

The PCB Is a Component of Your Op-Amp Design

By Bruce Carter, Senior Applications Specialist, Texas Instruments, Inc

Save an expensive board from becoming scrap because of design blunders in a small section of analog circuitry

Most analog designers are familiar with how to use integrated circuits (ICs) and passive components to implement a design. There is one additional circuit component, however, that must be considered for the design to be a success – the printed circuit board (PCB) on which the circuit is to be located. Analog circuitry is very different from digital circuitry, and should be partitioned in its own section of the board, using special layout techniques. Printed circuit board effects become most apparent in high-speed analog circuits, but common mistakes described below can even affect the performance of audio circuits. Any effect caused by the PCB itself should be minimized, such that the operation of the analog circuitry in production will be the same as the performance of the design and prototype.

Normal design cycles, particularly of large digital boards, dictate early layout of the PCB. The digital circuitry may have been simulated, but in most cases the production PCB itself is the prototype, and may even be sold to a customer. Digital designers can correct small mistakes by implementing "cuts and jumpers", reprogramming gate arrays or flash memories – and go on with the next project. This is not the case with analog circuitry. Some common analog design cannot be corrected by the "cut and jumper" method. They can and will render the entire PCB unusable. It is very important for the digital designer, who is used to "cuts and jumpers", to read and understand this article prior to releasing a board to a layout service. A little extra thought during development can save a board worth thousands of dollars from becoming "scrap" because of blunders in a tiny section of analog circuitry.

Noise Sources

Noise is conducted into the analog circuit through its connections to other circuits. Analog circuitry must be connected to the "outside world" by at least a ground connection, a power connection, an input, and an output. Noise can be conducted into the circuit through all of these paths, and others.

Noise is radiated into the analog circuitry from many external sources. The most common is high-speed digital logic including digital signal processing (DSP) chips that reside in the system or on the same PCB. High speed clocks and switching digital signals create considerable Radio Frequency Interference (RFI). Other sources of radiated noise are endless: a switching power supply, cellular telephones, broadcast radio and TV, fluorescent lighting, nearby computers, and lightning. Even if the analog circuitry only operates at audio frequencies, RFI may produce noticeable noise in the output.

PCB Materials – Choose the Right One for Your Application

Printed circuit board materials are available in various grades, as defined by the National Electrical Manufacturers Association (NEMA). It would be convenient for designers if this organization was closely allied with the electronics industry – controlling parameters such as resistivity and dielectric constant of the material. Unfortunately, that is not the case. NEMA is an electrical safety organization, and the different PCB grades primarily describe the flammability, high temperature stability, and moisture absorption of the board. Therefore, specifying a given NEMA grade does not guarantee electrical parameters of the material. If this becomes critical, consult the manufacturer of the raw board stock.

Raw PCB stock is graded in flammability ratings (FR) from 1 to 5, with 1 being the most flammable and 5 the least. FR-4 is commonly used in industrial quality equipment, while FR-2 is used in high volume consumer applications. Although there are no set rules for this, it appears to be an industry "standard". Deviating from it without good reason can limit the number of suppliers for raw board material, and the PCB houses that can fabricate the board, as their tooling is already set up for these materials.

In selecting a board material, pay careful attention to moisture absorption. Just about every desirable performance characteristic of the board will be negatively impacted by moisture. This includes surface resistance of the board, dielectric leakage, high voltage breakdown and arcing, and mechanical stability. Also pay attention to operating temperature. High operating temperatures can occur in unexpected places, such as in proximity to large digital chips that are switching at high speeds. Be aware that heat rises, so if one of those 300-pin IC's is located directly under a sensitive analog circuit – both the PCB and circuit characteristics may vary with the temperature.

How Many Layers are Best?

Many times, the number of board layers already has been determined by system constraints. If the designer has a choice, however, there are some guidelines.

Very simple consumer electronics are sometimes fabricated on single sided PCB's, keeping the raw board material inexpensive (FR-1 or FR-2) with thin copper cladding. These designs frequently include many jumper wires – simulating the circuit routing on a double-sided board. This technique is only recommended for low frequency circuitry. For reasons described below this type of design is extremely susceptible to radiated noise. Therefore, it is actually more complex to design a board of this type due to the many things that can go wrong.

The next level of complexity is double sided. Initially, this type of board would seem to lend itself to easier routing, because now there are two layers of foil, and it is possible to route signals by crossing traces on different layers. While that is certainly possible, it is not recommended for analog circuitry. 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. Having it continuous on the bottom layer usually makes the most sense for circuit routing.
- It increases mechanical strength of the board.
- It lowers the impedance of all ground connections in the circuit, which reduces undesirable conducted noise.
- It adds a distributed capacitance to every net in the circuit - helping to suppress radiated noise.
- It acts 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 or high-speed designs. The most common board thickness is 1.5 mm. This separation is too great for full realization of some of the benefits listed above. Distributed capacitance, for example, is very low due to the separation. Critical designs call for multi-layer boards. Although more expensive, they provide the following benefits:

- 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 a complex one, balancing board cost with performance. Be sure to consider the cost of qualification testing, if any. Multi-layer boards present much lower design risk.

Grounding

Good grounding is a system level design consideration. Proper grounding should be planned into the product from the first conceptual design reviews.

Separate grounding for analog and digital portions of circuitry is one of the simplest and most effective methods for noise suppression. One or more layers on multi-layer PCB's are usually devoted to ground planes. If the designer is not careful, the analog circuitry will be connected directly to these "ground" planes. The analog circuitry return, after all, is the same net in the netlist as digital return. Auto-routers respond accordingly – and connect all of the grounds together, creating a disaster.

Ground and power planes are at the same AC potential, due to de-coupling capacitors and distributed capacitance. Therefore, it is important to isolate the power planes as well.

Do not overlap digital and analog planes (figure 1). Place analog power coincident with analog ground, and digital power coincident with digital ground. If any portion of analog and digital planes overlap, the distributed capacitance between the overlapping portions will couple high-speed digital noise into the analog circuitry. This defeats the purpose of isolated planes.

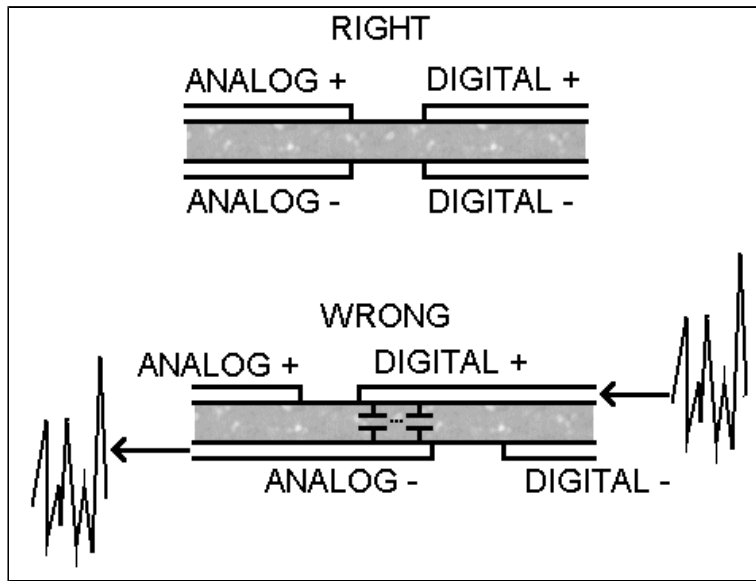


Figure 1: No Digital and Analog Plane Overlap

"Separate grounds" does not mean that the grounds are electrically separate in the system. They have to be common at some point, preferably a single, low impedance point (figure 2). System-wise, there is only one ground – the electrical safety ground in an AC powered system or 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. Insuring only one chassis ground point is one of the most difficult aspects of system design.

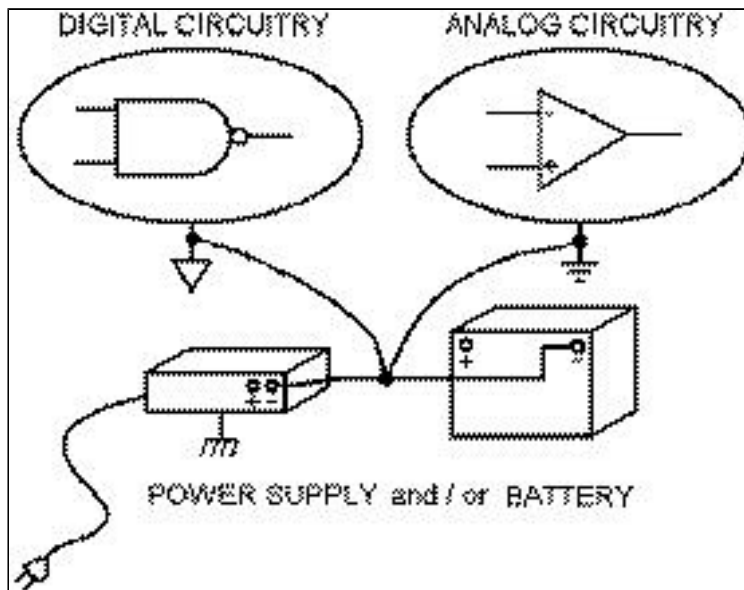


Figure 2: A Single Low Impedance Point

If at all possible, dedicate separate connector pins to separate returns, and combine the returns only at system ground. Aging and repeated mating causes connector pins to increase in contact resistance, so several pins are needed. Many digital boards consist of many layers and hundreds or thousands of nets. The addition of one more net is seldom an issue, but the addition of a several connector pins almost always is. If this cannot be done, then it will be necessary to make the two returns a single net on the PCB – with very special routing precautions.

It is important to keep digital signals away from analog portions of the circuit. It makes little sense to isolate planes, keep analog traces short, and place passive components carefully if there are high-speed digital traces running right next to the sensitive analog traces. Digital signals must be routed around analog circuitry, and not overlap analog ground and power planes (figure 3).

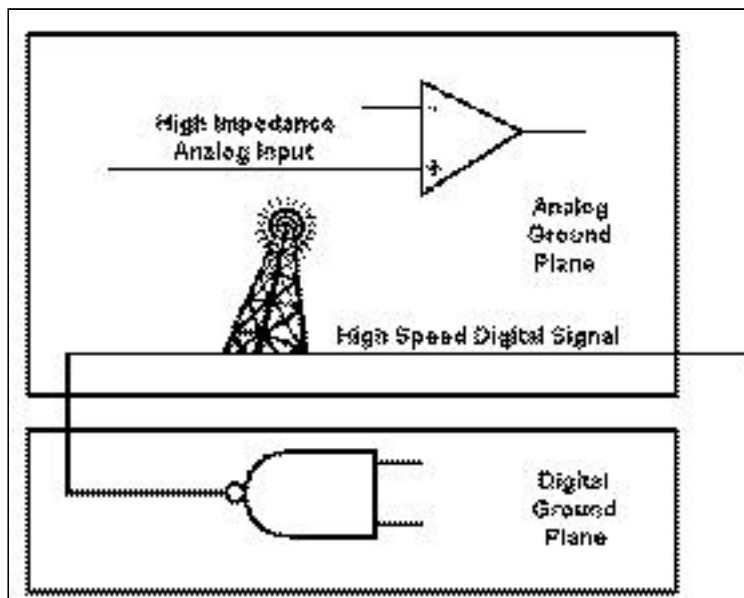


Figure 3: Route Digital Signals Around Analog Circuitry

Most digital clocks are high enough in frequency that even small capacitances between traces and planes can couple significant noise. Remember that it is not only the fundamental frequency of the clock that can cause a potential problem, but also the higher frequency harmonics.

It is a good idea to locate analog circuitry as close as possible to the I/O connections of the board. Digital designers, used to high current IC's, will be tempted to make a 50 mil trace run several inches to the analog circuitry, thinking that reducing the resistance in the trace will help get rid of noise. What they have actually done is create a long, skinny capacitor that couples noise from digital ground and power planes into the op-amp, making the problem worse!

The Frequency Characteristics of Passive Components

Many designers are ignorant of the frequency limitations of the passive components they use in analog circuitry. Passive components have limited frequency ranges, and operation of the part outside of that range can have some unexpected results. In most cases, a "right" passive component will fit on the same pads as a "wrong" passive component, but not always. Start the design process by carefully considering the high frequency characteristics of passive components, and putting the correct part outline on the board from the start.

Resistors

High frequency performance of resistors is approximated by the schematic shown in figure 4:

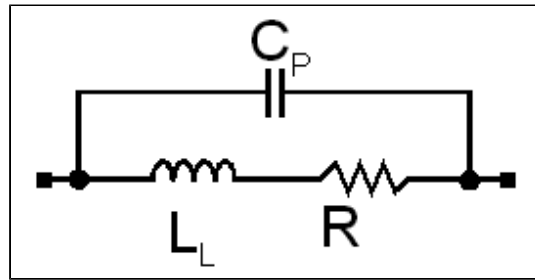


Figure 4: High Frequency Performance Schematic

Resistors are constructed three ways, wire wound, carbon composition, and film. It does not take a lot of imagination to understand how wire wound resistors can become inductive, because they are coils of resistive wire. Film resistors are also coils of thin metallic film. Therefore, they are also inductive at high frequencies.

The end caps of resistors are parallel, and there will be an associated capacitance. Usually, the resistance will make the parasitic capacitor so "leaky" that the capacitance does not matter. For very high resistances, the capacitance will appear in parallel with the resistance, lowering its impedance at high frequencies.

Capacitors

High frequency performance of capacitors is approximated by the figure 5.

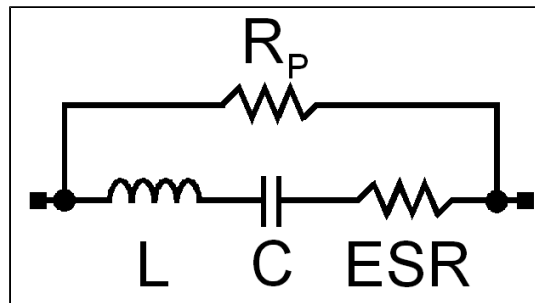


Figure 5: High Frequency Performance Capacitor

Film and electrolytic capacitors have layers of material wound around each other, which creates a parasitic inductance. Self-inductance effects of ceramic capacitors are much smaller, giving them a higher operating frequency. There is also some leakage current from plate to plate R_P , which appears as a resistance in parallel with the capacitor. The most important parasitic component in a capacitor is the Equivalent Series Resistance (ESR). It is due to resistance within the plates and electrolyte of an electrolytic capacitor. Capacitors used for de-coupling should be low ESR types, as any series resistance limits the effectiveness of the capacitor for ripple and noise rejection. Elevated temperatures severely increases ESR, and can be permanently destructive to capacitors.

The leads of through-hole parts also add a parasitic inductance. For small values of capacitance, it is important to keep the lead lengths short. The combination of parasitic inductance and capacitance can produce resonant circuits.

PCB Trace Antennas

A board is vulnerable to radiated interference because the pattern of traces and component leads form antennas. Common wires and PCB traces have inductance that varies between 6 and 12 nH per centimeter. At frequencies above 100 kHz, most PCB traces are inductive – not resistive.

A 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 10 cm trace will begin to be a fairly good antenna at frequencies above 150 MHz. Remember that although the clock generator on a digital PCB may not be operating at a frequency as high as 150 MHz, it approximates a square wave. Square waves will have harmonics throughout the frequency range where PCB conductors become efficient antennas.

A loop can also form an antenna. Without realizing it, most digital designers are familiar with loop antenna theory. They know not to make loops in critical signal pathways. Some designers that would never think of making a loop with a high-speed clock or reset signal, however, will turn right around and create a loop by the technique they use to layout the analog section of the board. Loop antennas constructed as loops of wire are easy to visualize. Slot antennas are harder to visualize, but just as efficient. Consider the three cases illustrated in figure 6:

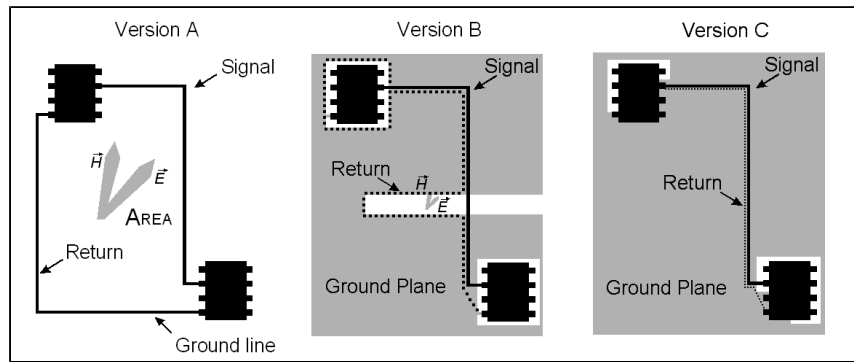


Figure 6: Keep Signal and Return Coincident to Avoid Inadvertent Antennas

Version "A" is a bad design. It does not utilize an analog ground plane at all. A loop is formed by the ground and signal traces. An electric field "E" and perpendicular magnetic field "H" are created, and form the basis of a loop antenna.

Version "B" is a better design, but there is intrusion into the ground plane, presumably to make room for a signal trace. A much smaller slot antenna is formed by the difference in pathways between signal and return.

Version "C" is the best design. Signal and return are coincident with each other, eliminating loop antenna effects completely. Note that there are cutouts for the IC's, but they are located away from the return path for the signal.

When a PCB trace turns a corner at a 90° angle, a reflection can occur. This is primarily due to the change of width of the trace. At the apex of the turn, the trace width is increased to 1.414 times its normal width. This upsets the transmission line characteristics, especially the distributed capacitance and self-inductance of the trace – resulting in the reflection. It is a given that not all PCB traces can be straight. Some will have to turn corners. Most CAD systems give some rounding effect on the trace – sharp 90° traces are a relic of the "tape-up" days of PCB layout. The rounding effects of CAD programs, however, still do not maintain constant width as the trace round the corner. Figure 7 shows progressively better techniques of rounding corners. Only the last example maintains constant trace width and minimizes reflections. Most CAD programs support these methods, but they can entail a little more work to master.

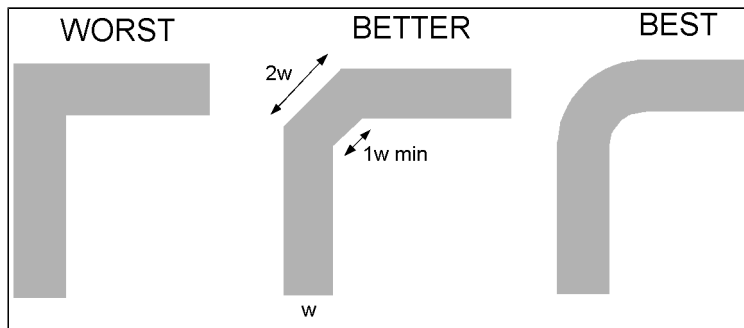


Figure 7: Round the Corners to Minimize Reflections

Trace to Plane Capacitors

PCB traces, being composed of foil, form capacitance with other traces that they cross on other layers. Two traces crossing each other on adjacent planes seldom cause a problem. Coincident traces (those that occupy the same routing on different layers), form a long, skinny capacitor. Fortunately, these capacitances are usually small, only affecting high frequency designs. It is important, however, to minimize capacitance at the inverting input to an op-amp in high-speed designs. Otherwise, oscillation may occur.

Capacitance can be reduced by shortening the PCB traces and reducing trace width. If oscillation still occurs, the resistors can be lowered a decade or two to damp it out. This will also result in lower circuit noise, but at the penalty of increased power consumption.

Trace to Trace Capacitors and Inductors

PCB traces are not infinitely thin. They have some finite thickness, as defined by the "ounce" parameter of the copper clad foil. The higher the number of ounces, the thicker the copper. If two traces run side-by-side, then there will be capacitance and inductive coupling between them. The formulas for these parasitic effects can be found in transmission line and / or microstrip references, but are too complex for inclusion here.

Signal lines should not be routed parallel to each other, unless transmission line or microstrip effects are desired. Otherwise, a gap of at least three times the signal trace width should be maintained (figure 8).

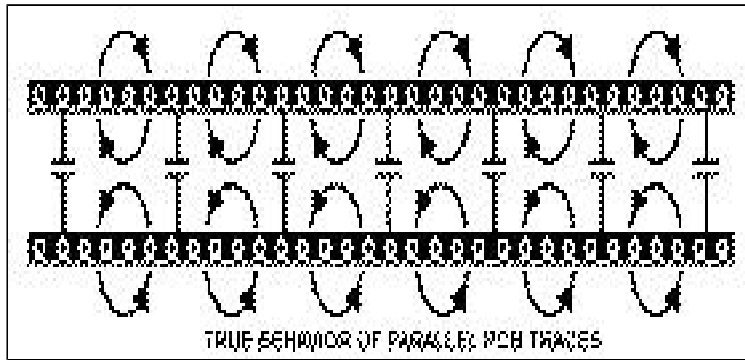


Figure 8: True Behavior of Parallel PCB Traces

Inductive Vias

Whenever routing restraints force vias, parasitic inductors are formed. This inductance is small, but can become troublesome at high frequencies. It is best to avoid vias and route all signals on one layer of the board.

Flux Residue Resistance

An unclean board can affect analog circuit performance. If the circuit has very high resistances – even in the low Megohms – pay special attention to cleaning. A finished assembly may be adversely affected by flux or cleansing residue. The electronics industry, in the past few years, has joined the rest of the world in becoming environmentally responsible. Hazardous chemicals are being removed from the manufacturing process, including flux that has to be cleaned with organic solvents. Water-soluble fluxes are becoming more common, but water itself can become easily contaminated with impurities. These impurities will lower the insulation characteristics of the PCB substrate. It is vitally important use freshly distilled water every time a high impedance circuit is cleaned. There are applications that may call for the older organic fluxes and solvents, such as very low power battery powered equipment with resistors in the 10’s of Megohm range. Nothing can beat a good vapor de-fluxing machine for ensuring that the board is clean.

De-Coupling

Noise can propagate into analog circuitry through the power pins of the circuit as a whole and op-amp itself. If analog circuitry is located on the same board with digital circuitry, it is important to de-couple the analog from the digital circuitry.

Table 1 is a rough guideline describing the maximum useful frequencies of common capacitor types.

Aluminum Electrolytic	100 kHz
Tantalum Electrolytic	1 MHz
Mica	500 MHz
Ceramic	1 GHz

Table 1: Frequencies of Common Capacitor Types

Recommended Maximum Frequencies for Capacitors

Tantalum electrolytic capacitors are useless for frequencies above 1 MHz. Effective high frequency de-coupling at higher frequencies demands a ceramic capacitor. Self-resonances of the capacitor must be known and avoided or the capacitor may not help, or even make the problem worse. Figure 9 illustrates the typical self-resonance of two capacitors commonly used for bypassing – 10µF tantalum electrolytic and 0.01µF ceramic.

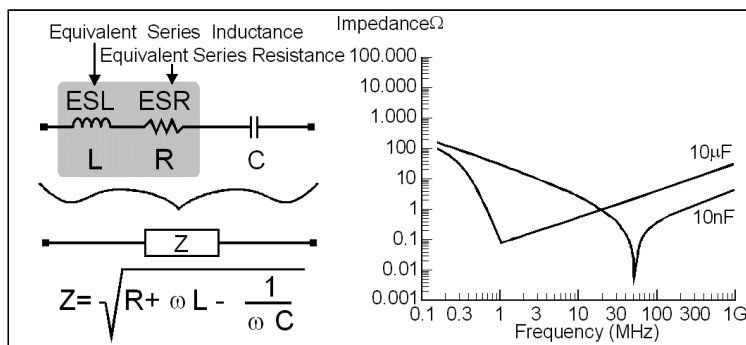


Figure 9: Typical Self-resonance Values

Consider these resonances to be typical values, the characteristics of actual capacitors can vary from manufacturer to manufacturer and grade of part to grade of part. The important thing is to make sure that the self-resonance of the capacitor occurs at a frequency above the range of the noise you are trying to reject. Otherwise, the capacitor is inductive.

Don't assume that a single 0.1µF capacitor will de-couple all frequencies. Smaller capacitors may work better at higher frequencies than larger ones. When poor de-coupling is suspected, try a smaller capacitor rather than a larger one.

A de-coupling capacitor must be included on every op-amp IC package – there are 1, 2 or 4 op-amps per package. The value of capacitor must be picked carefully to reject the type of noise present in the circuit.

In particularly troublesome cases, it may be necessary to add a series inductor into the power supply line connecting to the op-amp. This inductor is in addition to the de-coupling capacitors, which are the first line of defense. The inductor should be located before, not after the capacitors.

Another technique that is lower in cost is to replace the series inductor with a small resistor in the 10 to 100 range. The resistor forms a low-pass filter with the de-coupling capacitors. There is a penalty to pay for this technique – depending on the power consumption of the op-amp, it will reduce the rail-to-rail voltage range. The resistor forms a voltage divider with the op-amp as a resistive active component in the lower leg of the divider. Depending on the application, this may or may not be acceptable.

There is usually enough low frequency ripple on the power supply at the board input to warrant a bulk de-coupling capacitor at the power input. This capacitor is used primarily to reject low frequency signals, so an aluminum or tantalum capacitor is acceptable. An additional ceramic cap at the power input will de-couple any stray high frequency switching noise that may be coupled from other boards.

Input and Output Isolation

Many noise problems are the result of noise being conducted into the circuit through its input and output pins. Due to the high frequency limitations of passive components, the response of the circuit to high frequency noise can be quite unpredictable.

In situations where conducted noise is substantially different in frequency from the normal operating range of the circuit, the solution may be as simple as a passive RC low-pass filter that rejects RF frequencies while having negligible effect at audio frequencies. A good example is RF noise being conducted into an audio op-amp circuit. Be careful, though. A low-pass filter loses its characteristics at 100 to 1000 times f_{3db} . More stages may be required to cover different frequency ranges. If this is the case, make the highest frequency filter the one nearest to the source of noise. Inductors or ferrite beads can also be used in a noise rejection filter network to eliminate conducted noise. Ferrite beads are inductive up to a specified frequency, and then become resistive.

The effect of radiated energy coupling into an analog circuit can be so bad that the only solution to the problem may be to completely shield the circuit from radiated energy. This shield is called a "Faraday Cage", and must be carefully designed not to allow frequencies that are causing the problem to enter the circuit. This means that the shield must have no holes or slots larger than $1/20$ the wavelength of the offending frequency. This can be a demanding requirement. It is a good idea to design a PCB from the beginning to have enough room to add a metal shield if it becomes necessary. If a shield is used, frequently the problem will be severe enough that ferrite beads will also be required on all connections to the circuit.

Packages

Op-amps are commonly packaged one, two, or four per package. Single op-amps often contain additional inputs for features such as offset nulling. Op-amps packaged two and four per package only offer inverting and non-inverting inputs, and the output. If the additional features are important, the only package choice is single. Be aware, though, that the offset-nulling pins on a single op-amp package can act as secondary inputs, and must be treated carefully. Consult the data sheet on the particular device you are using.

The single op-amp package places the output on the opposite side from the inputs. This can be a disadvantage at high-speeds, because it forces longer PCB traces. It may make more sense to use a dual op-amp package, even if the second op-amp is unused. Figure 10 illustrates trace length for an inverting op-amp stage.

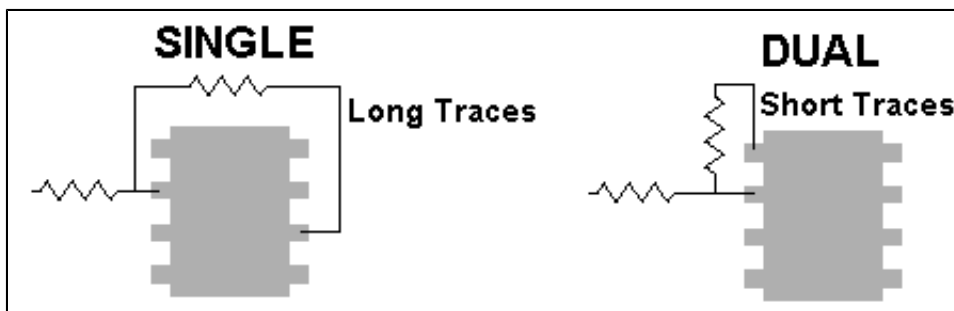


Figure 10: Trace Length for an Inverting Op-amp Stage

Unused Sections

In many op-amp designs, one or more op-amps may be unused. If this is the case, the unused section must be terminated properly. Improper termination can result in greater power consumption, more heat, and more noise in op-amps on the same physical IC. If the unused section of the op-amp is connected as shown in "better" illustration on figure 11, it will be easier to use it for design changes.

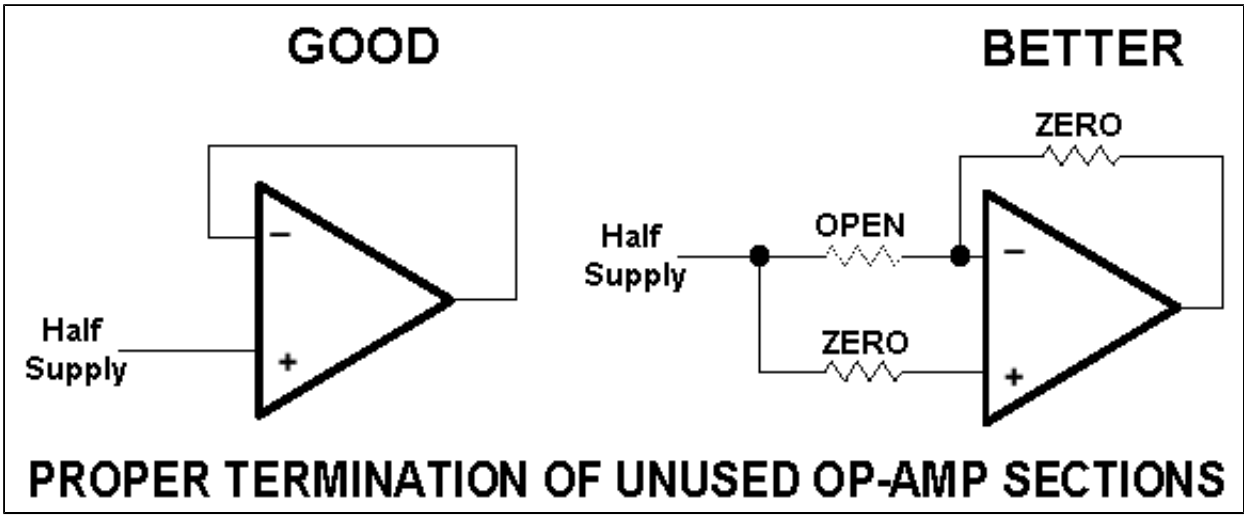


Figure 11: Proper Termination