Optimised For Audio

Analogue PCB design for the digital era.

David G. Tyas   IKON AVS Ltd, 238 Ikon Estate, Hartlebury, Worcs.. DY10 4EU
www.ikonavs.com

1.   Introduction

With increasing emphasis on digital audio, PCB design requirements for
analogue and mixed signal audio are often neglected. This short seminar
revisits many design considerations often neglected or overlooked in striving
to achieve optimum audio performance and reduction of EMC and Digital
interaction.

Normally to cover the important aspects of good analogue and mixed signal
PCB design would take at least a couple of days and then many years to
perfect, so in half an hour I can only give a general overview of some of the
potential problems and possible solutions.
But first….

2.   A Brief History

In the early days of PCB design, the design goal for many manufacturers was
simply to make it all fit, generally onto a single sided board. Often this task
was given over to the 'drawing office' and carried out with little in depth
understanding of how the circuit operated and how to optimise performance.
Over the years as circuit complexity increased and PCBs move from single
sided through double sided and onto multi layer assemblies, the nature of the
laid circuits has also changed with digital integrated circuits often of previously
unimagined functionality now very common.

As analogue circuit complexity increases, so do the requirements for
understanding the circuits operations so as to meet increasingly demanding
design requirements of integrating both analogue and digital circuitry into ever
smaller spaces whilst not compromising quality and ensuring compliance to
EMC and other regulations.

3. Topics to be covered

The seminar is split into five short sections each covering a different aspect of the overall topic and cumulating in a real design example.

4. Choice of Materials & Method

The PCB itself is an important component of any design and as with other components you need to select the correct part for the application.

4.1 Material

Whilst raw PCB material is generally available in five grades known as FR-1 to FR-5, where FR relates to its fire rating, the two most common two used in the electronics industry are FR-2 (Phenolic cotton paper) and FR-4 (Woven glass and epoxy). FR-2 is often referred to as SRBP and is commonly found on high volume and low cost consumer products, generally as a single sided board. FR-4, commonly referred to as fibreglass, is almost always found when PCBs use two or more layers.

When selecting a PCB material, pay careful attention to the moisture absorption and to the operating temperature. High operating temperatures can occur in unexpected places, such as in proximity to large digital chips that are switching at high speeds. Heat rises, so be aware that if one of those 160-pin digital ICs is located directly under a sensitive analogue circuit, both the PCB and the circuit characteristics may vary with the temperature.

4.2 How Many Layers?

Often the number of board layers has already been determined by system constraints, but if the designer has a choice here are some guidelines.

Very simple consumer electronics are sometimes fabricated on single-sided PCBs, keeping the raw board material inexpensive. These designs frequently include many jumper wires but this technique is only recommended for low-

frequency circuitry, as this type of design is extremely susceptible to radiated noise.

Today the most common PCB is double-sided. This type of board would seem to lend itself to easier routing by allowing the crossing of signals on different layers. While that is certainly possible, it is not actually ideal for analogue circuitry, particularly when using through hole components. With the increasing use of surface mount components though it is possible to implement good analogue designs with double sided boards.

Wherever possible, the bottom layer should be devoted to a continuous ground plane, and all other signals routed on the top layer. A ground plane provides several benefits:

- Ground is frequently the most common connection in the circuit so having it continuous on the bottom layer usually makes circuit routing easier.
- It lowers the impedance of all ground connections in the circuit, reducing conducted noise.
- It adds a distributed capacitance to every net in the circuit, helping to suppress radiated noise.
- It acts as a shield to radiated noise coming from underneath the board.

Double-sided boards, in spite of their benefits, are not the best method of construction, especially for sensitive designs. The most common board thickness is 1.5 mm and this separation is too great for full realisation of some of the benefits just listed. Distributed capacitance, for example, is very low due to this separation.

For critical designs multi-layer board, although more expensive, provide a number of benefits:

- They have better routing for power as well as ground connections. If the power is also on a plane, it is available to all points in the circuit simply by adding vias.
- Other layers are available for signal routing, making routing easier.
- There will be distributed capacitance between the power and ground planes, reducing high-frequency noise.

The decision to use multi-layer boards is complex, requiring the designer to weigh board cost against performance.

For the rest of this seminar we will only be considering the use of double sided boards.

4.3   Tracking Methods

When a PCB track turns a corner at a 90° angle, a reflection can occur. This is primarily due to the change of width of the track. At the apex of the turn, the track width is increased to 1.414 times its normal width (see below).
This upsets the transmission line characteristics, especially the distributed capacitance and self-inductance, resulting in the reflection. Of course not all PCB traces can be straight and have to turn corners but most CAD systems give some rounding effect on the trace, sharp 90⁰ tracks are a relic of the hand taping days of PCB layout. The rounding effects of CAD programs, however, still do not maintain constant width as it rounds the corner. The diagram shows progressively better techniques of rounding corners. Only the last example maintains constant track width and minimises reflections. Most CAD programs support these methods, but they can entail a little more work to master.
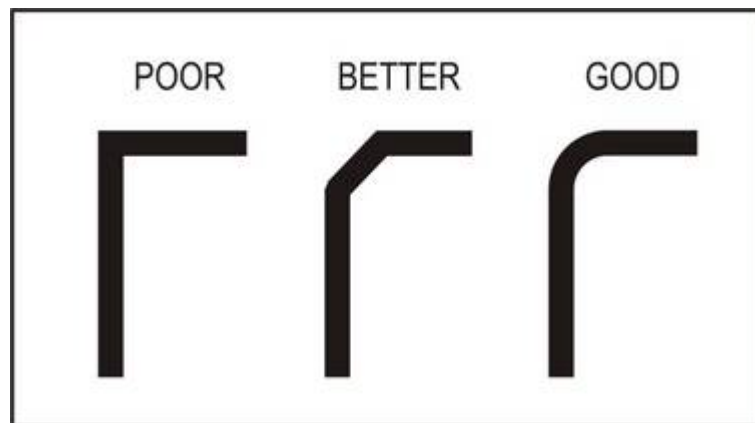

5.   Analogue Inputs

Analogue inputs are vulnerable to radiated interference because the pattern of tracks and component leads forms antennas. Common wires and PCB tracks have an inductance that varies between 6nH and 12nH per centimetre and at frequencies above 100 KHz, most PCB tracks are inductive, not resistive.

A good rule of thumb for antennas is that they begin to couple significant energy at about 1/20 of the wavelength of the received signal. Therefore, a 10cm track will begin to be a fairly good antenna at frequencies above 150 MHz and although the clock generator on the digital section of the PCB may not be operating at a frequency as high as 150 MHz, it approximates a square wave and will have harmonics throughout the frequency range where PCB conductors become efficient antennas.

A loop also can form an antenna and while most designers know not to make loops in critical signal pathways. Some designers will create a loop by the

technique they use for layout of the analogue section of the board. Consider the two cases illustrated below.
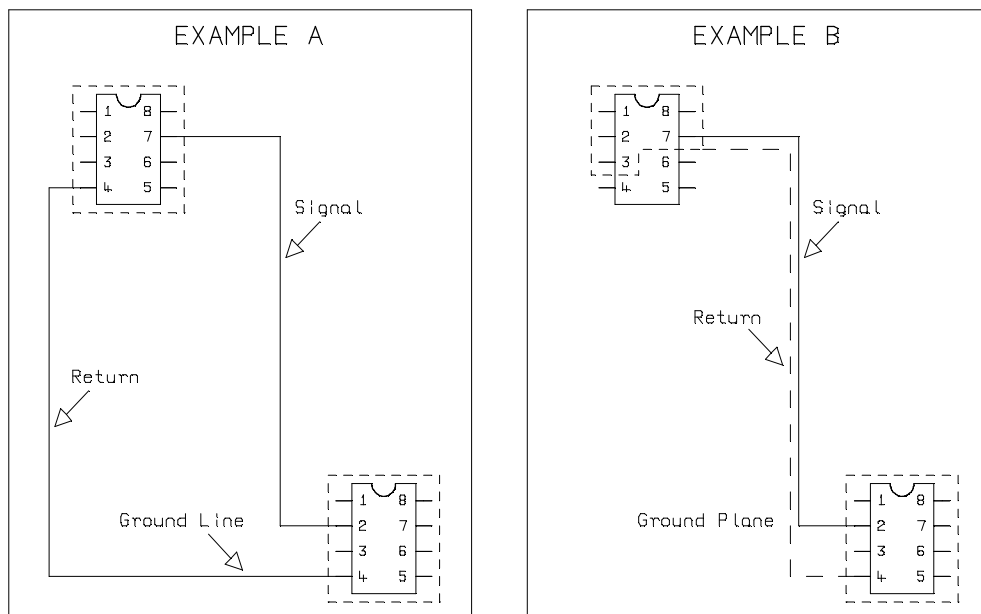
Example A is a bad design. It does not utilise an analogue ground plane at all. A loop is formed by the ground and signal traces.

Example B is a better design. Signal and return are coincident with each other, eliminating loop antenna effects completely.

Note that there are ground plane clearances for the ICs, but they are located away from the return path for the signal.

5.1 Track-to-Ground Plane Capacitors

PCB tracks, being composed of copper foil, form a capacitance with other tracks that they cross on other layers. For two tracks crossing each other on adjacent layers, this is seldom a problem. Coincident tracks (those that occupy the same routing on different layers), form a long skinny capacitor.

For this example let's assume a track width of 0.75mm and a coincident run of 7.5mm. This capacitance would then be in the order of 1.1pF. Of course, the antenna effect on a 7.5-mm trace would be devastating, so this example is a bit extreme.

Ignoring the antenna effects for now, there are cases in which even a very small parasitic capacitance like 1pF is unacceptable particularly if this occurs at the inverting input of the op-amp. It can cause a doubling of the output amplitude near the bandwidth limit of the op-amp and is an invitation to oscillation, especially since the trace is an efficient antenna above 180 MHz.

There are numerous fixes to this problem. The most obvious would be to shorten the length of the tracks. Another not-so-obvious fix would be to use a thinner track width to reduce the capacitance. Another fix is to remove the ground plane under the inverting input and the tracks leading to it.

The inverting inputs of op-amps, particularly high-speed op-amps, are especially prone to oscillation in high gain circuits due to unwanted capacitance on the input stage.

On op-amps it is important to minimise capacitance on the inverting input by reducing track width and placing components as close as possible to this input. If this input still oscillates, it may be necessary to scale the input and feedback resistors lower by a decade or two to change the resonance of the circuit. Scaling the resistors up will seldom help, as the problem is also related to the impedance of the circuit. The use of surface mount components is preferred as these assist with achieving the short tracks preferred.

Another possibility is to use dual op-amps as these generally keep both the inverting and non-inverting inputs on the same side of the device as the output allowing the track lengths between the op-amp and associated components to be minimised.
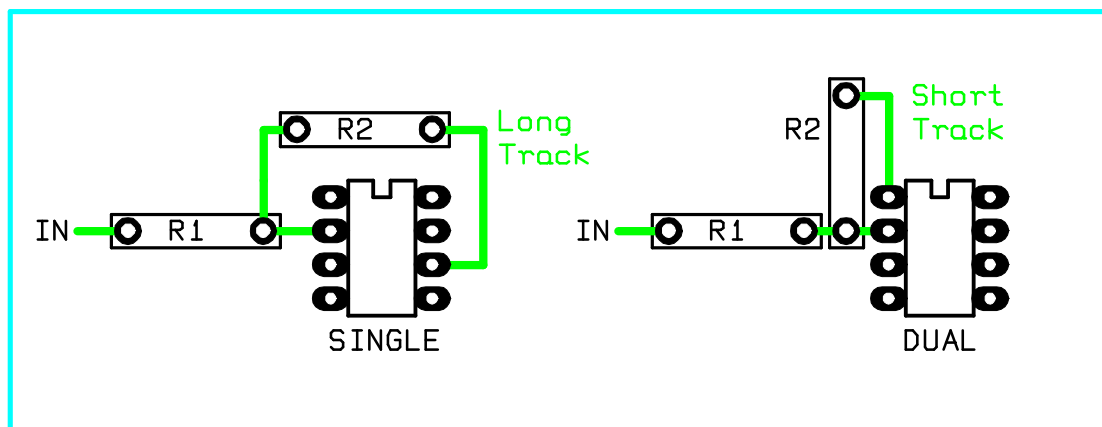

5.2 Input Grounding

Before moving onto grounding and ground planes, lets take a look at handling the input ground for both an unbalanced and balanced inputs circuits.

Consider the following two simplistic circuits, whilst very basic they will serve to illustrate the point. Both have a ground reference for the op-amp and both have an input ground, but how this input ground is handled can have a significant effect on the overall noise performance.
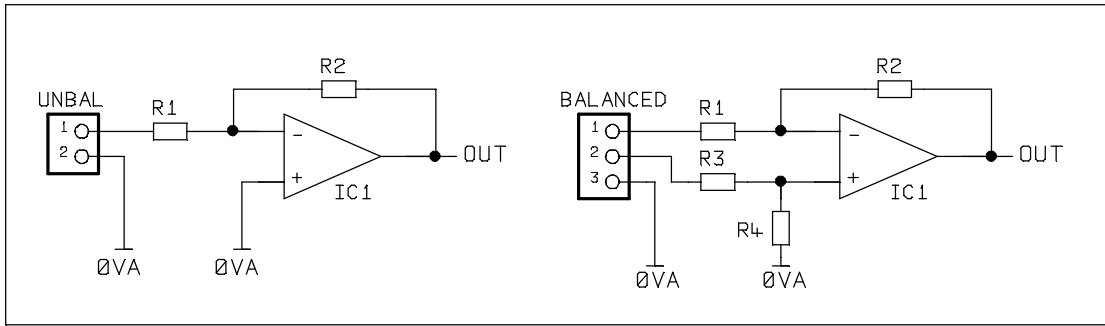
Looking first at the balanced input, here we treat the phase and anti-phase signals quite correctly as the send and return ensuring that both are routed to the active device as directly as possible and with minimum track length. The ground connection here is not part of the signal, it only provided RF and EMC screening, so it can quite safely be connected direct to any ground plane to return any induced currents to the reference ground and not the input circuit.
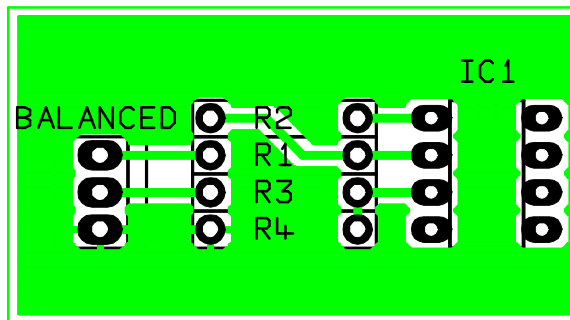
With the unbalanced input the ground is the active return for the input signal. It is best connected direct to the return connection of the op-amp (the non-inverting input in this case) rather than the ground plane. By doing this the input path, both send and return, are both referenced at the op-amp to minimise the effects of external interference. Whilst making little direct impact upon the circuits inherent noise performance, it will have a greater impact in the presence of external interference.
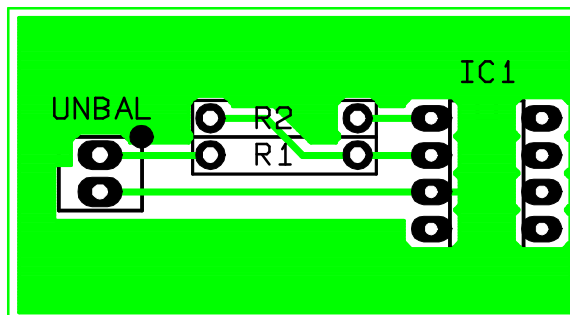
6.   Grounding and Ground Planes

Good grounding should be planned into a product from the start as a poorly grounded system is likely to have high distortion, noise, crosstalk, and RF susceptibility. Using separate grounding for analogue and digital sections of the circuitry is one of the simplest and most effective methods for noise suppression.

'Separate grounds' does not mean that the grounds are electrically separate in a system. They have to be common at some point, preferably a single low-impedance point. In a system there is only one ground, the electrical safety ground in an AC powered system or the battery ground in a DC powered system. Everything else 'returns' to that ground. Refer to everything that is not a ground as a 'return'. All returns should be connected together at a single point, which is system 'ground'. At some point, this will be the chassis. It is important to avoid ground loops by multiple connections to the chassis.

The ground in any system must serve two purposes. First, as above, it is the return path for all currents flowing to a device. Second, it is the reference voltage for both digital and analogue circuits. Grounding would be a simple exercise if the voltage at all points of the ground could be the same but this is not possible as all wires and tracks have a finite resistance, so whenever there is current flowing through the ground, there will be a corresponding voltage drop. Any loop of wire (even component leads) also forms an inductor increasing the ground impedance at high frequencies.

While designing the best ground system for a particular application is not a simple task, there are some general guidelines.

6.1   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 so it is best to ensure that all digital signal traces have a corresponding ground path via a continuous ground plane on the layer immediately adjacent to the signal layer. This layer should cover the same area as the digital signal trace and have as few interruptions in its continuity as possible.

6.2   Keep Ground Currents Separate

Ground currents for digital and analogue circuits must be separated to prevent digital currents from adding noise to the analogue circuits. The best way to accomplish this is through correct component placement. If all the analogue and digital circuits are placed on separate parts of the PCB, the ground currents will naturally be isolated. For this to work well, the analogue section must contain only analogue circuits on all layers of the PCB.

6.3   Use the Star Grounding Technique for Analogue Circuits

Star grounding uses a single point on the PCB as the official ground point. This point, and only this point, can be considered to be at ground potential. Do not think of currents as flowing into the ground plane and disappearing; rather consider all ground currents as flowing back to this ground point.

For high current audio devices, typically headphone amplifiers, provide dedicated return paths for ground returns. Isolation allows these currents to flow back to the official ground point without affecting the voltage of other parts of the ground plane.

6.4   Maximize the Effectiveness of Bypass Capacitors

Nearly all devices require bypass capacitors. To minimise the inductance between the capacitor and the device supply pin, locate these capacitors as close as possible to the supply pin that they are bypassing. Any inductance reduces the effectiveness of the bypass capacitor so surface mount devices are preferable. Similarly, the capacitor must be provided with a low-impedance connection to ground to minimise the capacitor's high-frequency impedance. If possible directly connect the ground side of the capacitor to the ground plane.

6.5   Flood All Unused PCB Area with Ground

Whenever two pieces of copper run near each other, a small capacitive coupling is formed between them. By running ground flood near signal tracks, unwanted high-frequency energy in the signal lines can be shunted to ground through the capacitive coupling.

6.6   Optimise the location of Analogue Circuitry

It is a good idea to locate analogue circuitry as close as possible to the I/O connections of the board. Digital designers used to high-current ICs will be tempted to use a wide track running several centimetres to the analogue circuit thinking that reducing the resistance in the track will help get rid of noise. What will actually result is a long, skinny capacitor that couples noise from digital ground and power planes into the op-amp, making the problem worse.

6.7   Example Grounding

The example below shows a well-grounded system with the PCB partitioned into an analogue section at the bottom and side as well as a digital section at the top. The only signals crossing the partition boundary are control signals, and these have a direct return path following the signal track. This layout ensures that digital signals will remain in the digital section of the board and that no digital ground currents will be blocked by the splits in the ground plane. Also note that most of the ground plane is uninterrupted.
For this example the star point is in the upper left corner of the PCB. Also note that the headphone jack has a dedicated trace returning the headphone ground current to the star point.
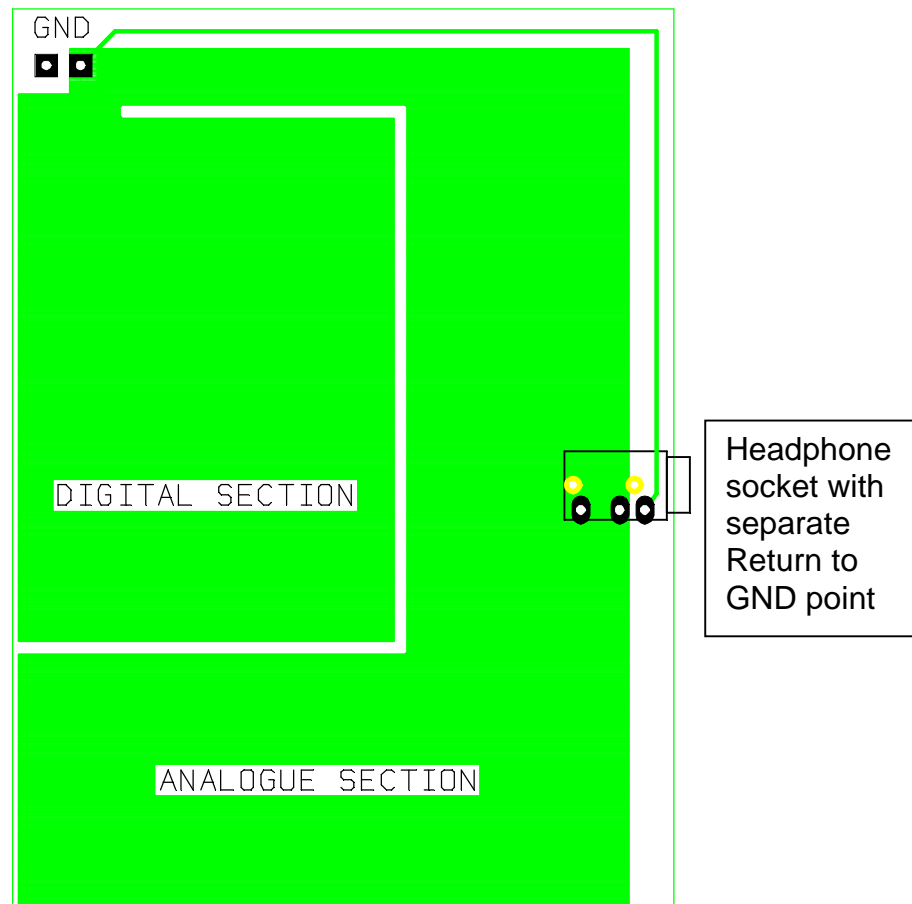
7.   *Analogue to Digital Boundary*

In most designs the analogue and digitals sections will use separate ground and power supplies even if these are derived from a common source. As we saw with ground / return connections, the analogue and digital power should

also return to the source without passing through the other section, even if this requires a longer route. This way any noise due to parasitic capacitance can be minimised.

With most A/D (analogue to digital) and D/A (digital to analogue) converters the manufactures have designed the component to allow the splitting of these sections and allow a simple split ground plane beneath the device, so don't compromise this by routing signals in this area. However there are always exceptions and you do need to study the manufactures data sheet for further information.

7.1 Analogue to Digital Converters

Early A/D converters used balanced inputs to allow designers greater flexibility to position the converter away from sensitive analogue circuitry. They also usually required a method of compensating for any DC offset on the incoming signal so as to maximise the conversion bits available with this adding to the signal pre-conditioning required.

Now, as the level of integration increases, we have the dual benefits of a smaller internal die to the converters, so lower RF emissions, as well as the creation of onboard digital filters to compensate for DC offsets. This now results, in most cases, in a much simplified single ended input but with the greater need to watch the routing of the signal (ground) returns.
Very often the recommended signal conditioning, whilst apparently analogue, is actually powered by the digital supplies, so it all needs treating as part of the digital circuit and located accordingly. Again you do need to study the manufactures data sheet for further information.

7.2   Digital to Analogue Converters

Fortunately D/A converters are in general much more benign than A/D converters and whilst the importance of separating the two ground planes is still essential, this is often made very easy by the chip manufacturers.

Other than the links to the digital audio (usually 4), in general there are no control pins for D/A converter, allowing the separate digital and analogue sections and support components to be laid easily.

7.3   When is Analogue Ground not Analogue Ground?

Cirrus Logic has a Digital Audio Interface Receiver (CS8416) that used both an analogue and digital supply and associated grounds. No problem, treat these as separate and return to the appropriate source. However on studying

the data sheet you read that the analogue and digital ground are actually the same and should be connected under the device as a digital ground.

Whilst not a startling revelation, it does simply illustrate that all is often not as straightforward as it appears, **so read and understand the data sheets**.

8   Typical Layout Example

To round off this seminar, lets take a look at an actual circuit and how this can be translated into an effective PCB layout.

Block Diagram
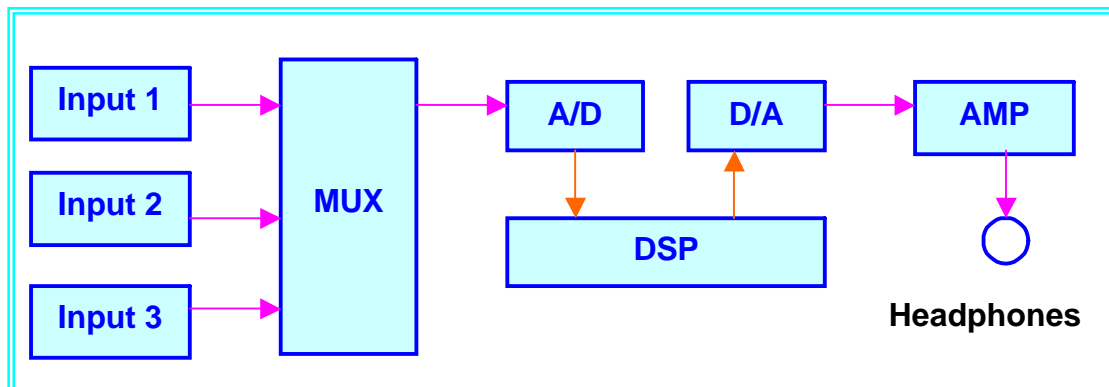
The first step of any PCB design is choosing where to place the components, as careful component placement can ease signal routing and ground partitioning. It also minimises noise pickup and the board area required. Circuits that contain a mixture of digital and analogue circuitry must be separated to prevent noise from the digital portion from interfering with the sensitive analogue circuits. Partitioning the PCB into a digital and an analogue region simplifies this task.

Once the PCB has been segregated into analogue and digital sections, the component placement within the analogue section must be selected. Components should be placed to minimise the distance that audio signals travel. Locate the audio amplifier as close to the headphone socket as possible. Place the devices supplying the analogue audio as close to the amplifier as possible to minimise noise pickup on the amplifier inputs.
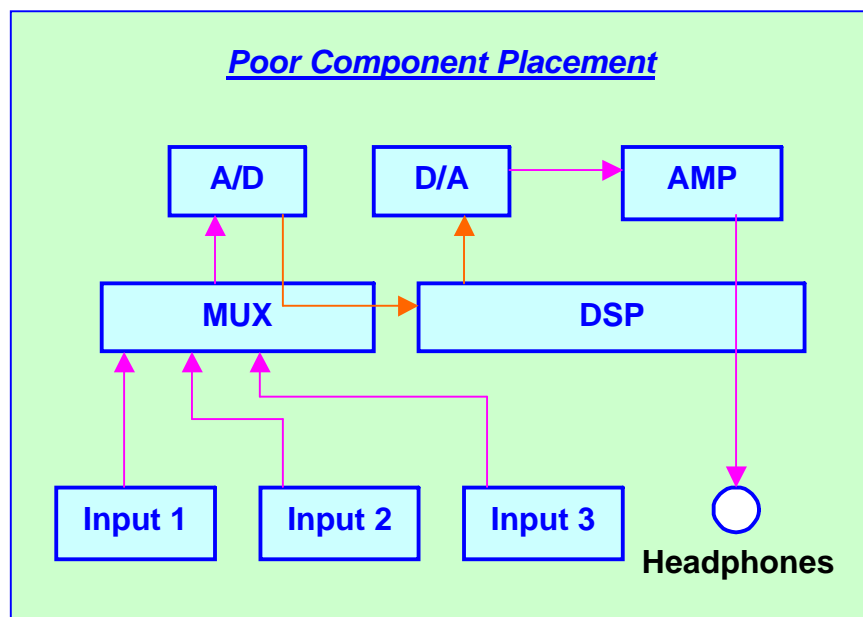
Poor Component Placement

The drawing below shows an example of poor audio component placement with the most serious problem being the audio amplifier as it is placed quite close to and between digital components making it prone to interference from noisy digital circuitry.

Other problems with the component placement are;-

> The amplifier is not placed near the headphone socket increasing the track resistance and therefore decreasing the power delivered to the load.
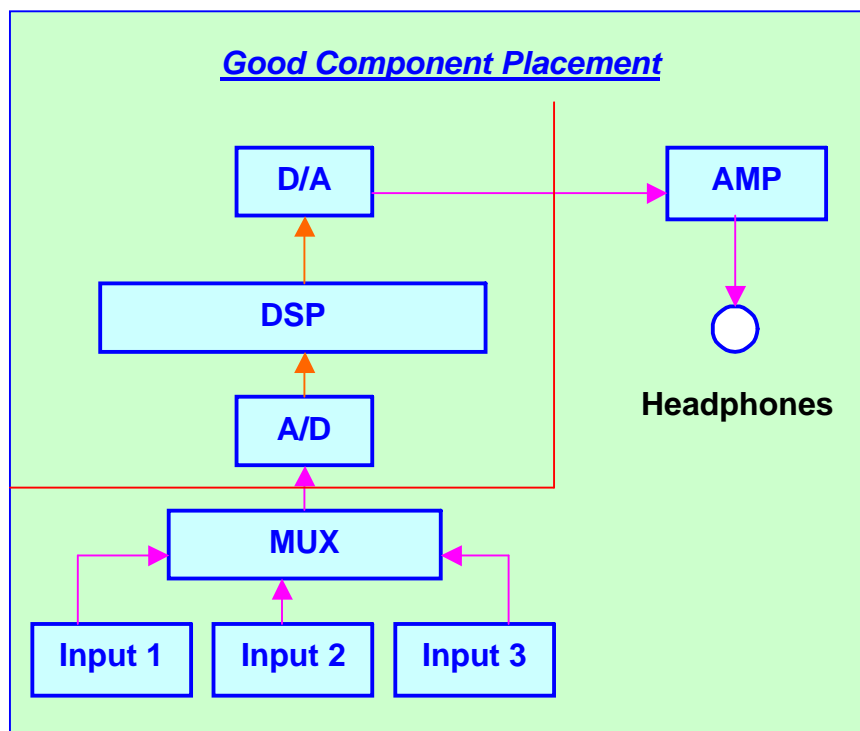
> The digital signals from the A/D pass near the analogue multiplexer and can cause interference. It will also be virtually impossible to provide segregated ground planes.

> Finally, since the components are so spread apart, the tracks connecting the components will be routed near other subsystems increasing the difficulty of routing the tracks

Poor Component Placement

| A/D | D/A | AMP |

| MUX | DSP |

| Input 1 | Input 2 | Input 3 | Headphones

Good Component Placement

The alternative layout below shows the same components but rearranged to use the space more effectively and to minimise track lengths. Notice how all the audio circuitry has been partitioned to be near the headphone socket. The audio input and output tracks are much shorter and the non audio circuitry has been moved to a different part of the PCB. This design will have lower overall system noise, be less susceptible to RF interference, and be easier to layout.

Good Component Placement

```
        D/A  ------------->  AMP
         ↑                    |
                              ↓
        DSP                  (O)
         ↑                Headphones
        A/D
         ↑
        MUX
       ↑ ↑ ↑
  Input 1  Input 2  Input 3
```

Acknowledgements

I would like to thank the following companies for their assistance with information for this seminar.

Texas Instruments Inc.
Maxim Integrated Products
Cirrus Logic Inc.