC concerns of not creating any unintentional loop or dipole antennas.

formance and the EMC performance of the board will be better with the single ground plane. Should split ground planes ever be used? I can think of at least three instances where they would be appropriate. Some

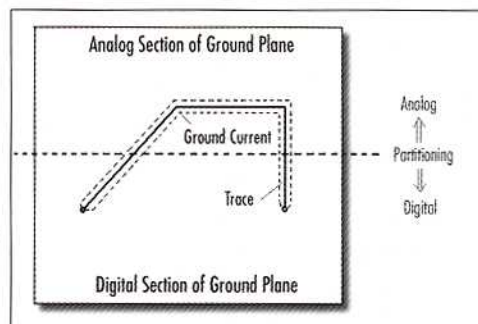


Figure 7 - An improperly routed digital logic trace. Logic ground current now flows in the analog section of the ground plane.

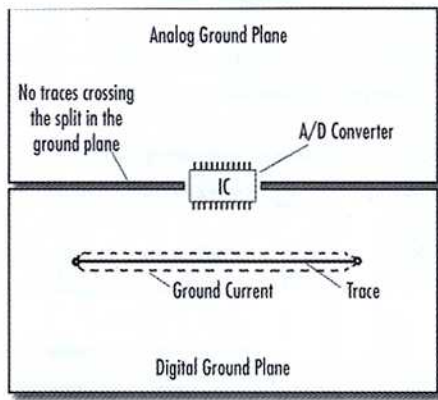


Figure 8 - An acceptable layout of a mixed-signal board with a single A/D converter and a split ground plane.

medical equipment with very low leakage requirements between the power mains and the portion of the circuit connected to the patient, some industrial process control equipment where the outputs are connected to very noisy, high power electromechanical equipment, and possibly when a PCB is improperly laid out.

In the first two cases listed above, signals that cross the split in the ground plane are usually optical or transformer coupled, thereby satisfying the requirement of no traces crossing the split in the ground plane. The last case, however, is of more interest for our present discussion. It can clearly be demonstrated that if a PCB is poorly laid out, its performance can often be improved by using a split ground plane.

Consider the situation that was shown in Figure 7, where a high-speed digital trace was routed over the analog section of the board (a clear violation of the partitioning rules). Since the digital return current flows under the signal trace, it flows in a portion of the analog ground plane. Splitting the ground plane in this case will improve the performance of the PCB by containing the

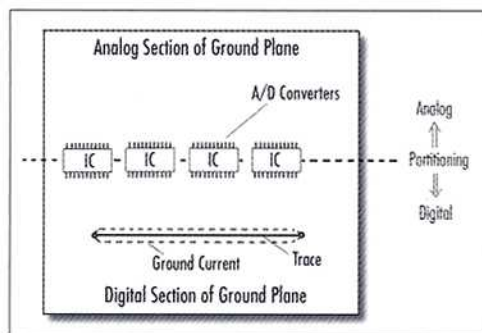


Figure 9 - A properly partitioned mixed-signal board with multiple A/D converters and a single ground plane.

digital return current to the digital ground plane as shown in **Figure 10**. The real problem, however, is the improper routing of the high-speed signal trace. A better solution would have been to route the digital signal properly in the first place and not split the ground plane.

On a mixed-signal PCB it usually is desirable to separate analog and digital power. Therefore, split power planes can and should be used. However, no trace on any layer adjacent to the power plane can cross over the power plane split. All traces crossing over the power plane split must be on a layer adjacent to the solid ground plane. In some circumstances a split power plane can be avoided by running the analog power as a trace rather than as a plane.

Mixed-signal design checklist

- Partition your PCB with separate analog and digital sections.
- Place components properly.
- Straddle the partition with the A/D converters.
- Do not split the ground plane. Use one solid plane under both analog and digital sections of the board.
- Route digital signals only in the digital section of the board. This applies to all layers.
- Route analog signals only in the analog section of the board. This applies to all layers.
- Separate analog and digital power planes.
- Do not route traces over the split in the power planes.
- Traces that must go over the power plane split must be on layers adjacent to the solid ground plane.
- Think about where and how the ground return currents are actually flowing.
- Use routing discipline.

Remember the key to a successful PCB layout is partitioning and the use of routing discipline, not the isolation of ground planes. It is almost always better to have only a single reference plane (ground) for your system.

Footnotes

1. The magnitude of the radiation from a small dipole antenna is proportional to

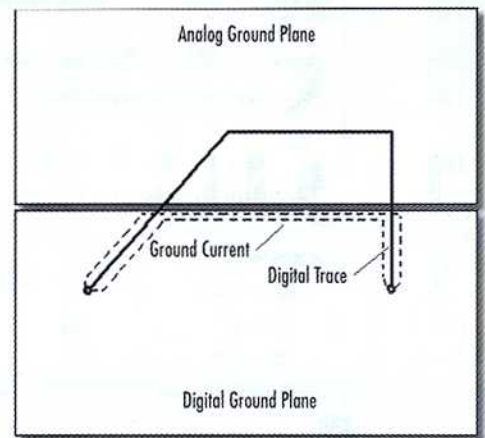


Figure 10 - A mixed-signal board with a split ground plane and an improperly routed digital logic trace. Note that the split ground plane confines the logic ground current to the digital ground plane; however, the real problem is the improperly routed digital trace.

the length of the wire, the amount of current in the wire, and the frequency.

2. The magnitude of the radiation from a small loop antenna is proportional to the area of the loop, the amount of current in the loop, and the frequency squared.
3. The reason for this is because most A/D converters do not have the analog and digital grounds connected together internally. Therefore, they have to rely on this external connection between the ground pins to provide this connection.

Henry W. Ott is president and principal consultant of Henry Ott Consultants, an EMC consulting and training organization located in Livingston, NJ. Prior to starting his own business, he was a distinguished member of the technical staff at AT&T Bell Labs. Ott has over 30 years experience in the EMC field, and he holds an M.S.E.E degree from New York University.

Web Resources

- Henry Ott Consultants, www.hottconsultants.com
- High Speed Digital Design, www.sigcon.com
- UltraCad, www.ultracad.com
- Analog Dialogue — Ask the Application Engineer, www.analog.com/publications/magazines/Dialogue/Anniversary/1.html