本文描述如何设计混合信号的印刷电路板。混合信号印刷电路板包括模拟电路和数字电路，这些不同类型的电路经常为印刷电路板设计人员和设计工程师造成问题。模拟电路和数字电路有不同的额定功率、电流、电压引出和散热要求。模拟电路的功率和电流相比之下高得多，而且更容易制造噪音。最好的方法是在不同层次布置模拟迹线和数字迹线。如果在两种电路之间没有平面分隔，经验之谈就是使两种电路相对垂直，以保持信号可靠性。本文提供如何最好地分隔模拟电路和数字电路的提示。

In The Mix: Tips For MIXED-SIGNAL DESIGN

Avoiding preconceived notions is a must if you want to design efficient mixed-signal boards.

by SYED RIZVI and IMRAN KHAN

In many cases, engineers have preconceived notions about mixed-signal designs and how analog and digital placement, partitioning and associated design should be performed. However, the truth of the matter is that every mixed-signal board design is different and demands interactive design negotiations between expert mixed-signal designers and the OEM customer.

When laying out components for a mixed-signal PCB, certain considerations are critical to achieving optimum performance. Mixed-signal is particularly tricky to design since analog devices possess different characteristics compared to digital components: different power rating, current, voltage callouts and heat dissipation requirements, to name a few. Consequently, it is important for design engineers to recognize certain problematic areas and then implement them correctly in their mixed-signal designs.

Circuitry placement on mixed-signal PCBs is among the more crucial design points. First, there's the wrong way. As shown in FIGURE 1a, there is a separation between the analog and digital ground planes, appearing as an L- or S-shaped figure, with traces crossing that separation. This is an example of poor layout due to the trace discontinuity in the plane separation.

What's missing is a respective solid plane located underneath the trace to allow for continuous impedance on that trace. Also, the crossover of the traces from analog to digital plane prevents a good return path for these particular traces. The idea is to maintain a distinct separation of split planes between the analog and digital ground planes and to keep analog traces under the analog split plane and digital traces under the digital split plane. Otherwise, noise is created on the board and signals deteriorate.

Ideally, the analog section of a board must be totally isolated in terms of placement, routing and plane creation. Analog traces should be run only underneath their analog reference power or ground plane. Conversely, digital traces should run only under the digital section with respective power and ground planes, as seen in FIGURE 1b. Hence, impedance is kept constant, and there's a good return path for signals.

However, there are worst-case scenarios involving mechanical and/or space constraints. The best approach here is to run analog and digital traces on separate layers. If there is no plane separation between them, the rule of thumb is to run the analog and digital traces perpendicular to each other to maintain signal authenticity. They should never be run parallel to each other.

FIGURE 1c shows the best placement method, using an analog-to-digital converter (ADC) IC as an example. One side of the IC has analog pins resting directly atop the analog plane, while the other side's digital pins are squarely on the digital plane.

When separation isn't as exact and ideal as the above ADC IC example, mixed-signal circuitry separation at the device level requires careful design techniques. Take, for example, a mixed-signal device packaged in a BGA. FIGURE 2 shows proper separation between analog and digital sections. Analog and digital

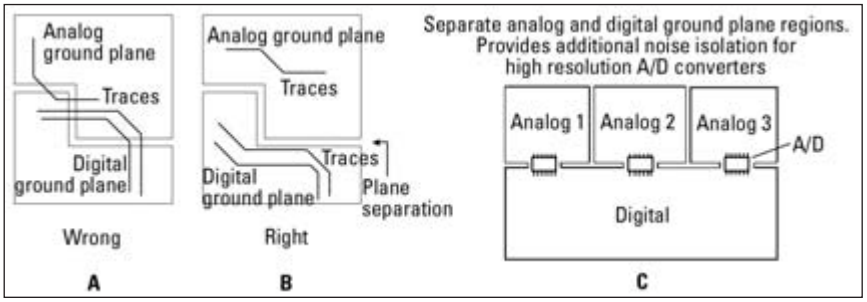

FIGURE 1. a) Signal traces cross over a split in the ground plane and it is void of a return path for the current. b) Analog and digital traces are routed on top of their respective analog and digital ground planes without crossing over the separation between the two planes. c) Layout has a single ADC IC and a split ground plane.

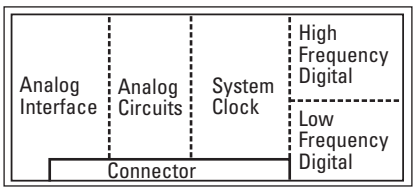

FIGURE 2. A close-up view of a BGA packaged mixed-signal device shows the required separation between the analog and digital power supplies.

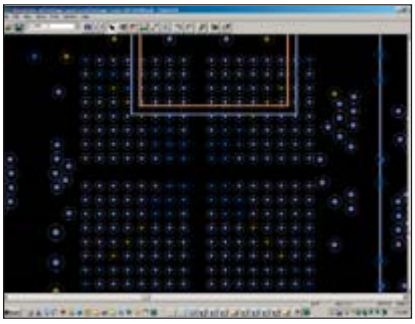

FIGURE 3. This ideal partitioning of a mixed-signal board shows analog components placed at the left and digital components on the right.

circuitry in the device is isolated with about a .015″ wide gap so that all analog signals and traces are are on the analog side and likewise on the digital side.

Correct separation is critical due to the different methodologies associated with analog and digital sections of the board. The analog section normally has higher voltage and current requirements, which in turn generate more heat that needs to be properly dissipated. Always keep in mind that analog circuitry is high-power and high-current, inherently creating more noise than digital. Consequently, this generated noise can adversely affect adjoining low-power and low-current digital circuitry if proper partitioning between the two isn't correctly implemented.

An example of effective partitioning places power supplies, analog interface converters and other analog circuits on the left side of the PCB. High- and low-frequency digital devices are placed on the right side with system clocking in the middle and connectors at the bottom edge as shown in **FIGURE 3**. In this way, analog and digital are totally separated, and traces can be easily routed separately underneath the return path, ground and power planes.

Also, since digital signals operate at high frequencies, they are sensitive to noise. Correct placement and routing of the digital section thus becomes important to completely isolate it from

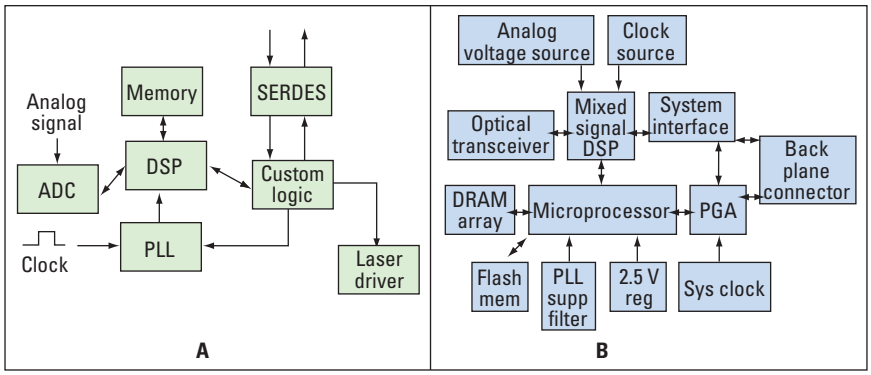

FIGURE 4. a) Even with a simple mixed-signal board layout, the smallest analog portion must be completely separated from digital components. b) A mixed-signal DSP on a board requires the same separation of analog power and ground planes from the digital power and ground planes.

noisy analog circuitry. And the clock is the most critical section in a mixed-signal PCB design. In this regard, ground shielding to protect clock traces or critical digital signals must be performed if they run through analog circuitry when those paths can't be avoided. It's best to run a clock net and then have a ground trace shielding the entire path of the clock trace.

FIGURE 4 shows examples of mixed-signal board layouts, with **FIGURE 4a** the more simple of the two. It includes several digital components, DSP, phase-locked loop (PLL) programmable logic, memory, serializer/deserializer (SerDes) logic and termination resistors. The only analog component is an ADC IC. As discussed earlier, it is important to keep signals and planes separate between analog and digital, regardless how small the analog portion.

An OC48 interface card layout (**FIGURE 4b**) contains a number of digital devices and appears to be a simple layout like the one mentioned earlier. However, in this case, the PCB design calls for a mixed-signal DSP. This device possesses analog and digital circuitry, thus proper design techniques must be implemented to provide each section the required power, allowing them to effectively perform their functions. In this case, depending on package pin assignment, partitioning the analog and digital circuit is performed in the middle of the power plane underneath the DSP chip.

Care in the layout process should be observed so that the analog and digital planes do not overlap each other. For example, it's important that a digital power plane does not overlap an analog power plane. That will produce capacitance between the overlapping areas, which is likely to cause RF emissions to pass from one plane to another (**FIGURE 5**).

Since analog circuitry is more demanding to work with than its digital counterpart, you must be ready to make certain design compromises that come down to cost versus benefits. You must first recognize that mixed-signal designs have certain characteristics that must be identified, and then you must implement them to have a successful design.

Correct component placement is of utmost importance. This is the determining factor in ensuring how well analog circuitry signals flow through the PCB, as well as how the planes are split to keep analog characteristics separate from the digital section. PCD&M

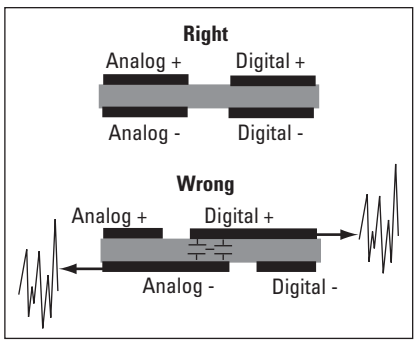

FIGURE 5. Overlapping analog and digital planes could cause unwanted capacitance between the two planes.

IMRAN KHAN (imran_khan@nexlogic.com) has been a PCB design engineer at Nexlogic for over four years. SYED RIZVI (syed@nexlogic.com) has eight years of experience as a PCB design engineer, with over four at Nexlogic.