Products Solutions Design Buy Support

Maxim > Design Support > Technical Documents > Application Notes > Audio Circuits > APP 4079

Keywords: layout, pcb, rf immunity, tdma noise, rf susceptibility, emc

APPLICATION NOTE 4079

Minimize Noise in Audio Channels with Smart PCB La

By: Adrian Rolufs
Sep 04, 2007

Abstract: This application note discusses several factors that affect audio functionality in a cell-phone PCB d article shows examples of a problematic and a well-designed PCB for a cell phone. The differences between layouts are discussed, with emphasis given to design considerations that improve audio function.

Introduction

For PCB layout engineers, cell phones provide the ultimate challenge. Each subsystem has conflicting requirements, and modern cell phones include nearly every subsystem found in a portable device. A well-designed PCB must both maximize the performance of each device connected to it and prevent the various subsystems from interfering with each other. Inevitably, the conflicting requirements of each subsystem result in some compromise. Finally, although audio functionality in cell phones is increasing, the audio circuitry is often given the least consideration during PCB design.

Component Placement

The first step of any PCB design is choosing where to place the components. This task is called "floor planni component placement can ease signal routing and ground partitioning. It minimizes noise pickup and the boa required.

Cell phones contain a mixture of digital and analog circuitry that must be separated to prevent noise from the portion from interfering with the sensitive analog circuits. Partitioning the PCB into a digital and an analog re simplifies the separation task.

The RF section of a cell phone is typically considered analog. Yet there is a common problem in many cell-p designs where noise coupled from the RF section into the audio circuitry is demodulated into audible noise. ˙ this, the RF and audio sections should be separated as much as possible.

Once the PCB has been partitioned into analog, digital, and RF sections, the component placement within th section must be selected. Components should be placed to minimize the distance that audio signals travel. L audio amplifier as close to the headphone jack and loudspeaker as possible. This positioning will minimize E from Class D speaker amplifiers, and minimize the noise susceptibility of low-amplitude headphone signals. devices supplying the analog audio as close to the amplifier as possible to minimize noise pickup on the amp All input signal traces will act as antennas to RF signals, but shortening the traces helps reduce the antenna frequencies typically of concern.

Example Component Placement

Figure 1 shows an example of poor audio component placement. The most serious problem is that the audi placed quite far from the audio sources. This distance increases the chances of noise coupling because the more prone to pass near noisy digital circuitry. The long traces can also be efficient RF antennas. In cell pho GSM technology, these antennas can pick up the GSM transmission and feed the signal into the audio ampl all amplifiers will demodulate the 217Hz envelope to some degree and generate unwanted noise on the outp worst case, this process causes noise on the output that completely overwhelms the desired audio signal. M input trace lengths thus prevents the signal from ever reaching the amplifier.

There is another problem with the component placement in Figure 1: the amplifier is not placed near the spe headphone jack. If the speaker amplifier is Class D, then the long speaker traces increase the EMI radiation amplifier. This radiation could potentially prevent the device from passing government-mandated testing. The headphone and speaker output traces both increase the trace resistance, thus decreasing the power deliver load.

Finally, since the components are so spread apart, the traces connecting the components will be routed nea subsystems in the phone. Not only does this distance increase the difficulty of routing the traces, but it also i difficulty of laying out other parts of the phone.

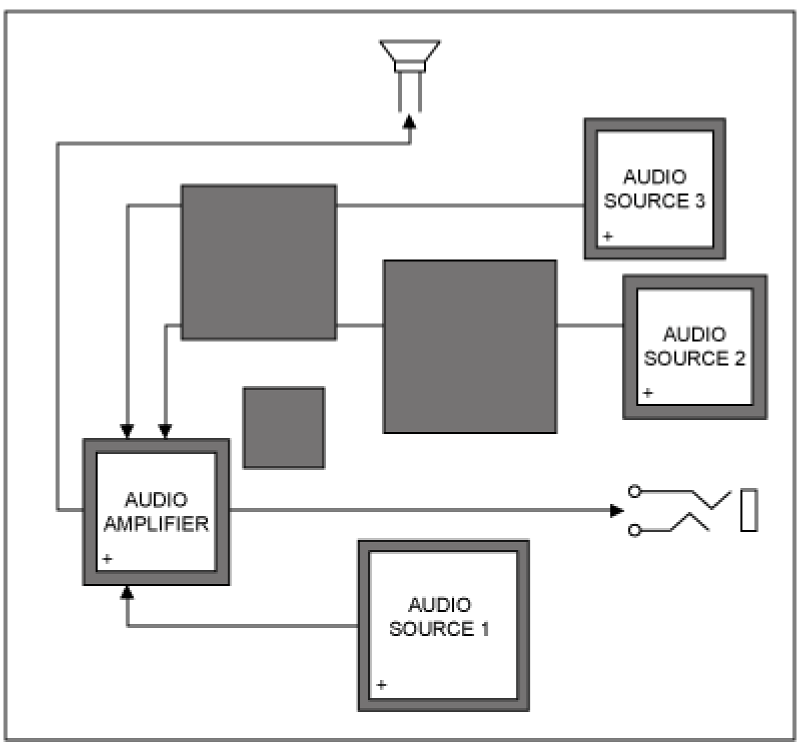


Figure 1. Example of poor component placement in a cell phone.

Figure 2 shows the same components as Figure 1, but rearranged to use the space more effectively and to trace lengths. Notice how all the audio circuitry has been partitioned to be near the headphone jack and the :

audio input and output traces are much shorter and the nonaudio circuitry has been moved to a different par
This design will have lower overall system noise, be less susceptible to RF interference, and be easier to lay

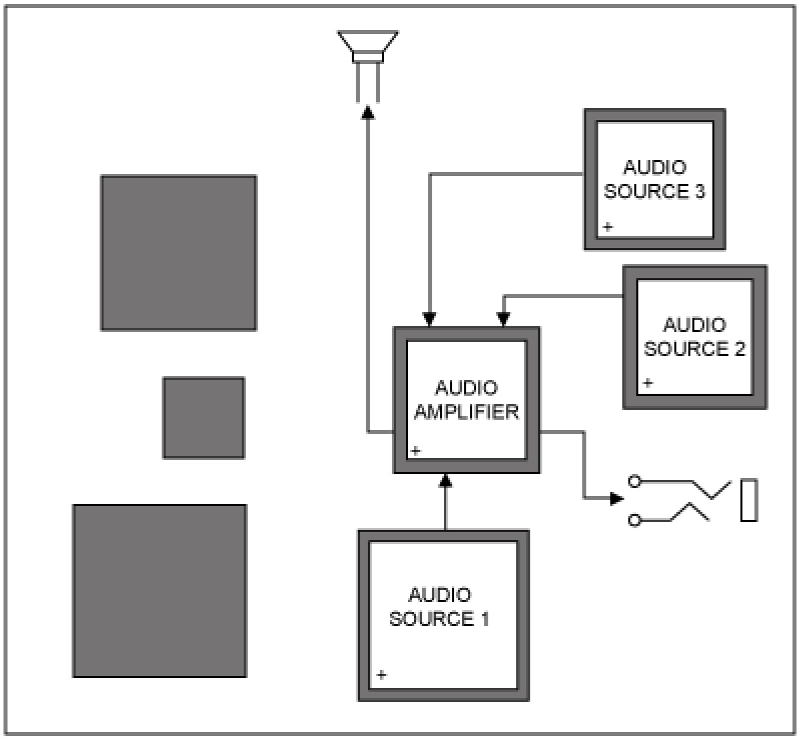


Figure 2. Example of good component placement in a cell phone.

Signal Routing

When considering noise and distortion on the audio output, signal routing typically has limited impact. There nonetheless, some steps to ensure that performance is not compromised.

Loudspeaker amplifiers typically are powered directly from the main system voltage and require relatively hig
Resistance in the trace will result in voltage drops that reduce the supply voltage of the amplifier and waste p
system. The trace resistance also causes the normal fluctuations in supply current to convert to fluctuations
To maximize performance, use short wide traces for all amplifier power supplies.

Differential signaling is an advantage that should be exploited whenever possible. Differential inputs provide immunity by rejecting any signal that is common to the positive and negative signal lines. There are several considerations to ensure that the differential amplifier is effective. Specifically, it is important that the differen pairs have the same length and the same impedance. The signal pairs should be routed as close to each oth possible to ensure that they pick up the same noise. Differential inputs on amplifiers are particularly effective noise from the digital circuits in the system.

Grounding

Grounding plays the single, most significant role in determining whether the device's potential is achieved by
A poorly grounded system will likely have high distortion, noise, crosstalk, and RF susceptibility. Although on question how much time should be devoted to system grounding, a carefully designed grounding scheme pr large number of problems from ever occurring.

The ground in any system must serve two purposes. First, it is the return path for all currents flowing to a dev it is the reference voltage for both digital and analog circuits. Grounding would be a simple exercise if the vol points of the ground could be the same. In reality, this is not possible. All wires and traces have a finite resist means that whenever there is current flowing through the ground, there will be a corresponding voltage drop wire also forms an inductor. This means that whenever current flows from the battery to a load, and back to t the current path has some inductance. The inductance increases the ground impedance at high frequencies.

While designing the best ground system for a particular application is no simple task, some general guideline
to all systems.

1. **Establish a Continuous Ground Plane for Digital Circuits**
   Digital current in the ground plane tends to follow the same route that the original signal took. This path smallest loop area for the current, thus minimizing antenna effects and inductance. The best way to ens digital signal traces have a corresponding ground path is to establish a continuous ground plane on the immediately adjacent to the signal layer. This layer should cover the same area as the digital signal trac as few interruptions in its continuity as possible. All interruptions in the ground plane, including vias, cau ground current to flow in a larger loop than is ideal, thereby increasing radiation and noise.
2. **Keep Ground Currents Separate**
   The ground currents for digital and analog circuits must be separated to prevent digital currents from ac the analog circuits. The best way to accomplish this is through correct component placement. If all the a digital circuits are placed on separate parts of the PCB , the ground currents will naturally be isolated. F work well, the analog section must contain only analog circuits on all layers of the PCB.
3. **Use the Star Grounding Technique for Analog Circuits**
   Star grounding uses a single point on the PCB as the official ground point. This point, and only this poir considered to be at ground potential. In a cell phone the battery ground terminal is a logical choice for tl Do not think of currents as flowing into the ground plane and disappearing; rather consider all ground cı flowing back to this ground point.

   Audio power amplifiers tend to draw relatively large currents that can adversely affect both their own an ground references in the system. To prevent this problem, provide dedicated return paths for bridged-aı power grounds and headphone-jack ground returns. Isolation allows these currents to flow back to the b without affecting the voltage of other parts of the ground plane. Remember that these dedicated return p not be routed under digital signal traces because they could block the digital return currents.
4. **Maximize the Effectiveness of Bypass Capacitors**
   Nearly all devices require bypass capacitors to provide instantaneous current. To minimize the inductar the capacitor and the device supply pin, locate these capacitors as close as possible to the supply pin v are bypassing. Any inductance reduces the effectiveness of the bypass capacitor. Similarly, the capacit provided a low-impedance connection to ground to minimize the capacitor's high-frequency impedance. connect the ground side of the capacitor to the ground plane, rather than routing it through a trace.
5. **Flood All Unused PCB Area with Ground**
   Whenever two pieces of copper run near each other, a small capacitive coupling is formed between the running ground flood near signal traces, unwanted high-frequency energy in the signal lines can be shu ground through the capacitive coupling.

Example Grounding

Figure 3 shows an example of a well-grounded system. Note, first, that the PCB is partitioned into a digital s bottom and an analog section at the top. The only signals crossing the partition boundary are I²C control sigr these have a direct return path following the signal trace. This layout ensures that digital signals will remain i section of the board and that no digital ground currents will be blocked by the splits in the ground plane. Also most of the ground plane is uninterrupted. Even in the digital section where there are interruptions, they are f apart to allow currents to flow freely.

For this example the star point is in the upper left corner of the PCB. The breaks in the analog portion of the allow the Class D and charge-pump currents to return to the star point without interfering with the general an plane. Also note that the headphone jack has a dedicated trace returning the headphone ground current to tl

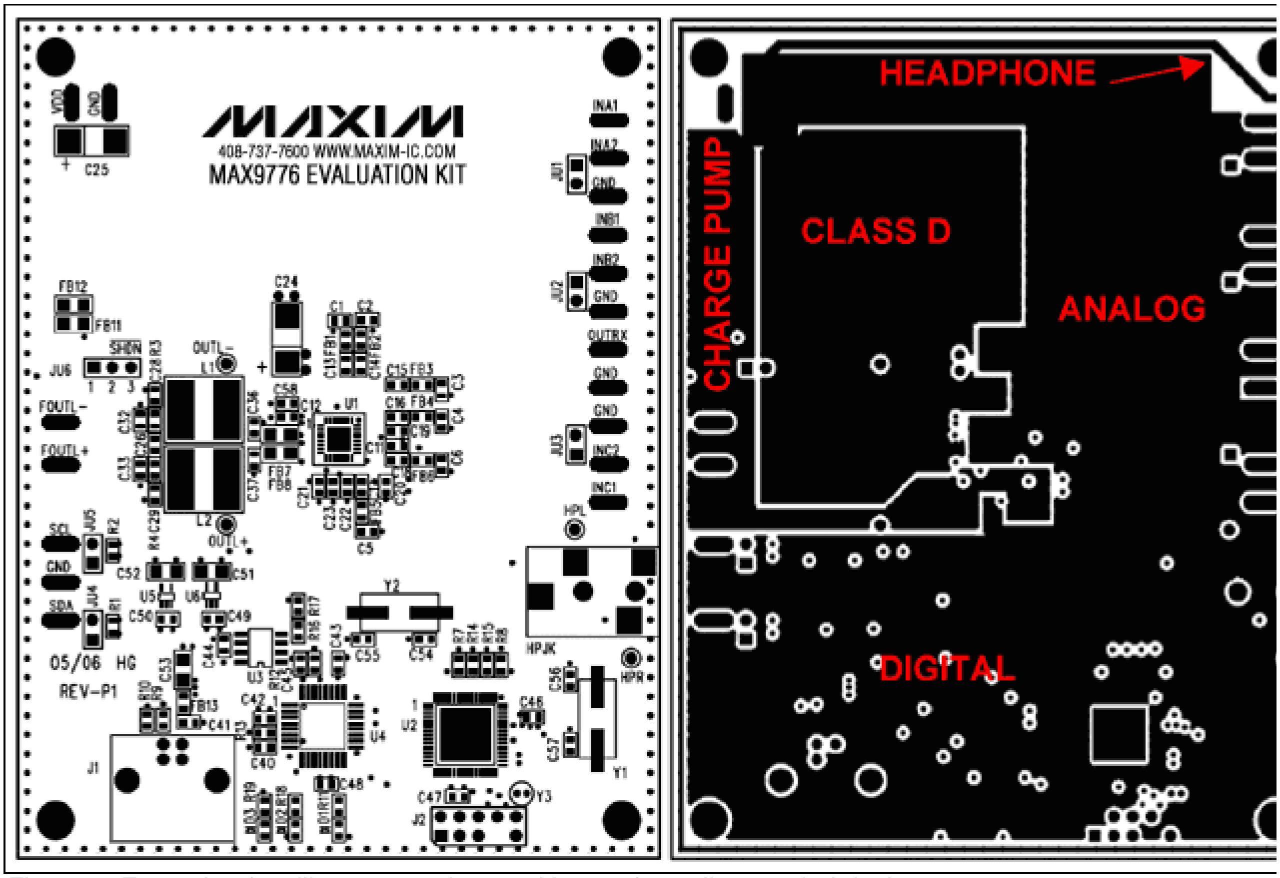


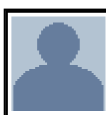
Figure 3. Example of a silkscreen and ground layer of a well-grounded design.

Conclusion

Although creating a well-designed PCB can be time-consuming and challenging, the investment is well wortl spent. The end result is a system with less noise, higher immunity to RF signals, and less distortion. The PC have better EMI performance and may require less shielding.

Ultimately, if the PCB is not carefully designed, preventable problems will be discovered when the product is These problems are much more difficult to fix once the layout is complete, and often demand significant time All too often the fixes require additional components that add to the total system cost and complexity.

An abbreviated version of this application note appeared in the July 16, 2007 issue of *EE Times*, a CMP pub



antoker - **Excellent article!**

There should be more articles like this one, that not only describes the process of creating a good layout, b
this process by real-world examples.

1 comment

Your Comments

Login or register to post a comment.

<h3><a href="http://www.maxim-ic.com/blogs/blog.mvp?page_num=1&doc_type=an&doc_id=4079&mode=r
target="_comments">We Want Your Feedback!</h3>

APP 4079: Sep 04, 2007
APPLICATION NOTE 4079, AN4079, AN 4079, APP4079, Appnote4079, Appnote 4079