Henry Ott Consultants

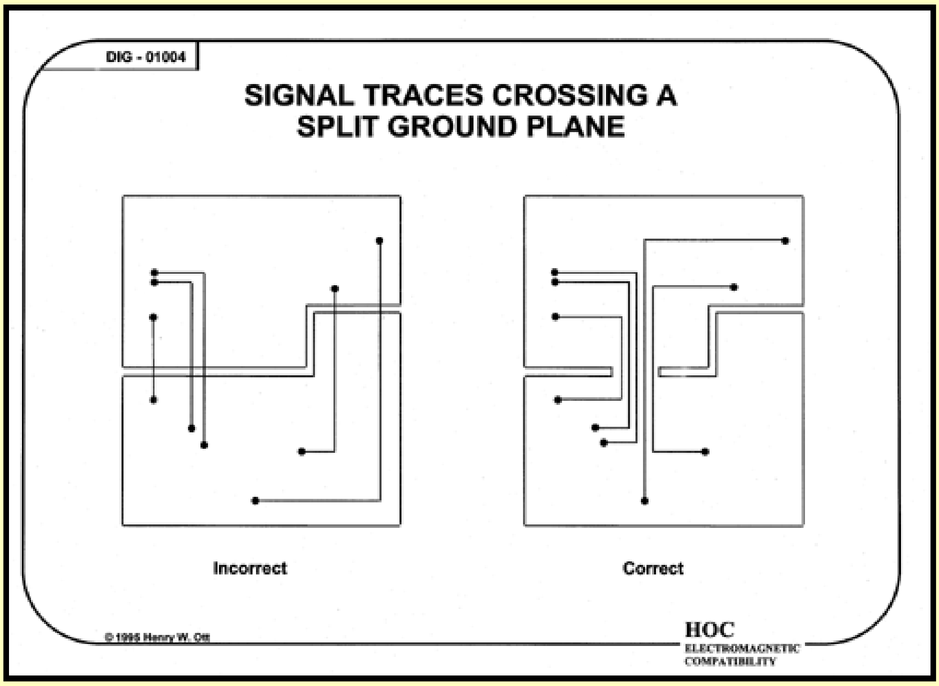
Electromagnetic Compatibility Consulting and Training

Grounding of Mixed Signal PCBs

A question that I hear often is: How do I prevent digital logic ground currents from contaminating my low level analog circuitry? This is a good question without a simple answer. Most A/D converter manufacturer's data books and application notes provide little if any useful information on the subject. If they do provide information, it is usually only applicable to a simple system containing only one A/D converter.

Some people suggest splitting the ground plane in order to isolate the digital ground currents from the analog ground currents. Although the split plane approach can be made to work, it has many potential problems especially in large complicated systems. Can you list some of these problems? One of the major ones is that you can not route a trace over the split in the plane (see Tech Tip **Slots in Ground Planes**). It is always better to have only a single reference plane for a system.

If you do split the ground plane and run traces across the split (left hand figure below), there will be no return path near the trace and the current will have to flow in a big loop. Current flowing in big loops produce radiation and high ground inductance. If you must split the ground plane and run traces across the split, you should do it as shown in the right hand figure below. By connecting the planes together at one point (a bridge) and routing all the traces so that they cross at this bridge point, you will have provided a return path for the current directly underneath each of the traces (hence a very small loop area).



High frequency digital ground return currents want to return directly underneath the signal trace. This is the lowest impedance (lowest inductance) path. The digital ground currents have no desire to flow through the analog portion of the ground plane and corrupt your analog signal. Why then do we need to split the ground to prevent the digital current from doing something that it does not want to do anyhow? Therefore, I prefer the approach of using only one ground plane and **partitioning** the PCB into digital and analog routing sections. Analog signals can then be routed only in the analog section of the board (on any layer), and digital signals can be routed only in the digital section of the board (on any layer). What causes problems is when a digital signal is routed in the analog section of the board, or visa versa.

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 the additional problems caused by a split ground plane. 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! The key to a successful mixed signal PCB layout, therefore, is **proper partitioning** and **routing discipline**, not a split ground plane.

Many A/D converter manufacturers, while suggesting the use of split ground planes, state 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 length. 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 recommendation is 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. Can you list some of these problems? What do you connect the ground side of the digital power decoupling capacitor to? The analog plane or the digital plane?

A much better way to satisfy the requirement of connecting AGND and DGND pins together through a low impedance, and not create additional problems in the process, is to use only one ground plane to begin with.

The key to determining the optimum board layout is to think, **how and where do the return currents flow?**

If you are still skeptical about using a single ground plane on your mixed signal boards I suggest you do the following experiment. Layout the board with a split ground plane, but provide means for connecting the two planes together every 1/2 inch with jumpers or zero ohm resistors. 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. Build the board and test it's functionality and EMC performance. Connect the planes together and test the board again for functionality and EMC performance. I think that you will find that in almost all cases, both the functional performance and the EMC performance of the board will be better with the single ground plane. If you do the experiment send me an e-mail letting me know of your results.

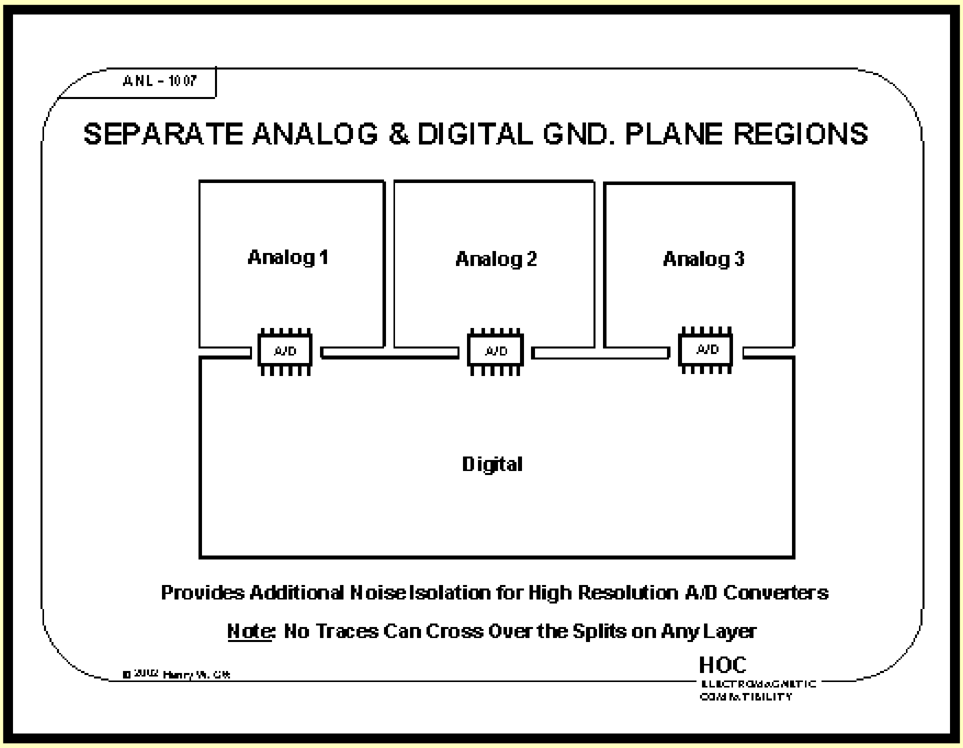
It is almost always better to have only a single reference plane for a system!

Analog ground plane noise voltages should be kept smaller than the minimum analog signal level of concern. In the case of an A/D (or D/A) converter the smallest resolvable signal voltage level [least significant bit (LSB)] is a function of the number of the bits and the full scale reference voltage of the A/D converter. The smaller the reference voltage and the larger the number of bits, the smaller the minimum resolvable signal voltage will be. The following table shows the resolution versus the number of bits for an A/D converter using a one volt reference. These resolution levels can be scaled for other reference voltages by multiplying the resolution by the appropriate factor. For example, if the converter uses a 5 volt reference then multiply the resolution numbers in the table by five.

Number of Bits	Resolution (LSB)
8	4 mV
10	1 mV
12	240 uV
14	60 uV
16	15 uV
20	1 uV
24	0.06 uV (60 nV)

The use of a single solid ground plane properly partitioned and routed (as discussed above) is usually adequate for most low to moderate resolution A/D converters (8, 10, or 12 bit). For higher resolution systems (14 bits and up) even more ground noise voltage isolation may be required for adequate performance. These converters have resolution voltages in the tens of microvolts, or less, level. In this case you might want to divide your board into separate isolated analog and digital ground plane regions, each solidly connected to the digital ground plane under each of the A/D converters as shown in the figure. This approach will provide additional ground noise isolation for the high resolution A/D converters while still maintaining a single ground plane for the system.

Notice that even in this case, the ground plane is not split -- it is all connected together. Also remember that no traces, on any layer, can cross over the isolating slots in the ground plane.



Summary

- **Partition** your PCB with separate analog and digital sections.
- **Do not** split the ground plane. Use one solid ground 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.
- The key to a successful PCB layout is the use of **routing discipline**.
-

 Henry Ott Consultants, 48 Baker Road Livingston, NJ 07039 (973) 992-1793

Bibliography

Ott, H. W., Partitioning and Layout of a Mixed Signal PCB, Printed Circuit Design, June 2001.

Correspondence

- Thank you, this article will assist me in a proposal I made earlier this week in a design review on how to properly layout and partition a PCB with analog and digital signals. As you can probably imagine I was met with major opposition when I proposed utilizing a bridge to connect the analog and digital sections. I have used this technique on other PCB's (different Business Unit) and had no problems although I never had the opportunity to conduct the experiment and analyze the data as you mention in your article, thank you very much. [e-mail from an engineer in Michigan]
- I am in the process of laying out a mixed-signal board myself and your article has provided some useful information. Thinking of the issue in terms of return current flow makes a lot of sense. [e-mail from a RF Engineer in California]
- Your tech tip "Grounding of Mixed Signal PCBs" was the most clear and concise presentation of the material that I found. Everyone else suggest separate planes connected at a single point but neglect to consider "big loops" as you suggest. Since you so clearly articulated (i.e., helped me understand the

real issue), I am planning on purchasing your book to hopefully better understand EMC design issues. [e-mail from an engineer in Arizona]

- I am busy designing a mixed signal board incorporating two 16-bit DACs and a high-speed digital side, and found your website very enlightening. It's amazing how much info there is out there by supposed "experts" that is clearly outdated or not very well thought through, and your website provided a welcome change. [e-mail from a post-graduate student in South Africa]

Return to top of page.

Return to HOC home page.

Henry Ott Consultants
48 Baker Road Livingston, NJ 07039
Phone: 973-992-1793,   FAX: 973-533-1442

March 25, 2004