SECTION XII
HARDWARE TECHNIQUES

- Leakage In Insulators:
  Guard Rings, Electrostatic Damage (ESD)

- Grounding and Signal Routing:
  Signal Return Currents, Ground Noise and Ground Loops, Star Grounds, Separate Analog and Digital Grounds, Ground Planes, Transmission Lines, System Grounds, Signal Routing

- Problem Areas:
  Limitations of Spice Modelling, Sockets, Prototyping

SECTION XII

HARDWARE TECHNIQUES

JAMES M. BRYANT

LEAKAGE IN INSULATORS

Just as conductors are improperly viewed as superconductors, so are insulators often mistakenly treated as perfect insulators, rather than very high resistances, which is the more accurate model.

Most printed circuit board materials are very good insulators, but they are not perfect, and inadequately cleaned PCB material may be quite a poor insulator. Furthermore, PCBs are anisotropic - even on a clean PCB different parts of the surface may have different resistivities, and the bulk resistance (between two plated through holes, for instance) is generally lower than the surface resistance between two tracks.

Since the insulation resistance is so variable (and it will vary further with temperature and humidity) it is hard to predict in any particular circumstances but it is safe to assume that it is unlikely that the resistance between two conductors on a clean PCB will drop below 10^{10} - 10^{11} ohms, and with teflon PCB material (which is very expensive) will usually be over 10^{12} ohms.

GUARD RINGS

In applications where high impedances and very low currents are involved a guard ring may be used to minimize the effects of low insulation resistance. If critical high impedance nodes are surrounded by a ring of conductor which is at (or very close to) the potential of the node itself then the leakage current at the node will be minimized. If the node is at, or near to, ground then a grounded guard ring will be appropriate, if it is at some other potential it may be necessary to use a high input impedance buffer amplifier, with its input connected to the node, to force the guard ring to the node potential. It is obvious that, in general, guard rings should be on both sides of the PCB with plated-through holes.

Nodes which are sufficiently sensitive to require guard rings should not contain plated through holes (unless the PCB is made of teflon) because, as mentioned above, the bulk resistivity of PCB material is less than the surface resistivity.

An alternative to the use of a guard ring is to use teflon stand-off insulator(s) to support the high impedance point(s). If virgin teflon is used insulation resistance of around 10^{15} ohms is possible ("Virgin teflon" is a solid piece of new teflon material which has been machined to shape and has not been welded together from powder or grains). The material of the rest of the circuit board need not have particularly high insulation resistance.

LEAKAGE RESISTANCE ON PCBs

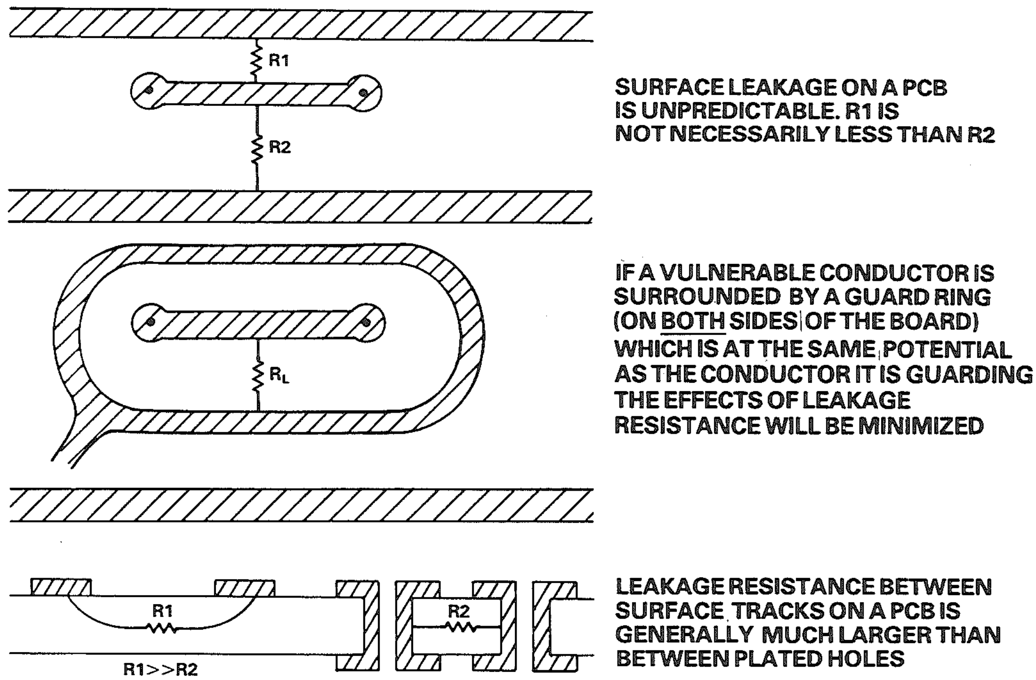


Figure 12.1

A VIRGIN TEFLON STANDOFF INSULATOR HAS MUCH LOWER LEAKAGE THAN A PCB TRACK

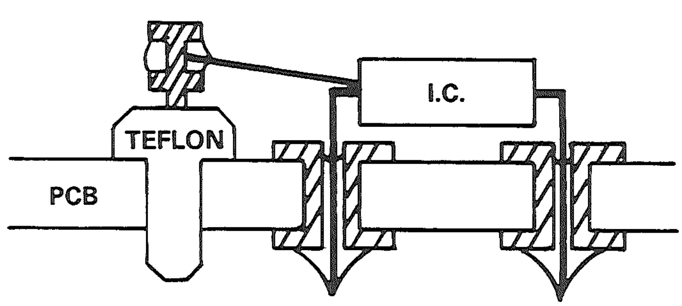


Figure 12.2

ELECTROSTATIC DAMAGE (ESD)

Where resistances are very high, especially in conditions of low humidity, there is always the possibility of electrostatic charge and electrostatic damage. A full discussion of electrostatic damage (ESD) and its prevention will be found in Analog Devices' Application Note on the subject, which is available free of charge from Analog Devices.[1]

This application note describes procedures to minimize the risk of electrostatic damage to sensitive devices. The basic principle of all ESD protection is to prevent a vulnerable item from being in the path of a discharge. Many of the precautions used in factories are designed to minimize the possibility of any damaging discharge, even in the event of carelessness. When experienced engineers handle ICs they may dispense with most of the ESD protection apparatus and merely ensure that the IC is never in any potential discharge path: when taking a circuit from conductive foam touch the foam to equalize charge before touching the circuit, similarly touch the foam with the hand before inserting the circuit in it, and hold your colleague's hand BEFORE passing the IC.

All integrated circuit structures are vulnerable to damage from the high voltages and high peak currents involved in even small electrostatic discharges but precision analog circuits suffer from a special disadvantage - the circuitry used to protect integrated circuit structures from ESD can often degrade the analog accuracy of the circuit where it is employed. Thus we have the choice between high performance and a high degree of protection. Which we choose will depend upon individual circumstances but it is essential to realize that the choice must be made - and if it is made in favour of accuracy then the circuit involved must not be exposed to electrostatic discharge.

A precision analog circuit exposed to ESD may not fail totally, but merely suffer degradation of its analog performance, and possible reduction of life expectancy. When an IC is returned to Analog Devices for failure analysis of inadequate performance the first check that is made when the package is opened is a visual inspection for evidence of electrostatic damage - and this is found in a large percentage of cases.

An interesting example of an unobvious effect of ESD occurred in Finland, where very cold winters produce very low humidity and particularly severe electrostatic problems. A customer complained that the AD549 low bias current BIFET op-amp had poor long-term reliability and that its noise performance deteriorated over a few years of use.

The amplifier was being used as a unity gain buffer with an electrochemical cell and the non-inverting input was connected to a platinum electrode and to nothing else. In use this electrode was immersed in electrolyte but after use it was washed (automatically) in deionized water and air dried. It was then left unconnected until the machine was next used.

Although there was no possibility of the electrode being touched at this time (it was in the very center of the machine) it could encounter random particles of electrostatically charged dust - and the pulse currents as these dust particles discharged were sufficient to cause gradual deterioration of the noise figure. As soon as arrangements were made to ground the electrode when it was not in use (with an NC reed relay for minimum leakage) the problem disappeared.

ELECTROSTATIC DISCHARGE (ESD)

ESD
PREVENTION MANUAL

ANALOG DEVICES

Figure 12.3

ELECTROSTATIC DISCHARGE PROTECTION

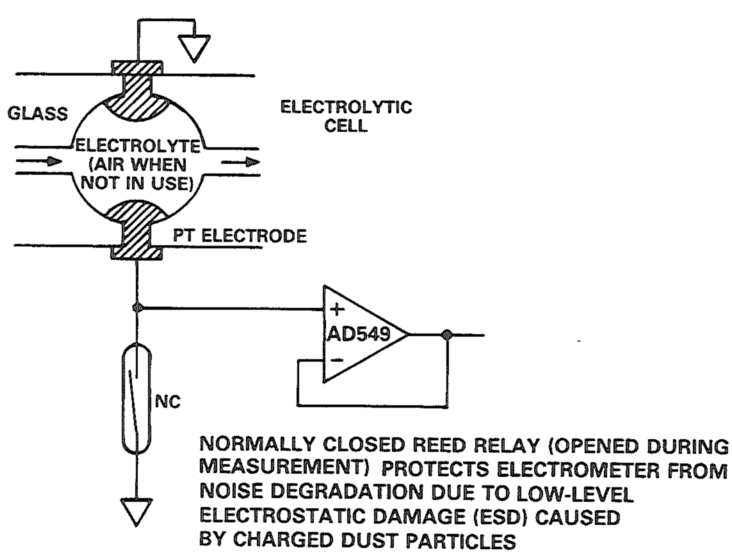


Figure 12.4

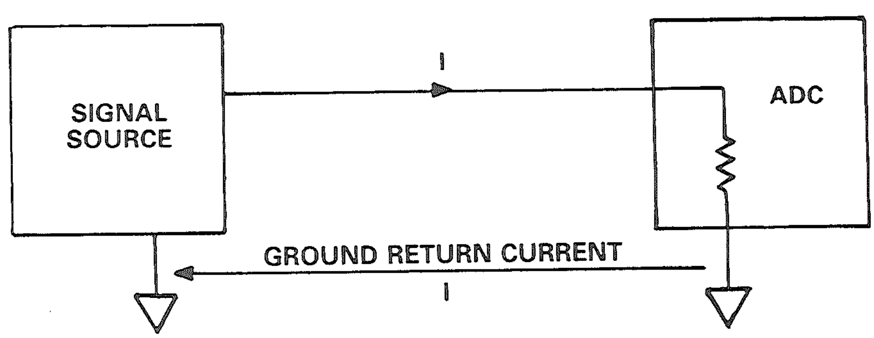
GROUNDING AND SIGNAL ROUTING

SIGNAL RETURN CURRENTS

Kirchoff's Law tells us that at any point in a circuit the algebraic sum of the currents is zero. This tells us that all currents flow in circles and, particularly, that the return current must always be considered when analyzing a circuit.

Most people consider the return current when considering a fully differential circuit, but when considering the more usual circuit where a signal is referred to "ground" it is common to assume that all the points on the circuit diagram where the ground symbol is to be found are at the same potential. This is unwise.

KIRCHOFF'S LAW



AT ANY POINT IN A CIRCUIT
THE ALGEBRAIC SUM OF THE CURRENTS IS ZERO
OR
WHAT GOES OUT MUST COME BACK
WHICH LEADS TO THE CONCLUSION THAT
ALL VOLTAGES ARE DIFFERENTIAL
(EVEN IF THEY'RE GROUNDED)

Figure 12.5

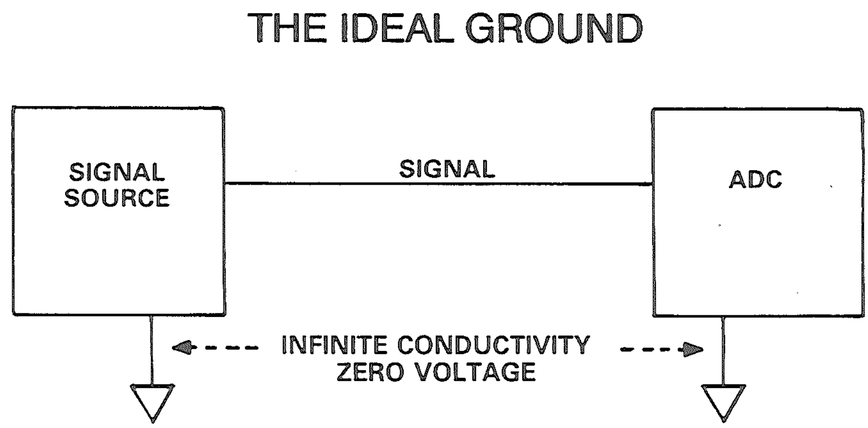


Figure 12.6

GROUND NOISE & GROUND LOOPS

A more realistic model of ground is shown in Figure 12.7. Not only does the return current flow in the complex impedance which exists between the two "ground" points shown in Figure 12.6, giving rise to a voltage drop in the total signal path, but external currents may also flow in the same path, generating uncorrelated noise voltages which are seen by the ADC.

It is evident, of course, that other currents can only flow in the ground impedance if there is a current path for them. Figure 12.7 shows such a path at "ground" potential, which is the notorious "Ground Loop", but equally severe problems could be caused by a circuit sharing an unlooped ground return with the signal source but drawing a large and varying current from its supply and ground return.

It is evident from Figure 12.9 that if a ground network contains loops there is a greater danger of it being vulnerable to EMFs induced by external magnetic fields, and of ground current "escaping" from high current areas to cause noise in sensitive regions. For these reasons ground loops are best avoided.

However, there are situations where looped grounds are unlikely to cause unacceptable noise and the configuration may actually offer benefits in the form of safety or reduced impedance. In such circumstances the optimum ground arrangement may contain loops. Sensible engineers should not allow the almost superstitious dread inspired by the term "ground loop" to prevent the adoption of such designs, if careful analysis and experiment has shown that they actually are optimum.

A MORE REALISTIC GROUND

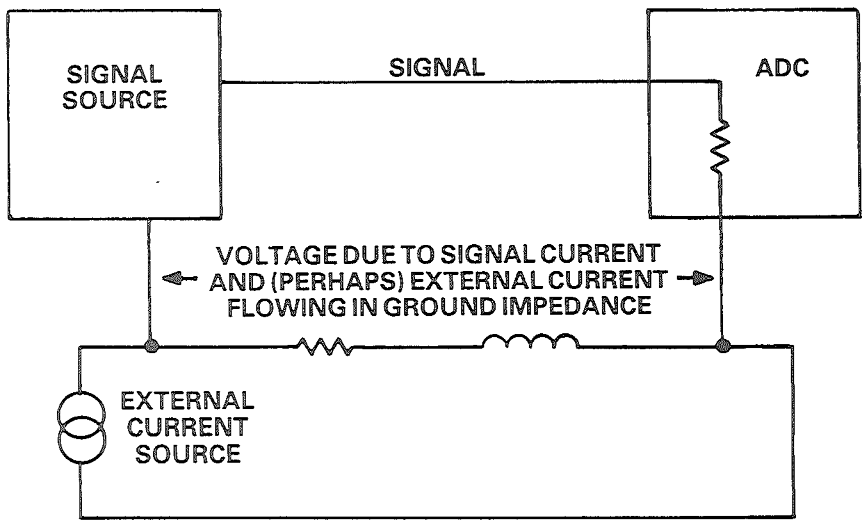


Figure 12.7

ANY CURRENT FLOWING IN A COMMON GROUND MAKES NOISE; A GROUND LOOP IS NOT NECESSARY

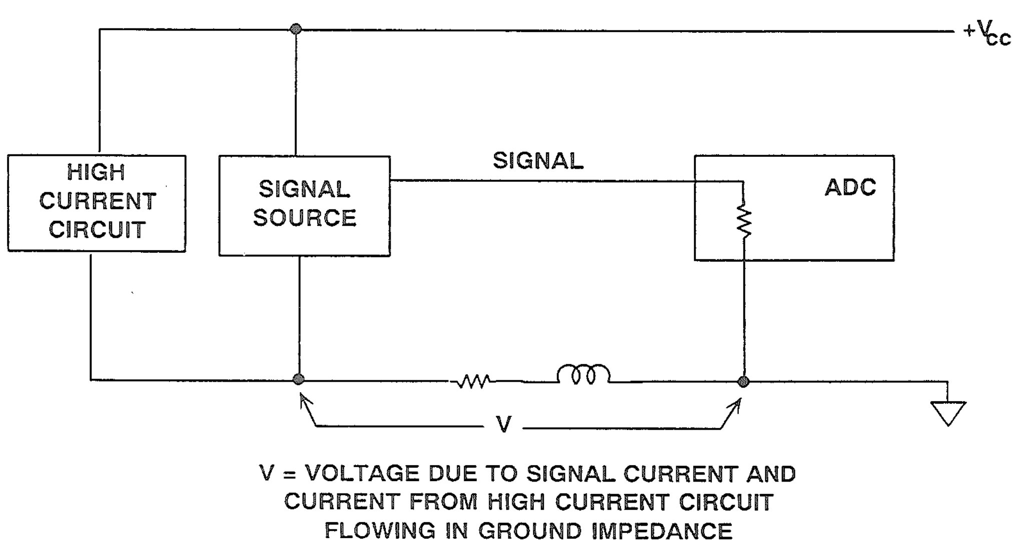


Figure 12.8

GROUND LOOP

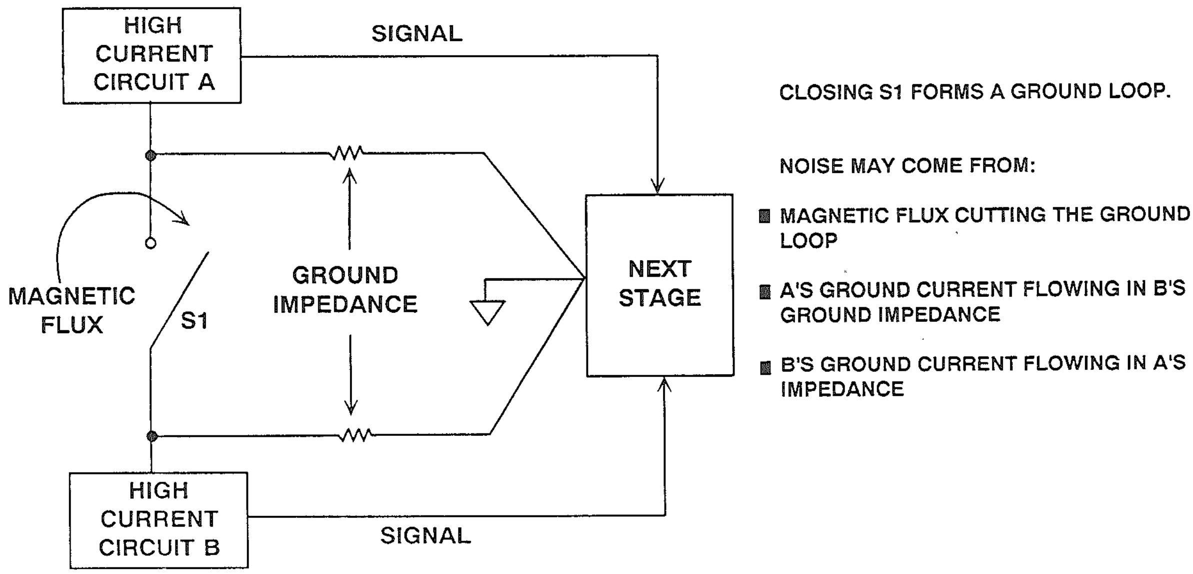


Figure 12.9

There are a number of possible ways of attacking the problem of ground noise, apart from the (presently) impracticable one of using superconducting grounds. It is rare for a single method to be used to the exclusion of all others, and systems generally contain a mixture of approaches. For the purposes of description, however, it is better to describe each approach separately.

Star Grounds

The "star" ground philosophy builds on the theory that there is a single point in a circuit to which all voltages are referred. This is known as the "star" point.

This philosophy is reasonable but frequently encounters practical difficulties. For example if we design a system with a star ground, drawing all the signal paths to minimize signal interaction and the effects of high impedance signal or ground paths, we frequently find, when the power supplies are added to the circuit diagram, that the power supplies either add unwanted ground paths or that supply currents, flowing in existing ground paths, are sufficiently large, or noisy, or both, as to corrupt the signal transmission. This problem may often be avoided by having separate power supplies for different parts of the circuit - separate analog and digital supplies, and separate analog and digital grounds joined at the star point, are common in mixed signal applications.

STAR GROUNDS

- **If all signal voltages in a system are measured with respect to a single point that point is said to be the *star* ground of the system.**

Figure 12.10

SEPARATE ANALOG AND DIGITAL GROUNDS

Digital circuitry is noisy. Saturating logic draws large fast current spikes from its supply during switching and, having noise immunity of hundreds of millivolts or more, has little need of high levels of supply decoupling.

Analog circuitry, on the other hand, is very vulnerable to noise in supplies or grounds. It is therefore sensible to separate analog and digital circuitry to prevent digital noise from corrupting analog performance. Such separation will involve separation of both grounds and power supplies, which may be inconvenient in a mixed signal system. Nevertheless, if a system is to give the full performance of which it is capable it is often essential to have separate analog and digital grounds and power supplies. The fact that some analog circuitry will operate from a single +5 V supply does <u>NOT</u> mean that it may safely be operated from the same noisy +5V supply as the microprocessor and dynamic RAM, the electric fan, and the solenoid jackhammer!

However, analog and digital ground in a system must be joined at some point to allow signals to be referred to a common potential. This star point, or analog/digital common point, is chosen so that it does not introduce digital currents into the ground of the analog part of the system - it is often convenient to make the connection at the power supplies.

Many ADCs and DACs have separate "analog ground" and "digital ground" pins, and users are advised, on the data sheets, to connect these pins together at the device package. This seems to conflict with the advice to connect analog and digital ground at the power supplies, and, in systems with more than one converter, with the advice to join the analog and digital ground at a single point.

SUPPLY & GROUND NOISE

- Digital circuitry is noisy
- Analog circuitry is quiet
- Circuit noise from digital circuitry carried by power and ground leads can corrupt precision analog circuitry
- It is advisable to separate the power and ground of the digital and analog parts of a system
- Analog and digital grounds must be joined at ONE point

Figure 12.11

ANALOG AND DIGITAL GROUND

- Monolithic and hybrid ADCs and DACs frequently have separate AGnd & DGnd pins which must be joined together at the device.
- This is not done from a desire to be difficult, but because the voltage drop in the bondwire is too large to allow the connection to be made internally.
- The best solution to the grounding problem arising from this requirement is to connect both pins to system "analog ground."
- It is likely that neither the digital noise so introduced in the system AGnd, nor the loss of digital noise immunity, will seriously affect the system performance.

Figure 12.12

There is, in fact, no conflict. The labels "analog ground" and "digital ground" on these pins refer to the parts of the converter to which the pins are connected, and not to the system grounds to which they must go. In general these two pins should be joined together and to the analog ground of the system. It is not possible to join the two pins within the IC package because the analog part of the converter cannot tolerate the voltage resulting from the digital current flowing in the bond wire to the chip.

If these pins are connected in this way the digital noise immunity of the converter is diminished by the amount of common- mode noise between the digital and analog system grounds. Since digital noise immunity is of the order of hundreds or thousands of millivolts this is unlikely to be important.

The analog noise immunity is diminished only by the external digital currents of the converter itself flowing in the analog ground. These currents should be quite small, and can be minimized by ensuring that the converter outputs do not drive large fanouts. If the logic supply to the converter is isolated with a small resistance and decoupled to analog ground with a 0.1 µF capacitor sited as close to the converter as possible all the internal digital currents of the converter will return to ground through the capacitor and will not appear in the external ground circuit. If the analog ground impedance is as low as it should be for adequate analog performance the additional noise due to the external digital ground current should rarely present a problem.

ANALOG GROUND (AGND)
AND DIGITAL GROUND (DGND)
OF ADCs/DACs
SHOULD BE RETURNED
TO SYSTEM ANALOG GROUND

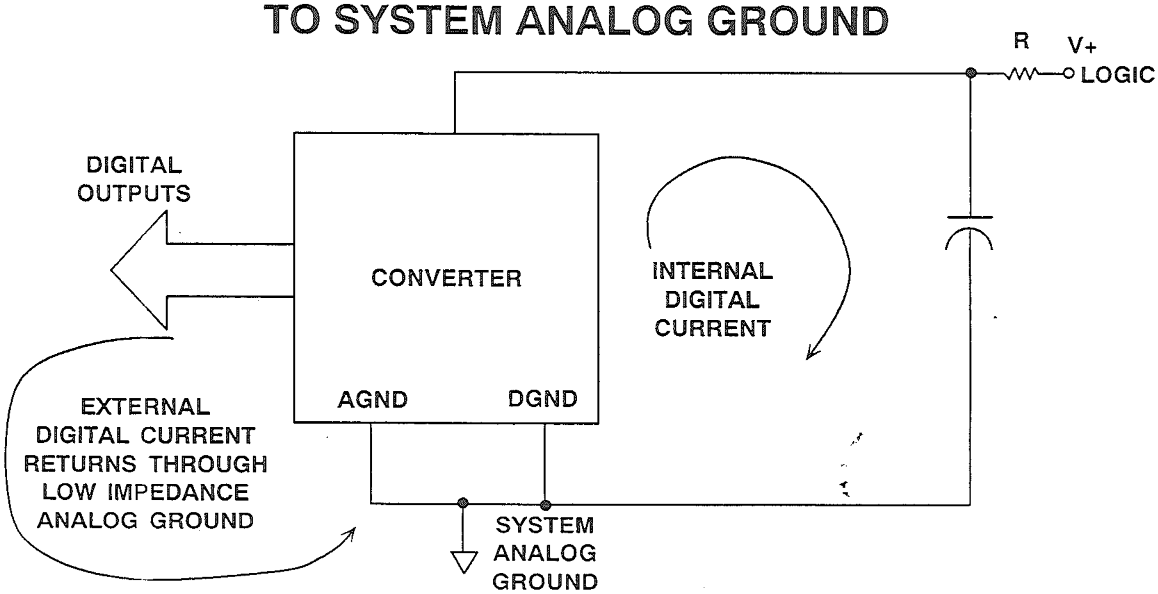


Figure 12.13

GROUND PLANES

Related to the star ground system is the use of a ground plane. One side of a double-sided PCB, or one layer of a multi-layer one, is made of continuous metal, which is used as ground. The theory behind this is that the large amount of metal will have low resistance and as low inductance as is possible.

It is sometimes argued that ground planes should not be used because they are liable to introduce problems in manufacture and assembly. Such an argument may have had a limited validity twenty years ago when PCB adhesives were less well developed, wave-soldering less reliable, and solder resist techniques less well understood, but today it should not be tolerated.

Ground planes solve many ground impedance problems, but not all. Even a continuous sheet of copper foil has residual resistance and inductance and in some circumstances they can be enough to prevent proper circuit function. Figure 12.15 shows such a problem - and a possible solution.

GROUND PLANES

- One entire side or layer of a PCB is continuous grounded conductor.
- This gives minimum ground resistance and inductance but is not always sufficient to solve all grounding problems.
- Breaks in ground planes can improve or degrade circuit performance--there is no general rule.
- Twenty years ago ground planes were difficult to fabricate. Today they are not.
- If your PCB facility objects to fabricating ground planes --- GET A NEW PCB FACILITY!

Figure 12.14

A SLIT IN A GROUND PLANE CAN RECONFIGURE CURRENT FLOW FOR BETTER ACCURACY

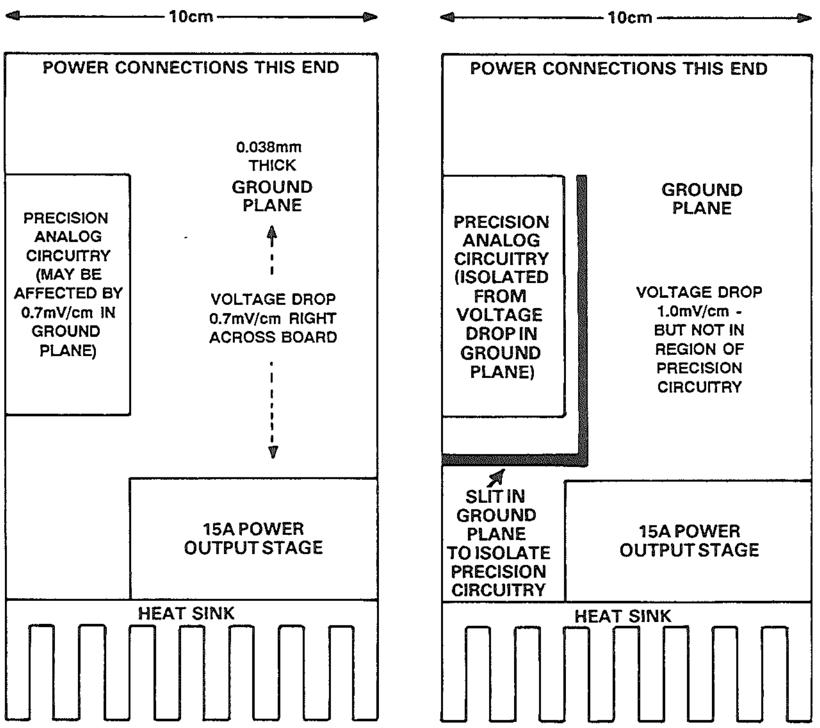


Figure 12.15

Consider a ground-plane PCB 100 mm wide with a ground connection at one end and a power amplifier at the other drawing 15A. If the ground plane is 0.038 mm thick and 15 A flows in it there will be a voltage drop of 68 µV/mm. This voltage drop would cause quite serious problems to any ground-referenced precision circuitry sharing the PCB. However, if we slit the ground plane so that high current does not flow in the region of the precision circuitry we can possibly solve the problem - even though the voltage gradient will increase in those parts of the ground plane where the current does flow.

TRANSMISSION LINES

A break in a ground plane is not always a good thing. We earlier considered the benefits of outward and return signal paths being close together so that inductance is minimized. As we saw in Figure 11.7, when an HF signal flows in a PC track running over a ground plane the arrangement functions as a microstrip transmission line and the majority of the return current flows in the ground plane underneath the line.

The characteristic impedance of the line will depend upon the width of the track and the thickness and dielectric constant of the PCB material. For most lower frequency applications the characteristic impedance will be unimportant, as the line will not be correctly terminated, but at UHF and higher it is possible to use PCB tracks as microstrip transmission lines in properly terminated systems. If losses in such systems are to be minimized the PCB material must be chosen for low high frequency loss. This usually means the use of expensive teflon PCB material.

MICROSTRIP TRANSMISSION LINE

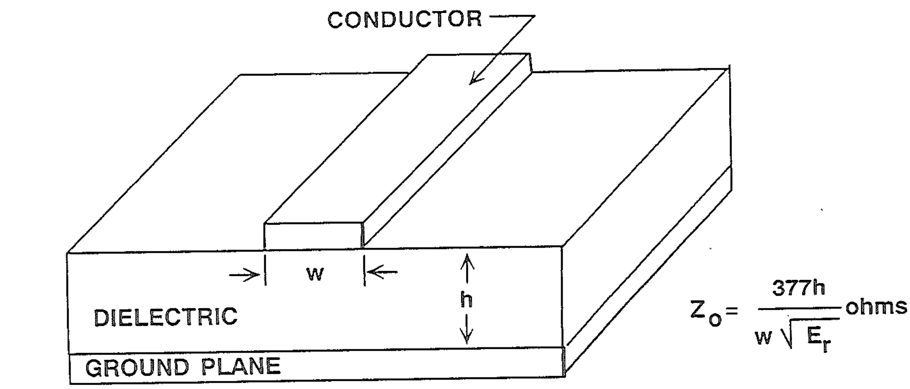


Figure 12.16

BREAKS IN GROUND PLANE RAISE INDUCTANCE

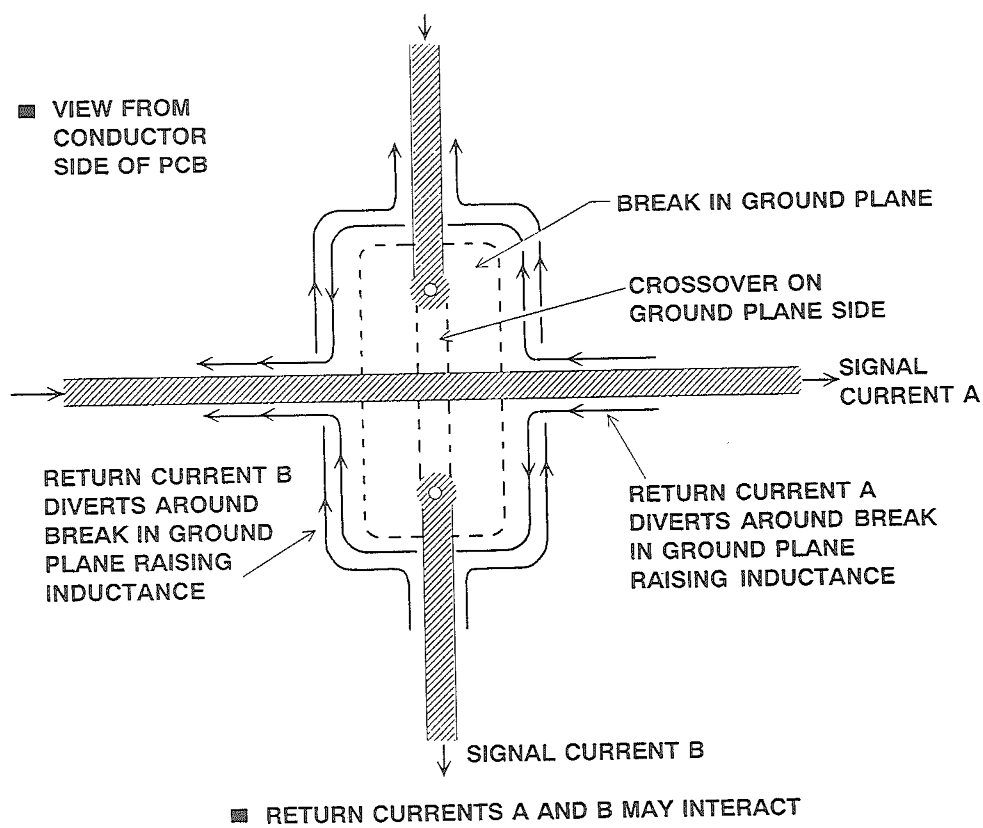


Figure 12.17

Where there is a break in the ground plane under a conductor the return current must flow around the break and both the inductance and the vulnerability of the circuit to external fields are increased.

Where such a break is made to allow a crossover of two perpendicular conductors it would be far better if the second signal were carried across both the first and the ground plane by means of a piece of wire. The ground plane then acts as a shield between the two signal conductors, and the two ground return currents, flowing in opposite sides of the ground plane as a result of skin effects, do not interact.

With a multi-layer board both the crossover and the continuous ground plane can be accommodated without the need for a wire link. Multi-layer PCBs are expensive and harder to trouble-shoot than simple double-sided boards but do offer even better shielding and signal routing. The principles involved remain unchanged but the range of layout options is increased.

Use of double-sided or multi-layer board with at least one continuous ground plane is undoubtedly one of the most successful approaches to the design of high performance mixed signal circuitry. Often the impedance of the ground plane is sufficiently low to permit the use of a single ground plane for both analog and digital parts of the system, but this does depend upon the resolution and bandwidth required and the amount of digital noise in the system.

SYSTEM GROUNDS

In systems where there are several PCBs grounding may be more of a problem. At first sight it would appear that the problem is similar to that of a single PCB where particular subsystems must be positioned so that large ground currents do not flow where ground noise must be minimized - in a multicard system the grounds of individual PCBs must be interconnected so that such harmful interactions are minimized.

There are three problems with this. First of all there is far less opportunity for rearranging the physical layout of a system consisting of a few cards connected to a common backplane. Secondly many multicard systems are designed to be reconfigured in a “mix ‘n’ match” arrangement to allow large numbers of system options - it can be impossible to predict what systems are going to be required and to ensure that all of them are noise free. Finally, multicard systems are likely to have higher ground currents than occur on single, relatively simple, PCBs - but these currents must flow in the higher impedances which are associated with the intercard connectors even when multiple ground pins are used.

The basic principles still apply: ground impedance must be as low as possible, high level and low level signals must be separated so that they do not interfere with each other, and capacitance and mutual inductance coupling must be avoided. Nevertheless, it must be accepted that situations can arise where it is not possible to transfer a high speed, high accuracy signal from one PCB to another without unacceptable signal degradation.

MULTIPLE CARD SYSTEMS

- **Multiple card systems are likely to have higher ground currents and higher ground impedances than are found on a single PCB.**
- **It is therefore more difficult to transfer ground-referenced signals accurately between cards than across a PCB.**
- **In some cases it will be IMPOSSIBLE to transfer ground-referenced signals between PCBs without unacceptable loss of quality.**

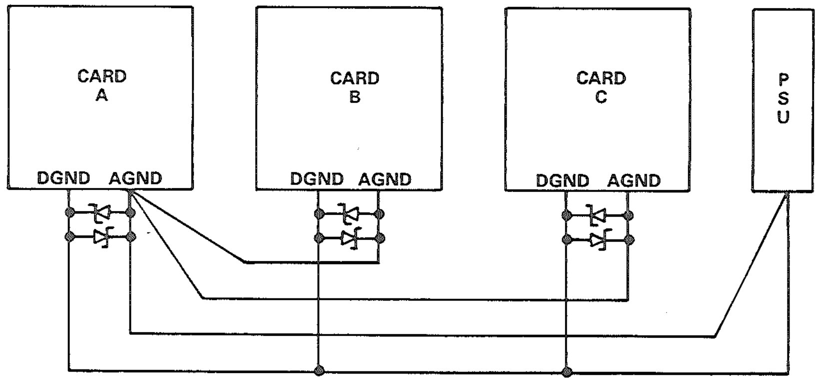
Figure 12.18

The best way of minimizing ground impedance in a multicard system is to use another PCB as a backplane and have a ground plane (or even two - one analog, one digital) on that mother card. If the earlier advice about multiple ground pins has been observed this arrangement is capable of excellent performance. Where there are several card cages (racks for PCBs) the ground planes of the several mother boards must be tied together and, probably, to the metal chassis holding the card cages - the exact layout of the inter-connections will depend on the overall system architecture.

If a mother board with a ground plane is not possible then the ground pins of the PCB sockets must be wired together, with due attention to probable current flows and common ground impedances, with heavy, multi-strand wire, having as low resistance as possible. In many cases the resulting ground screen will be tied to chassis ground at a number of points but it will sometimes be better to join them at a single star point.

It is not just the ground layout that is important in high performance mixed signal systems, the siting of different subsystems and the routing of signals is most important in determining overall system performance.

STAR ANALOG GROUND IN A MULTICARD SYSTEM



- Schottky diodes protect cards in the event of loss of analog ground
- This grounding system may be inadequate at high resolution or where large ground currents flow
- This MAY permit accurate intercard transmission of ground referenced signals

Figure 12.19

SIGNAL ROUTING

It is evident that we can minimize noise by paying attention to the system layout and preventing different signals from interfering with each other. High level analog signals should be separated from low level analog signals, and both should be kept away from digital signals. We have seen elsewhere that in waveform sampling and reconstruction systems the sampling clock (which is a digital signal) is as vulnerable to noise as any analog signal, but is as liable to cause noise as any digital signal, and so must be kept isolated from both analog and digital systems.

If a ground plane is used, as it should in be most cases, it can act as a shield where sensitive signals cross. Figure 12.21 shows a good layout for a data acquisition system where all sensitive areas are isolated from each other and signal paths are kept as short as possible. While real life is rarely as tidy as this the principle remains a valid one.

There are a number of important points to be considered when making signal and power connections. First of all a connector is one of the few places in the system where all signal conductors must run parallel - it is therefore a good idea to separate them with ground pins to reduce coupling between them.

Multiple ground pins are important for another reason: they keep down the ground impedance at the junction between the board and the backplane. The contact resistance of a single pin of a PCB connector is quite low (of the order of 10 mOhms) when the board is new - as the board gets older the contact resistance is likely to rise, and the board's performance may be compromised. It is therefore well worthwhile to afford extra PCB connector pins so that there are many ground connections (perhaps 20-30% of all the pins on the PCB connector should be ground pins). For similar reasons there should be several pins for each power connection,

SIGNAL ROUTING IN MIXED SIGNAL SYSTEMS

- Physically separate analog and digital signals.
- Avoid crossovers between analog and digital signals.
- Be careful with sampling clock and ADC analog input runs.
- Be careful with high impedance points.
- Use lots of ground plane.
- Use microstrip techniques for controlled impedances.

Figure 12.20

PCB FLOWCHART

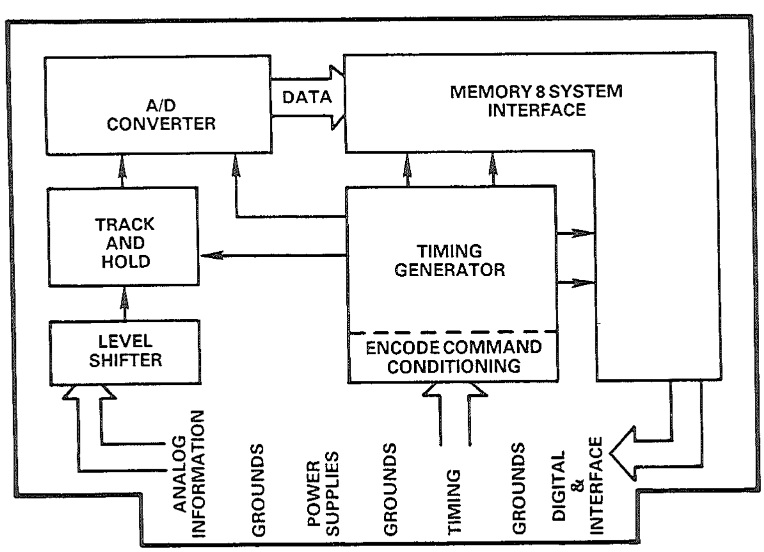


Figure 12.21

EDGE CONNECTIONS

- Separate sensitive signals by ground pins.
- Keep down ground impedance with multiple (20-30% of total) ground pins.
- Have several pins for each power line.
- Critical signals may require a separate connector (possibly coax).

Figure 12.22

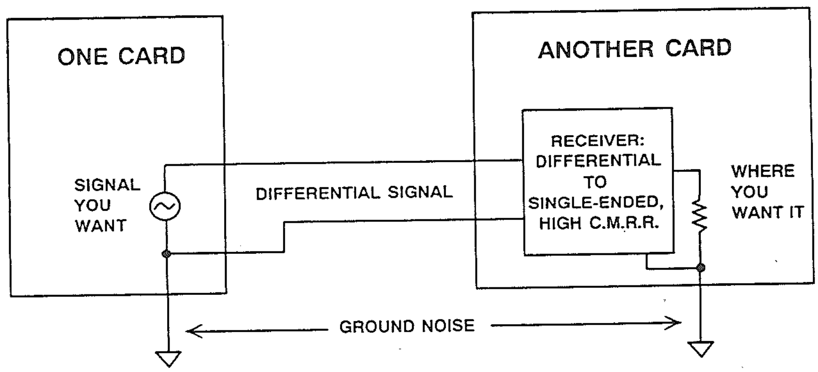
although there is no need to have as many as there are ground pins.

Modern high performance mixed signal systems handle signals with resolutions of 8 bits at sampling rates of over 500 MHz and resolutions of 14 bits sampled at more than 10 MHz. Preserving signal integrity between cards in a multi-card system is extremely difficult at such performance levels and may be impossible.

The use of balanced transmission lines can help but if the signal bandwidth extends to DC there will be a need for a very high performance instrumentation amplifier at the receiving end to restore a ground referenced signal.

The best, and in many cases the only, solution to problems of this sort, is to partition the system so that the highest quality signals are not transferred between boards.

DIFFERENTIAL TRANSMISSION MINIMIZES GROUND ERRORS



- At DC and LF the receiver will be an instrumentation amplifier
- At HF the receiver will be a transformer
- There is no ideal receiver for video signals which have components from DC to HF

Figure 12.23

VIDEO SIGNAL TRANSMISSION

- It is often IMPOSSIBLE to transmit very broadband high accuracy signals between the PCBs of a multicard system without unacceptable loss of quality.
- In such cases the system must be reconfigured to allow all the analog processing to take place on a single PCB.
- It may be inconvenient, but it's the only way you'll get it to work!

Figure 12.24

PROBLEM AREAS

LIMITATIONS OF SPICE MODELLING

As we have seen, real electronic circuits contain many "components" which were not present in the circuit diagram but which are there because of the physical properties of conductors, circuit boards, IC packages, etc. These components are difficult, if not impossible, to incorporate into computer modelling software and yet they have substantial effects on circuit performance at high resolutions, or high frequencies, or both.

It is therefore inadvisable to use SPICE modelling or similar software to predict the ultimate performance of such high performance analog circuits. After modelling is complete the performance must be verified by experiment.

This is not to say that SPICE modelling is valueless - far from it. Most modern high performance analog circuits could never have been developed without the aid of SPICE and similar programs, but it must be remembered that such simulations are only as good as the models used and these models are not perfect. We have seen the effects of parasitic components arising from the conductors, insulators and components on the PCB, but it is also necessary to appreciate that the models used within SPICE simulations are not perfect models.

Consider an operational amplifier. It contains some 20-40 transistors, almost as many resistors, and a few capacitors. A

SPICE MODELLING

- **SPICE modelling is a powerful tool for predicting the performance of analog circuits.**

HOWEVER

- **Models omit real-life effects**
- **No model can simulate all the parasitic effects of discrete components and a PCB layout.**

THEREFORE

- **Prototypes must be built and proven before production.**

Figure 12.25

12

complete SPICE model will contain all these components and probably a few of the more important parasitic capacitances and spurious diodes formed by the diffusions in the op-amp chip. This is the model that the designer will have used to evaluate the device during his design. In simulations such a model will behave very like the actual op-amp, but not exactly.

However, this model is not published, as it contains too much information which would be of use to other semiconductor companies who might wish to copy or improve on the design. It would also take far too long for a simulation of a system containing such models of a number of op-amps to reach a useful result. For these, and other, reasons the SPICE models of analog circuits published by manufacturers or software companies are "macro" models, which simulate the major features of the component but lack some of the fine detail. Consequently SPICE modelling does not always reproduce the exact performance of a circuit and should always be verified experimentally.

SOCKETS

It is tempting to mount expensive ICs in sockets rather than soldering them in circuit - especially during circuit development. Engineers would do well not to succumb to this temptation.

USE OF SOCKETS WITH HIGH PERFORMANCE ANALOG CIRCUITS

- DON'T! (If at all possible)
- Use "Pin sockets" or "Cage jacks" such as Amp Part No: 5-330808-3 or 5-330808-6 (Capped & uncapped respectively).
- Always test the effect of sockets by comparing system performance with and without the use of sockets.
- Do not change the type of socket used without evaluating the effects of the change on performance.

Figure 12.26

Sockets add resistance, inductance and capacitance to the circuit and may degrade performance to quite unacceptable levels. When this occurs, though, it is always the IC manufacturer who is blamed - not the use of a socket. Even low profile, low insertion force sockets cannot be relied upon not to degrade the performance of high performance (high speed or high precision or, worst of all, both) devices, and as the socket ages and the board suffers vibration the contact resistance of low insertion force sockets is very likely to rise. Where a socket must be used the least loss of performance is achieved by using individual pin sockets (sometimes called "cage jacks") to make up a multi-pin socket in the PCB itself.

It really is best not to use IC sockets with high performance analog and mixed signal circuits. If their use can be avoided it should be. However at medium speeds and medium resolutions the trade-off between performance and convenience may fall on the side of convenience. It is very important, when sockets are used, to evaluate circuit performance with and without the socket chosen to ensure that the type of socket chosen really does have minimal effect on the way that the circuit behaves. The effects of a change of socket on the circuit should be evaluated as carefully as a change of IC would be and the drawings should be prepared so that the change procedures for a socket are as rigorous as for an IC - in order to prevent a purchase clerk who knows nothing of electronics from devastating the system performance in order to save five cents on a socket.

PROTOTYPING HIGH PERFORMANCE ANALOG CIRCUITRY

As we have seen, circuit board layout is part of the circuit design of all high performance analog circuits. Prototyping techniques derived from the "node" theory, while ideal for logic breadboarding at low and medium speeds, are quite unsuitable for any analog circuits, or even for very fast digital ones. Vector board and wire wrap prototyping will tell an engineer nothing about the behavior of a properly laid out version of the analog circuit.

The best technique for analog prototyping is to use a prototype of the final PCB - certainly no design is complete until the final PCB layout has been proved to give the required performance. Nevertheless this approach may be a little limiting where a number of different possibilities are to be evaluated, or for a multicard system.

In this case components should be mounted on a board having a continuous copper ground plane (ideally on both sides of the board, though while convenient this is not essential), with ground connections made to the plane and short point to point wiring made above and below it. The overall component placing and signal routing should be as close as possible to the planned final layout.

As we have already indicated, IC sockets can degrade the performance of analog ICs. While directly soldered components are ideal for prototyping, an IC socket made of pin sockets mounted in the ground plane board may be acceptable (clear the copper, on both sides of the board, for about 0.5 mm around each ungrounded pin socket - solder the grounded ones to ground on both sides of the board).

Allowing wiring to float in the air can be a little tricky. There is a breadboarding system which is conceptually very similar to that described above but which provides adhesive PC pads which stick to the ground plane and allow more rigid component mounting and wiring. This system is manufactured by Wainwright Instru-

12

PROTOTYPING MIXED SIGNAL CIRCUITRY

- NEVER use vector boards or wire-wrap for the analog parts of the system (they can be invaluable for data buses and address lines in the digital part).
- Wherever possible avoid the use of sockets for analog ICs.
- Use a prototype of your final PCB layout as early as possible.

Figure 12.27

ments and is known as "Minimount" in Europe and "Solder Mounts" in the USA. The manufacturer's and distributors' addresses are given in the references at the end of this section.[2]

Manufacturer's evaluation boards are also useful in system prototyping since they have already been optimized for best performance. Analog Devices offers many evaluation boards for a wide array of products. They offer the designer an excellent starting point for the layout.

When the prototype layout is transferred to a CAD system for PCB layout it is important to disable, or at any rate override where necessary, any automatic routing or component placing software. The criteria used by such software are more closely related to "node" theory and aesthetically pleasing rows of components (which, admittedly, are also easier on automatic component placing machinery) than to optimizing stray inductance and capacitance and minimizing common ground impedances.

ADDITIONAL PROTOTYPING HINTS

- Pay *equal* attention to signal routing, component placing and supply decoupling in *both* the prototype and the final design.
- Verify performance as well as functionality at each stage of the design.
- For "freehand" prototyping use a copper-clad board, mount components to it by their ground pins and wire the remaining connections point-to-point (use Wainwright Instruments' Minimount/Solder Mount adhesive PC pads if aerial point-to-point wiring seems too fraught with peril).

Figure 12.28

NOTES

1 "E.S.D. Prevention Manual"
Available free from Analog Devices.

2 Wainwright Instruments Inc., 7770 Regents Rd., #113, Suite 371, San Diego, CA 92122, Tel: 619-558-1057. Wainwright Instruments GmbH, Widdersberger Strasse 14, DW-8138 Andechs-Frieding, Germany. Tel: +49-8152-2245, Fax: +49-8152-5174.