James M. Bryant

9. Analog Breadboarding

Introduction

While there is no doubt that computer analysis is one of the most valuable tools that the analog designer has acquired in the last decade or so, there is equally no doubt that analog circuit models are not perfect and must be verified with hardware. If the initial test circuit or "breadboard" is not correctly constructed it may suffer from malfunctions which are not the fault of the design but of the physical structure of the breadboard itself. This chapter considers the art of successful breadboarding of high-performance analog circuits.

The successful breadboarding of an analog circuit which has been analyzed to death in its design phase has the reputation of being a black art which can only be acquired by the highly talented at the price of infinite study and the sacrifice of a virgin or two. Analog circuitry actually obeys the very simple laws we learned in the nursery: Ohm's Law, Kirchoff's Law, Lenz's Law and Faraday's Laws. The problem, however, lies in Murphy's Law.

Murphy's Law is the subject of many engineering jokes, but in its simplest form, "If Anything Can Go Wrong—It Will!", it states the simple truth that physical laws do not cease to operate just because we have overlooked or ignored them. If we adopt a systematic approach to breadboard

MURPHY'S LAW

Whatever can go wrong, will go wrong.

Buttered toast, dropped on a sandy floor,
falls butter side down.

The basic principle behind Murphy's Law is that
all physical laws always apply -
when ignored or overlooked they do not stop working.

Figure 9–1.

construction it is possible to consider likely causes of circuit malfunction without wasting very much time.

In this chapter we shall consider some simple issues which are likely to affect the success of analog breadboards, namely resistance (including skin effect), capacitance, inductance (both self inductance and mutual inductance), noise, and the effects of careless current routing. We shall then discuss a breadboarding technique which allows us to minimize the problems we have discussed.

Resistance

As an applications engineer I shall be relieved when room-temperature superconductors are finally invented, as too many engineers suppose that they are already available, and that copper is one of them. The assumption that any two points connected by copper are at the same potential completely overlooks the fact that copper is resistive and its resistance is often large enough to affect analog and RF circuitry (although it is rarely important in digital circuits).

CONDUCTORS ARE NOT *SUPER*CONDUCTORS

Consider 10 cm of 1 mm PC track

Standard track thickness is 0.038 mm

ρ for copper is $1.724 \times 10^{-6}\ \Omega$ cm @ 25°C

$\therefore$ PCB sheet resistance is 0.45 mΩ/sq

Resistance of the track is 45 mΩ

THIS IS ENOUGH TO MATTER!

Figure 9–2.

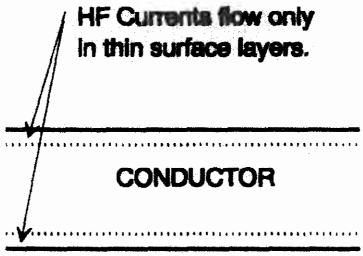
The diagram in Figure 9–2 shows the effect of copper resistance at DC and LF. At HF, matters are complicated by "skin effect." Inductive effects cause HF currents to flow only in the surface of conductors. The skin depth (defined as the depth at which the current density has dropped to 1/e of its value at the surface) at a frequency f is

$$\frac{1}{\sqrt{\mu\sigma\pi f}}$$

where μ is the permittivity of the conductor, and σ is its conductivity in Ohm-meters. $\mu = 4\pi\times10^{-7}$ henry/meter except for magnetic materials, where $\mu=4\mu_r\pi\times10^{-7}$ henry/meter (μ_r is the relative permittivity). For the purposes of resistance calculation in cases where the skin depth is less than one-fifth the conductor thickness, we can assume that all the HF current flows in a layer the thickness of the skin depth, and is uniformly distributed.

SKIN EFFECT

At high frequencies inductive effects cause currents to flow only in the surface of conductors.



Skin depth at frequency f in a conductor of resistivity ρ ohm-metre and permittivity μ henry/metre is

$$\sqrt{\frac{\rho}{\mu\pi f}}$$

In copper the skin depth is $6.61/\sqrt{f}$ cm and

the skin resistance is $2.6\text{X}10^{-7}\sqrt{f}$ Ω/sq

(Remember that current *may* flow in both sides of a PCB [this is discussed later] and that the skin resistance formula is only valid if the skin depth is less than the conductor thickness.)

Figure 9–3.

Skin effect has the effect of increasing the resistance of conductors at quite modest frequencies and must be considered when deciding if the resistance of wires or PC tracks will affect a circuit's performance. (It also affects the behavior of resistors at HF.)

Capacitance

Good HF analog design must incorporate stray capacitance. Wherever two conductors are separated by a dielectric there is capacitance. The formulae for parallel wires, concentric spheres and cylinders, and other more exotic structures may be found in any textbook but the commonest structure, found on all PCBs, is the parallel plate capacitor.

CAPACITANCE

Wherever two conductors are separated by a dielectric (including air or a vacuum) there is capacitance.

For a parallel plate capacitor $C = .0885 E_r A / d$ pF

where A is the plate area in sq.cm
d is the plate separation in cm
& E_r is the dielectric constant

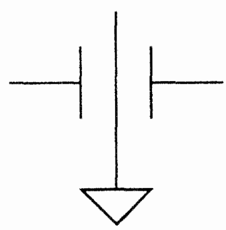
Epoxy PCB material is often 1.5 mm thick and E_r =4.7
Capacity is therefore approximately 2.8 pf/sq.cm

Figure 9–4.

When stray capacitance appears as parasitic capacity to ground it can be minimized by careful layout and routing, and incorporated into the design. Where stray capacity couples a signal where it is not wanted the effect may be minimized by design but often must be cured by the use of a Faraday shield.

Figure 9–5.

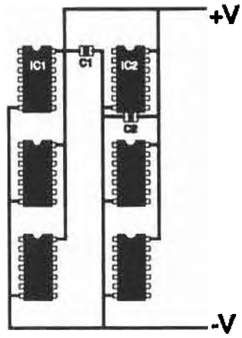
FARADAY SHIELDS



Capacitively coupled noise can be very effectively shielded by a grounded conductive shield, known as a Faraday Shield. But it must be grounded or it increases the problem.
For this reason coil and quartz crystal cans should always be grounded.

If inductance is to be minimized the lead and PC track length of capacitors must be kept as small as possible. This does not mean just generally "short," but that the inductance in the actual circuit function must be minimal. Figure 9–6 shows both a common mistake (the leads of the capacitor C1 are short, but the decoupling path for IC1 is very long) and the

CAPACITOR LEADS MUST BE SHORT



Although the leads of C1 are short the HF decoupling path of IC1 is far too long. The decoupling path of IC2 is ideal.

Figure 9–6.

correct way to decouple an IC (IC2 is decoupled by C2 with a very short decoupling path).

Inductors

Any length of conductor has inductance and it can matter. In free space a 1cm length of conductor has inductance of 7–10nH (depending on diameter), which represents an impedance of 4–6Ω at 100MHz. This may be large enough to be troublesome, but badly routed conductors can cause worse problems as they form, in effect, single turn coils with quite substantial inductance.

INDUCTANCE

Any conductor has some inductance

A straight wire of length L and radius R (both mm & L>>R)

has inductance $0.2L\left[\ln\left(\frac{2L}{R}\right) - .75\right]$ nH

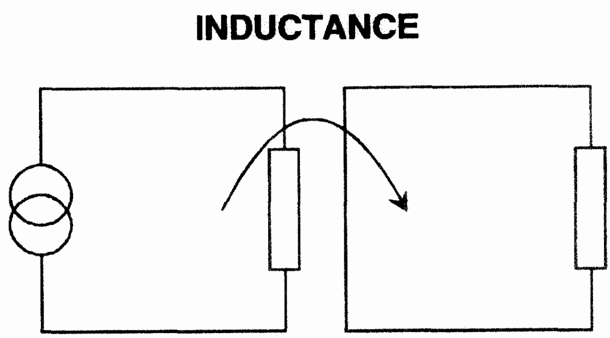
A strip of conductor of length L, width W and thickness H (mm) has inductance

$$0.2L\left[\ln\left(\frac{2L}{W+H}\right) + 0.2235\left(\frac{W+H}{L}\right) + 0.5\right] \text{nH}$$

1 cm of thin wire or PC track is somewhere between 7 and 10 nH

Figure 9–7.

Figure 9–8.

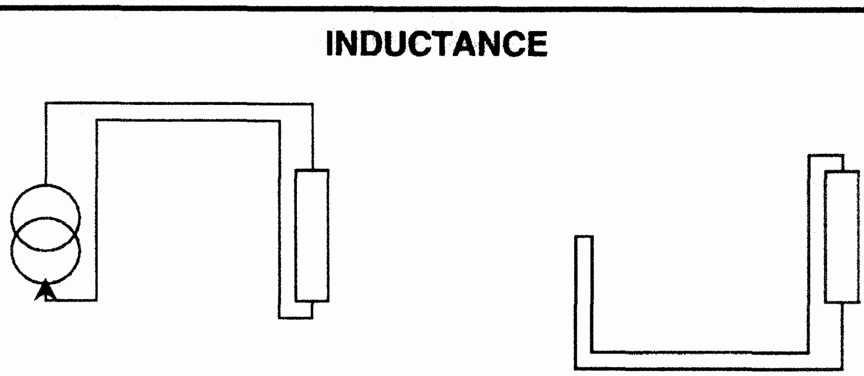


A loop of conductor has inductance - two adjacent loops have mutual inductance.

If two such coils are close to each other we must consider their mutual inductance as well as their self-inductance. A change of current in one will induce an EMF in the other. Defining the problem, of course, at once suggests cures: reducing the area of the coils by more careful layout, and increasing their separation. Both will reduce mutual inductance, and reducing area reduces self inductance too.

It is possible to reduce inductive coupling by means of shields. At LF shields of mu-metal are necessary (and expensive, heavy and vulnerable to shock, which causes loss of permittivity) but at HF a continuous Faraday shield (mesh will not work so well here) blocks magnetic fields too, provided that the skin depth at the frequency of interest is much less

Figure 9–9.



Inductance is reduced by reducing loop area - mutual inductance is reduced by reducing loop area and increasing separation.

Since the magnetic fields around coils are dipole fields they attenuate with the *cube* of the distance - so increasing separation is a very effective way of reducing mutual inductance.

Figure 9–10.

MAGNETIC SHIELDS
At LF magnetic shielding requires Mu-Metal which is heavy, expensive and vulnerable to shock.
At HF a conductor provides effective magnetic shielding provided the skin depth is less than the conductor thickness.
PC foil is an effective magnetic shield above 10-20 MHz.

than the thickness of the shield. In breadboards a piece of copper-clad board, soldered at right angles to the ground plane, can make an excellent HF magnetic shield, as well as being a Faraday shield.

Magnetic fields are dipole fields, and therefore the field strength diminishes with the *cube* of the distance. This means that quite modest separation increases attenuation a lot. In many cases physical distance is all that is necessary to reduce magnetic coupling to acceptable levels.

Grounds

Kirchoff's Law tells us that return currents in ground are as important as signal currents in signal leads. We find here another example of the "superconductor assumption"—too many engineers believe that all points marked with a ground symbol on the circuit diagram are at the same potential. In practice ground conductors have resistance and inductance—and potential differences. It is for this reason that such breadboarding techniques as matrix board, prototype boards (the ones where you poke component leads into holes where they are gripped by phosphor-bronze contacts) and wire-wrap have such poor performance as analog prototyping systems.

The best analog breadboard arrangement uses a "ground plane"—a layer of continuous conductor (usually copper-clad board). A ground

Figure 9–11.

KIRCHOFF'S LAW
The net current at any point in a circuit is zero.
OR
What flows in flows out again.
OR
Current flows in circles.
THEREFORE
All signals are differential.
AND
Ground impedance matters.

plane has minimal resistance and inductance, but its impedance may still be too great at high currents or high frequencies. Sometimes a break in a ground plane can configure currents so that they do not interfere with each other; sometimes physical separation of different subsystems is sufficient.

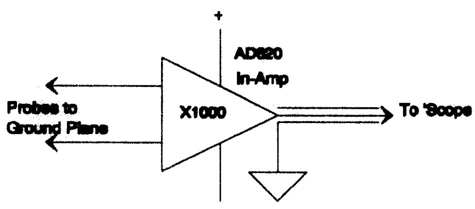
Figure 9–12.

GROUND PLANE BREADBOARD

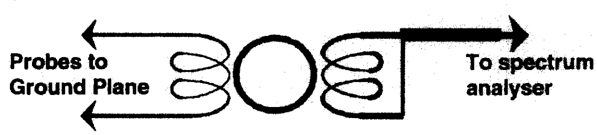
The breadboard ground consists of a single layer of continuous metal, usually (unetched) copper-clad PCB material. In theory all points on the plane are at the same potential, but in practice it may be necessary to configure ground currents by means of breaks in the plane, or careful placement of sub-systems. Nevertheless ground plane is undoubtedly the most effective ground technique for analog breadboards.

Figure 9–13.

GROUND PLANE



NOTE: Oscilloscope, in-amp power ground and ground plane must be common for bias currents. Some Common-mode voltage does not matter.



To measure voltage drop in ground plane it is necessary to use a device with high common-mode rejection and low noise. At DC and LF an instrumentation amplifier driving an oscilloscope will give sensitivity of up to 5 μV/cm - at HF and VHF a transmission line transformer and a spectrum analyser can provide even greater sensitivity.

It is often easy to deduce where currents flow in a ground plane, but in complex systems it may be difficult. Breadboards are rarely that complex, but if necessary it is possible to measure differential voltages of as little as 5µV on a ground plane. At DC and LF this is done by using an instrumentation amplifier with a gain of 1,000 to drive an oscilloscope working at 5 mV/cm. The sensitivity at the input terminals of the inamp is 5µV/cm; there will be some noise present on the oscilloscope trace, but it is quite possible to measure ground voltages of the order of 1µV with such simple equipment. It is important to allow a path for the bias current of the inamp, but its common-mode rejection is so good that this bias path is not critical.

The upper frequency of most inamps is 25–50kHz (the AD830 is an exception—it works up to 50 MHz at low gains, but not at ×1,000). Above LF a better technique is to use a broadband transmission line transformer to remove common-mode signals. Such a transformer has little or no voltage gain, so the signal is best displayed on a spectrum analyzer, with µV sensitivity, rather than on an oscilloscope, which only has sensitivity of 5mV or so.

Decoupling

The final issue we must consider before discussing the actual techniques of breadboarding is decoupling. The power supplies of HF circuits must be short-circuited together and to ground at all frequencies above DC. (DC short-circuits are undesirable for reasons which I shall not bother to discuss.) At low frequencies the impedance of supply lines is (or should be) low and so decoupling can be accomplished by relatively few electrolytic capacitors, which will not generally need to be very close to the parts of the circuit they are decoupling, and so may be shared among several parts of a system. (The exception to this is where a component draws a large LF current, when a local, dedicated, electrolytic capacitor should be used.)

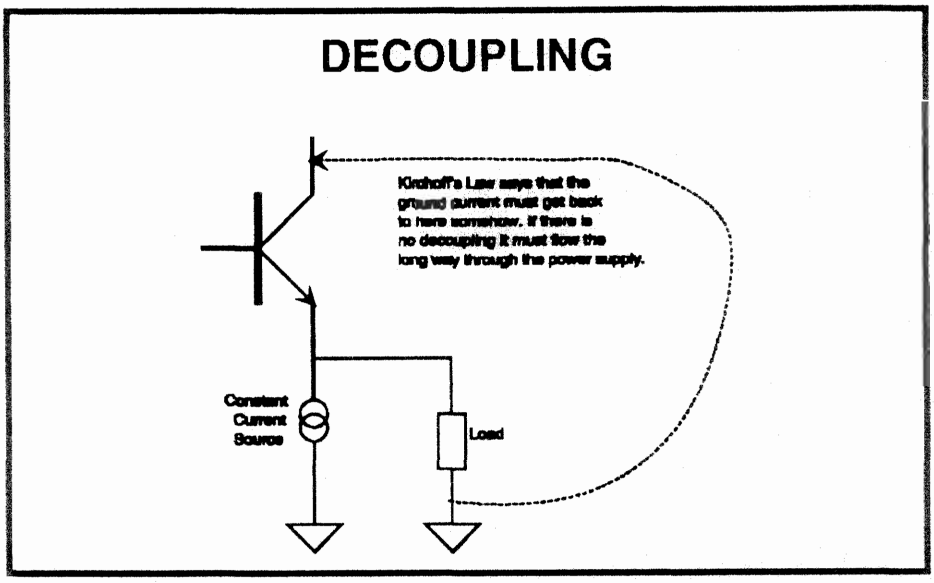
At HF we cannot ignore the impedance of supply leads (as we have already seen in Figure 9–6) and ICs must be individually decoupled with low inductance capacitors having short leads and PC tracks. Even 2–3mm of extra lead/track length may make the difference between the success and failure of a circuit layout.

DECOUPLING

Supplies must be short-circuited to each other and to ground at *all* frequencies.
(But not at DC.)

Figure 9–14.

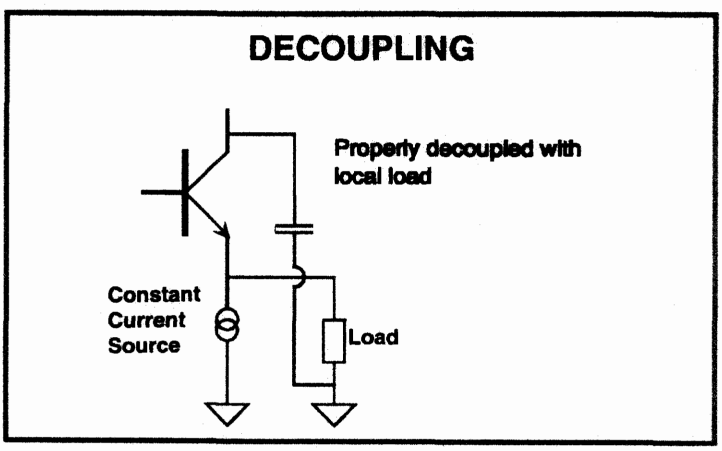
Figure 9–15.



Where the HF currents of a circuit are mostly internal (as is the case with many ADCs) it is sufficient that we short-circuit its supplies at HF so that it sees its supplies as stiff voltage sources at all frequencies. When it is driving a load, the decoupling must be arranged to ensure that the total loop in which the load current flows is as small as possible. Figure 9–15 shows an emitter follower without supply decoupling—the HF current in the load must flow through the power supply to return to the output stage (remember that Kirchoff's Law says, in effect, that currents must flow in circles). Figure 9–16 shows the same circuit with proper supply decoupling.

This principle is easy enough to apply if the load is adjacent to the circuit driving it. Where the load must be remote it is much more difficult, but there are solutions. These include transformer isolation and the use of a transmission line. If the signal contains no DC or LF compo-

Figure 9–16.



nents, it may be isolated with a transformer close to the driver. Such an arrangement is shown in Figure 9–17. (The nature of the connection from the transformer to the load may present its own problems—but supply decoupling is not one of them.)

A correctly terminated transmission line constrains HF signal currents so that, to the supply decoupling capacitors, the load appears to be adjacent to the driver. Even if the line is not precisely terminated, it will constrain the majority of the return current and is frequently sufficient to prevent ground current problems.

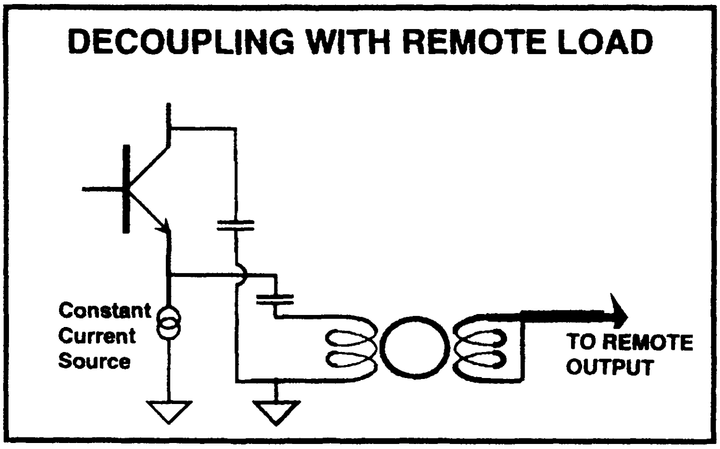


Figure 9–17.

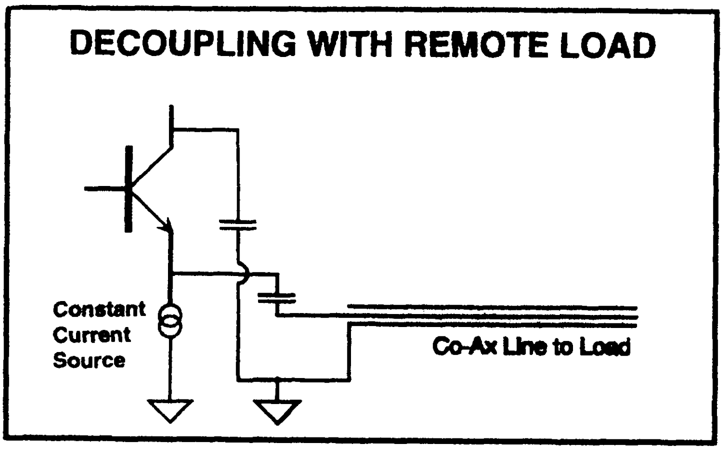


Figure 9–18.

Breadboarding Principles

Having considered issues of resistance, capacitance, and inductance, it is clear that breadboards must be designed to minimize the adverse effects of these phenomena. The basic principle of a breadboard is that it is a temporary structure, designed to test the performance of a circuit or system, and must therefore be easy to modify.

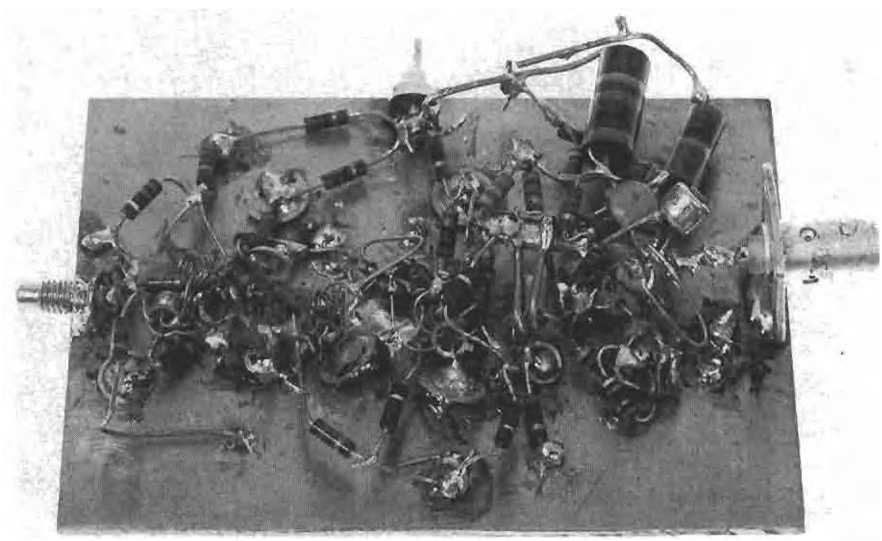
There are many commercial breadboarding systems, but almost all of them are designed to facilitate the breadboarding of digital systems, where noise immunities are hundreds of millivolts or more. (We shall discuss the exception to this generality later.) Matrix board (Veroboard, etc.), wire-wrap, and plug-in breadboard systems (Bimboard, etc.) are, without exception, unsuitable for high performance or high frequency analog breadboarding. They have too high resistance, inductance and capacitance. Even the use of IC sockets is inadvisable. (All analog engineers should practice the art of unsoldering until they can remove an IC from a breadboard [or a plated-through PCB] without any damage to the board or the device—solder wicks and solder suckers are helpful in accomplishing this.)

Practical Breadboarding

The most practical technique for analog breadboarding uses a copper-clad board as a ground plane. The ground pins of the components are soldered directly to the plane, and the other components are wired together above it. This allows HF decoupling paths to be very short indeed. All lead lengths should be as short as possible, and signal routing should separate high-level and low-level signals. Ideally the layout should be similar to the layout to be used on the final PCB.

Pieces of copper-clad may be soldered at right angles to the main ground plane to provide screening, or circuitry may be constructed on both sides of the board (with connections through holes) with the board itself providing screening. In this case the board will need legs to protect the components on the underside from being crushed.

Figure 9–19.



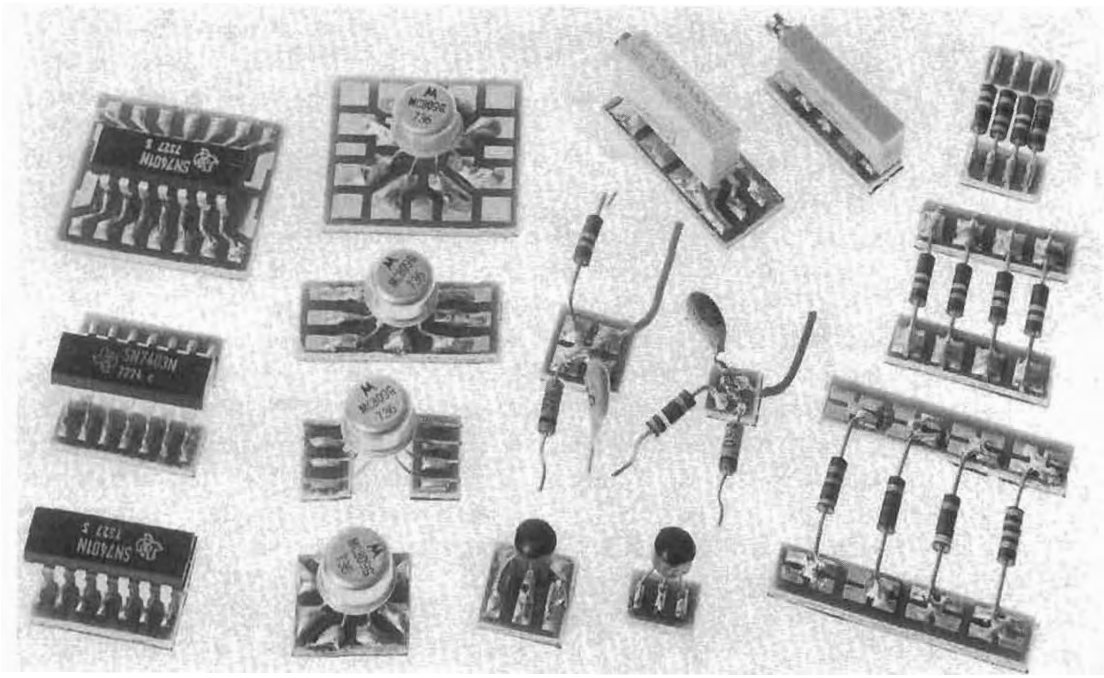


Figure 9–20.

When the components of a breadboard of this type are wired point-to-point in the air (a type of construction strongly advocated by Robert A. Pease of National Semiconductor[1] and sometimes known as "bird's nest" construction) there is always the risk of the circuitry being crushed and resulting short-circuits; also, if the circuitry rises high above the ground plane, the screening effect of the ground plane is diminished and interaction between different parts of the circuit is more likely. Nevertheless the technique is very practical and widely used because the circuit may so easily be modified.

However, there is a commercial breadboarding system which has most of the advantages of "bird's nest over a ground plane" (robust ground, screening, ease of circuit alteration, low capacitance, and low inductance) and several additional advantages: it is rigid, components are close to the ground plane, and where necessary node capacitances and line impedances can be calculated easily. This system was invented by Claire R. Wainwright and is made by WMM GmbH in the town of Andechs in Bavaria and is available throughout Europe and most of the world as "Mini-Mount" but in the USA (where the trademark "Mini-Mount" is the property of another company) as the "Wainwright Solder-Mount System."[2] (There is also a monastery at Andechs where they brew what is arguably the best beer in Germany.)

Solder-Mounts consist of small pieces of PCB with etched patterns on one side and contact adhesive on the other. They are stuck to the ground plane and components are soldered to them. They are available in a wide variety of patterns, including ready-made pads for IC packages of all sizes from 8-pin SOICs to 64-pin DILs, strips with solder pads at intervals (which intervals range from .040" to .25"; the range includes strips with 0.1" pad spacing which may be used to mount DIL devices), strips with conductors of the correct width to form microstrip transmission lines (50Ω, 60Ω, 75Ω or 100Ω) when mounted on the ground plane, and a variety of pads for mounting various other components. A few of the many types of Solder-Mounts are shown in Figure 9–20.

The main advantage of Solder-Mount construction over "bird's nest" is that the resulting circuit is far more rigid, and, if desired, may be made far smaller (the latest Solder-Mounts are for surface-mount devices and allow the construction of breadboards scarcely larger than the final PCB, although it is generally more convenient if the prototype is somewhat larger). Solder-Mounts are sufficiently durable that they may be used for small quantity production as well as prototyping—two pieces of equipment I have built with Solder-Mounts have been in service now for over twenty years.

Figure 9–21 shows several examples of breadboards built with the Solder-Mount System. They are all HF circuits, but the technique is equally suitable for the construction of high resolution LF analog circuitry. A particularly convenient feature of Solder-Mounts at VHF is the ease with which it is possible to make a transmission line.

If a conductor runs over a ground plane it forms a microstrip transmission line. The Solder-Mount System has strips which form microstrip lines when mounted on a ground plane (they are available with impedances of 50Ω, 60Ω, 75Ω and 100Ω). These strips may be used as transmission lines, for impedance matching, or simply as power buses. (Glass fiber/epoxy PCB is somewhat lossy at VHF and UHF, but the losses will probably be tolerable if microstrip runs are short.)

It is important to realize that current flow in a microstrip transmission line is constrained by inductive effects. The signal current flows only on the side of the conductor next to the ground plane (its skin depth is calculated in the normal way) and the return current flows only directly beneath the signal conductor, not in the entire ground plane (skin effect naturally limits this current, too, to one side of the ground plane). This is helpful in separating ground currents, but increases the resistance of the circuit.

It is clear that breaks in the ground plane under a microstrip line will force the return current to flow around the break, increasing impedance. Even worse, if the break is made to allow two HF circuits to cross, the two signals will interact. Such breaks should be avoided if at all possible. The best way to enable two HF conductors on a ground plane to cross without interaction is to keep the ground plane continuous and use a microstrip on the other side of the ground plane to carry one of the signals past the other (drill a hole through the ground plane to go to the other side of the board). If the skin depth is much less than the ground plane thickness the interaction of ground currents will be negligible.

Figure 9–21

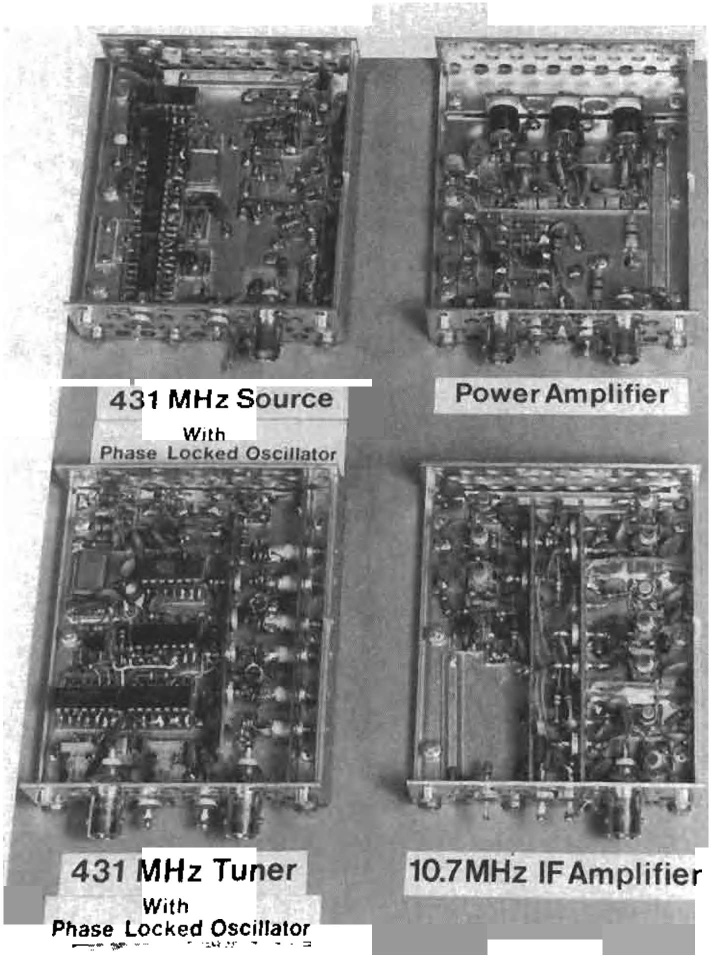
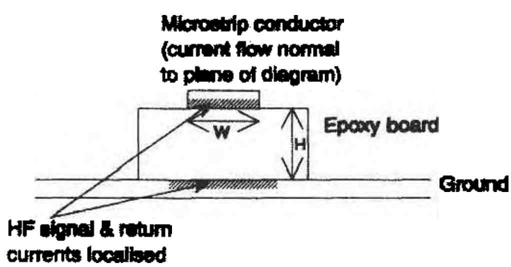


Figure 9–22.

MICROSTRIP TRANSMISSION LINE



When a conductor runs over a ground plane it forms a microstrip transmission line.

The characteristic impedance is $\frac{377H}{W\sqrt{E_r}}$ Ω (note that the units of H and W are unimportant)

The transmission line determines where both the signal *and return* currents flow

Conclusion

It is not possible in a short chapter to discuss all the intricacies of successful analog breadboard construction, but we have seen that the basic principle is to remember all the laws of nature which apply and consider their effects on the design.

Figure 9–23.

SUCCESSFUL ANALOG BREADBOARDS
Pay attention to: Resistance Capacitance Inductance Decoupling Ground & Separating sensitive circuits from noisy ones

In addition to the considerations of resistance, skin effect, capacitance, inductance and ground current, it is important to configure systems so that sensitive circuitry is separated from noise sources and so that the noise coupling mechanisms we have described (common resistance/inductance, stray capacitance, and mutual inductance) have minimal opportunity to degrade system performance. ("Noise" in this context means a signal we want [or which somebody wants] in a place where we don't want it; not natural noise like thermal, shot or popcorn noise.) The general rule is to have a signal path which is roughly linear, so that outputs are physically separated from inputs and logic and high level external signals only appear where they are needed. Thoughtful layout is important, but in many cases screening may be necessary as well.

A final consideration is the power supply. Switching power supplies are ubiquitous because of their low cost, high efficiency and reliability, and small size. But they can be a major source of HF noise, both broadband and at frequencies harmonically related to their switching frequency. This noise can couple into sensitive circuitry by all the means we have discussed, and extreme care is necessary to prevent switching supplies from ruining system performance.

Prototypes and breadboards frequently use linear supplies or even batteries, but if a breadboard is to be representative of its final version it should be powered from the same type of supply. At some time during

Figure 9–24.

SWITCHING POWER SUPPLIES

Generate noise at every frequency under the Sun (and some interstellar ones as well).

Every mode of noise transmission is present.

If you must use them you should filter, screen, keep them far away from sensitive circuits, and still worry!

development, however, it is interesting (and frightening, and helpful) to replace the switching supply with a battery and observe the difference in system performance.

Figure 9–25.

OBEY THE LAW

Unexpected behaviour of analog circuitry is almost always due to the designer overlooking one of the basic laws of electronics. Remember and obey Ohm, Faraday, Lenz, Maxwell, Kirchoff **and MURPHY.**

"Murphy always was an optimist" - Mrs. Murphy.

References

1. Robert A. Pease, *Troubleshooting Analog Circuits* (Butterworth-Heinemann, 1991).
2. Wainwright Instruments Inc., 7770 Regents Rd., #113 Suite 371, San Diego, CA 92122 (619) 558 1057 Fax: (619) 558 1019.

   WMM GmbH, Wainwright Mini-Mount-System, Hartstraße, 28C, D-82346 Andechs-Frieding, Germany, (+49)8152-3162 Fax: (+49)8152-4025.