EDN ACCESS
FOR DESIGN, BY DESIGN

Design Feature: ***February 15, 1996***

Prototyping techniques help verify analog-circuit performance

Walt Kester, Analog Devices

Despite the pressure for system engineers to simulate every design, a simulation of a high-speed, high-performance analog circuit cannot substitute for a quality prototype. A review of prototyping methods helps you choose a technique suitable for your design.

Analog designers use as many tools as possible to ensure that the final system design performs correctly. The first step is the intelligent use of IC macromodels, if available, to simulate the circuit. The second step is the construction of a prototype board to further verify the design. The final pc-board layout should as closely as possible duplicate the prototype layout.

Unfortunately, system designers are under increasing pressure to verify their designs, sometimes exclusively, with computer simulations before committing to board layouts and hardware. Simulating complex digital designs is beneficial, because such simulations often let you eliminate the prototype phase. Bypassing the prototype phase in high-speed, high-performance analog or mixed-signal circuit designs can be risky for many reasons, however.

The models available to system designers are only gross approximations of the analog components they emulate (see box, "The limitations of analog-circuit simulation"). Even if semiconductor manufacturers made more detailed models available, simulation times would be impractically long, and the simulations might fail to converge. Thus, designers of analog circuits must become proficient at prototyping to experimentally verify their analog circuit's performance. (An analog circuit is one that uses ICs, such as op amps, instrumentation amps, programmable-gain amps, voltage-controlled amps, log amps, mixers, and analog multipliers. A mixed-signal circuit is an ADC, DAC, or combinations of these ICs with some digital signal processing, which may exist on the same IC.)

The basic principle of a breadboard or prototype is that it is a *temporary* structure to test the performance of a circuit or system and must, therefore, be easy to modify. Many commercial prototyping systems exist, but almost all facilitate the prototyping of digital systems, in which noise immunities are hundreds of millivolts or more. Non-copper-clad matrix board, non-copper-clad Vectorboard (Vector Electronic Company, Sylmar, CA), wire-wrap, and plug-in breadboard systems are, without exception, unsuitable for high-performance or high-frequency analog prototyping. The resistance, inductance, and capacitance of these breadboards are too high. Even the use of standard IC sockets is inadvisable in many prototyping applications.

An important consideration in selecting a prototyping method is the requirement for a large-area ground plane. A large ground plane is necessary for high-frequency circuits and low-speed precision circuits, especially when those circuits include ADCs or DACs. The differentiation between high-speed and high-precision mixed-signal circuits is difficult to make. For example, 16+-bit ADCs and DACs may operate on high-speed clocks greater than 10 MHz with rise and fall times of less than a few nanoseconds, even though the effective throughput rate of the converters may be less than 100k samples/sec. Successful prototyping of these circuits requires that you pay equal attention to good high-speed and high-precision circuit techniques.

The "dead-bug" technique

The simplest technique for analog prototyping uses a solid copper-clad board as a ground plane (References 1 and 2). You solder the ground pins of the ICs directly to the plane and wire together the other components above the plane. This arrangement allows high-frequency decoupling paths to be short. All lead lengths should be as short as possible, and signal routing should separate high- and low-level signals. You should locate all connection wires close to the surface of the board to minimize the possibility of stray inductive coupling. You should not bundle parallel runs because of possible coupling. Ideally, the layout (at least the relative placement of the components on the board) should be similar to the layout of the final pc board. This approach is often called "dead-bug" prototyping, because the ICs mount upside down with their leads up in the air (with the exception of the ground pins, which are bent and soldered directly to the ground plane). The upside-down ICs look liked dead bugs; hence, the name.

Figure 1 shows a hand-wired breadboard of two high-speed op amps, which gives excellent performance despite its lack of aesthetic appeal. The IC op amps mount upside down on the copper board with the leads bent. Short point-to-point wiring connects the signals. The characteristic impedance of a wire over a ground plane is about 120 Ohms, although this number can vary as much as 140%, depending on the distance from the plane. The decoupling capacitors connect directly from the op amps' power pins to the copper-clad ground plane. When you are working at frequencies of several hundred megahertz, use only one side of the board for ground. Many people drill holes in the board and connect both sides with short pieces of wire soldered to both sides of the board. If you're not careful, however, this connection can result in unexpected ground loops between the two sides of the board, especially at radio frequencies.

You can solder pieces of copper-clad board at right angles to the main ground plane to provide screening, or you can construct circuitry on both sides of the board with connections through holes, and the board itself provides screening. If the board provides screening, the board needs standoffs at the corners to protect the components on the underside from being crushed.

When you construct a breadboard of this type using point-to-point wiring in the air, sometimes called "bird's nest" construction, you risk crushing the circuit and causing short circuits (Reference 2). Also, if the circuitry rises high above the ground plane, the screening effect of the ground plane decreases, and interaction between parts of the circuit is more likely. Nevertheless, the technique is practical and popular, because it makes the circuit easy to modify--assuming that the person doing the modifications is adept at using a soldering iron, a solder wick, and a solder sucker.

Predrilled copper-clad prototypes

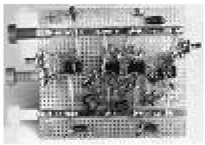


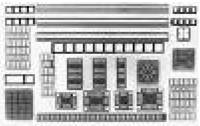
Figure 2 shows another prototype in which a single-sided copper-clad Vectorboard has predrilled holes on 0.1-in. centers. Power buses are at the top and bottom of the board. The power pins of each IC have their own decoupling capacitors. Because of the loss of copper area due to the predrilled holes, this technique does not provide as low a ground impedance as a completely covered, copper-clad board.

A variation of this technique is to mount the ICs and other components on the non-copper-clad side of the board. The holes serve as vias, and you do the point-to-point wiring on the copper-clad side of the board. To prevent shorts, you must drill out the copper surrounding each hole used for a via. This approach requires that all IC pins be on 0.1-in. centers. You can use low-profile sockets for low-frequency circuits, and the socket pins allow for easy point-to-point wiring.

One commercial breadboarding system has most of the advantages of robust ground, screening, ease of circuit alteration, low capacitance, low inductance, and several additional advantages. The system is rigid, has components that are close to the ground plane, and lets you easily calculate node capacitances and line impedances. The product is available as "Mini-Mount" in Europe from Wainwright Instruments GmbH (Andechs-Frieding, Germany) and as "Solder-Mount" in the United States (where the trademark "Mini-Mount" is the property of another company) from Wainwright Instruments Inc (San Diego, CA).

Table 1--Comparison of DIP and SOIC performance

AD8001 package	**Signal gain**	**Optimum R_F (Ohms)**	**Optimum R_G (Ohms)**	**0.1-dB flatness bandwidth (MHz)**
DIP (AD8001AN)	-1	649	649	105
	+1	1050	--	70
	+2	750	750	105
SOIC (AD8001AR)	-1	604	604	130
	+1	953	--	100
	+2	681	681	120



Solder-Mount comprises small pieces of pc board with etched patterns on one side and contact adhesive on the other (Figure 3a). The pc-board pieces stick to the ground plane, and you can solder components to the pieces. The board pieces are available in a variety of patterns, including ready-made pads for eight-pin SOICs to 64-pin dual-in-line types; strips with solder pads at intervals ranging from 0.040 to 0.25 in., including strips with 0.1-in. pad spacing, which you can use to mount dual-in-line devices; strips with 50, 60, 75, or 100_ conductors to form microstrip transmission lines when you mount them on the ground plane; and a variety of pads for mounting various other components.

The conductor-strip feature of Solder-Mount at VHF is convenient. You can use these strips for transmission lines, impedance matching, or power buses. Glass-fiber/epoxy pc board is somewhat lossy at VHF and UHF, but the losses are probably tolerable if microstrip runs are short. Self-adhesive, tinned copper strips and rectangles (LO-PADS) are also available as tie-points for connections. These strips have a relatively high capacitance to ground and, therefore, serve as low-inductance decoupling capacitors. The strips come in sheet form, and you can cut them with a knife or scissors.

The main advantage of Solder-Mount construction over bird's nest or dead-bug construction is that the circuit resulting from using Solder-Mount is stiffer and, if an application requires, smaller. The latest Solder-Mounts for surface-mount devices let you construct breadboards scarcely larger than the final pc board. It is generally more convenient if the prototype is somewhat larger than the final breadboard, however. Solder-Mount is durable enough for low-quantity production. A 2.5-GHz PLL prototype built with Solder-Mount (Figure 3b) is a high-speed circuit, but the technique is equally suitable for the construction of high-resolution, low-frequency analog circuitry.

Both the dead-bug and Solder-Mount techniques become tedious for complex analog or mixed-signal circuits. Formal layout techniques often produce better prototypes of large circuits. One approach to prototyping more complex analog circuits is to lay out a double-sided board using CAD. PC-based layout packages offer easy layout and schematic capture to verify connections. Such packages are available from PADS Software (Marlborough, MA) and Accel Technologies (San Diego, CA). Although most layout software offers some autorouting capability, this feature is best left to digital designs. After you place the components in position, manually route the interconnections following layout guidelines (Reference 3). After you complete the layout, the software verifies the connections against the schematic diagram's netlist.

The limitations of analog-circuit simulation

A typical IC op amp contains 20 to 40 transistors, almost as many resistors, and a few capacitors. A complete Spice model (Reference 4) contains all these components and a few of the important parasitic capacitances and spurious diodes that diffusions in the op-amp chip form. For high-speed ICs, the model may also include the package and wire-bond parasitics.

This model is the type that IC designers use to optimize the device during the design phase, and the designer typically runs the simulation on a CAD workstation. In a simulation, this detailed model, a so-called micromodel, behaves much--but not exactly--like an actual op amp.

In the micromodel, the IC designer uses transistor and other device models based on the component's actual fabrication process. Semiconductor manufacturers invest considerable amounts of time and money developing and refining these device models to provide IC designers with confidence that the first silicon will work and to minimize mask changes, which cost additional time and money.

The IC manufacturer publishes neither these device models nor the IC micromodels, because they contain design information that other companies could copy or improve on. Also, a simulation of a system containing several ICs, each represented by its own micromodel, would take too long to reach a useful result. Spice micromodels of analog ICs often fail to converge, especially under transient conditions, and multiple IC circuits increase the possibility of convergence problems.

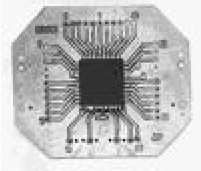
For these reasons, IC manufacturers and software companies publish so-called macromodels--as opposed to micromodels--of analog circuits. Macromodels simulate the major features but lack the detail of a component. Most manufacturers of linear ICs, including Analog Devices, provide these macromodels for components such as op amps, analog multipliers, and references (References 5 and 6). These models represent approximations to the actual circuit; the model rarely includes parasitic effects, such as package capacitance and inductance and pc-board layout.

Macromodels work with various versions of Spice simulation programs, such as PSpice from Microsim Corp (Irvine, CA), and run on workstations or PCs. The models are simple enough that you can simulate multiple ICs in a reasonable amount of computation time and with high certainty of convergence. Consequently, Spice modeling does not always exactly reproduce a circuit's performance. You should experimentally verify the analog circuit using a carefully built prototype.

Finally, some mixed-signal ICs, such as ADCs and DACs, lack Spice models, or the models they do have do not simulate dynamic performance, such as S/N ratio and effective bits. You should also build prototypes of circuits that use these devices.

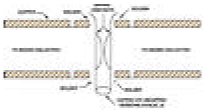
Many design engineers find that they can use CAD techniques to lay out simple boards or work closely with a layout person who has experience in analog-circuit boards. The result is a pattern-generation tape or Gerber file, which you normally would send to a pc-board-manufacturing facility, which makes the final board. Rather than use a pc-board manufacturer, however, you could use automatic drilling and milling machines that directly accept the pattern-generation tape. These prototype-board cutters are available from LPKF CAD/CAM Systems Inc (Beaverton, OR) and T-Tech Inc (Atlanta). These systems directly produce single- and double-sided circuit boards by drilling all holes and then using a milling technique to remove copper, create insulation paths, and create the finished board.

The result is a board similar to the final manufactured double-sided pc board, except that the final board lacks plated-through-hole capability, and you must wire and solder on both sides any vias between the two layers of the board. Minimum trace widths of 25 mils (1 mil=0.001 in.) and 12-mil spacing between traces are standard, although you can achieve smaller trace widths. The size of the milling bit, which is typically 10 to 12 mils, dictates the minimum spacing between lines.



An example of such a prototype board is a daughterboard that interfaces an AD9562 dual PWM board in a 44-pin PLCC package to a test-set motherboard (Figure 4). The leads are on 50-mil centers, and the traces are approximately 25 mils wide. This board illustrates the resolution of the milling machine, but you can use the technique to produce more complex boards.

Use sockets rarely and selectively



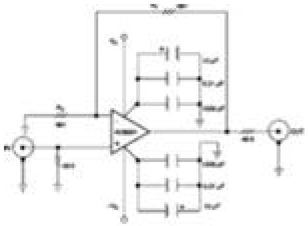
IC sockets can degrade the performance of high-speed or high-precision analog ICs. Although low-profile sockets ease prototyping, even these sockets often introduce enough parasitic capacitance and inductance to degrade circuit performance. If you must use sockets in high-speed circuits, an IC socket comprising individual pin sockets, sometimes called "cage jacks," mounted in the ground plane board may be acceptable (Figure 5). Clear the copper on both sides of the board by about 0.5 mm around each ungrounded pin socket and then solder the grounded pin sockets to ground on both sides of the board. Both capped and uncapped versions of these pin sockets are available from Amp (Harrisburg, PA) (part no. 5-330808-3 and 5-330808-6, respectively). The pin sockets protrude through the board far enough to allow point-to-point wiring interconnections between them.

The spring-loaded, gold-plated contacts within the pin socket make good electrical and mechanical connection to the IC pins. Multiple insertions, however, may degrade the performance of the pin socket. The uncapped versions allow the IC pins to extend from the bottom of the socket. After the prototype is functional and requires no further changes, you can solder the IC pins directly to the bottom of the socket, thereby making a permanent and rugged connection.

Special prototyping considerations

These prototyping techniques apply only to single- or double-sided pc boards. Multilayer pc boards do not easily lend themselves to standard prototyping techniques. If a design requires multilayer-board prototyping, you can use one side of a double-sided board for ground and the other side for power and signals. You can use point-to-point wiring for additional runs, which would normally exist on the additional layers a multilayer board provides. Unfortunately, it's difficult to control the impedance of the point-to-point wiring runs. The high-frequency performance of a circuit prototyped in this manner may differ significantly from that of the final multilayer board.

Other difficulties in prototyping may occur with op amps and other linear devices that have bandwidths greater than a few hundred megahertz. Variations of greater than 1 pF in parasitic capacitance between the prototype and the final board can cause subtle differences in bandwidth and settling time.



If you use DIP packages for the prototype but SOICs for the production packages, you can see differences between the performance of the prototype and the final pc board. For instance, the AD8001 current-feedback op amp (approximately 800-MHz bandwidth for G=1) is available as both an eight-pin DIP and an eight-pin SOIC.Table 1, which contains data collected with the use of an evaluation board, reflects the difference in performance between the two packages after you optimize the feedback (R_G) and feed-forward (R_F) resistors (Figure 6a). The resistor values in Table 1 produce the highest 0.1-dB flatness bandwidths. The SOIC package's bandwidth is higher because of lower package parasitics. All resistors and capacitors on the board are surface-mount types for low parasitics.

Evaluation boards can be extremely useful in evaluating new analog ICs; these boards let you verify the IC's performance with minimum effort and without constructing your own prototype. Evaluation boards can range from relatively simple ones, with just op amps, for example, to rather complex ones for mixed-signal ICs, such as ADCs. ADC evaluation boards often have onboard memory and DSP µPs for analyzing the ADC's performance. IC manufacturers often provide software so that these more complex boards can interface with a PC to perform complex signal analysis, such as histogram and FFT testing.

Most manufacturers of analog ICs provide evaluation boards, usually at a nominal cost. Regardless of the product, the manufacturer takes proper precautions regarding grounding, layout, and decoupling to ensure optimum device performance. The layout of the components on the evaluation board can guide both the prototype and the final pc-board layout. The artwork or CAD file is usually free, should you wish to copy the layout directly or change it to suit the application.

Figure 6a shows the schematic for the SOIC-package, 800-MHz AD8001 op-amp-evaluation board. Figures 6b and 6c, respectively, show the top and bottom sides of the pc board. The circuit connects the amplifier in the noninverting mode. The top (Figure 6b) shows the SOIC package along with input and output SMA connectors. The ground plane is cut around the SOIC to minimize parasitic capacitance. The bottom of the board (Figure 6c) shows the surface-mount resistors and capacitors that comprise the op-amp gain-setting and power-supply decoupling circuits, respectively.

In high-speed, high-precision ICs, pay special attention to power-supply decoupling. For example, fast-slewing signals into relatively low-impedance loads produce high-speed transient currents at an op amp's power-supply pins. These transient currents produce corresponding voltages across any parasitic impedance that exists in the power-supply traces. These voltages can couple to the amplifier output because of the op amp's finite power-supply rejection at high frequencies.

A three-capacitor decoupling scheme for the AD8001 evaluation board ensures a low-impedance path to ground at all transient frequencies. The 1000-pF and 0.01-µF ceramic capacitors shunt the highest frequency transients to ground. These capacitors sit as close to the power-supply pins as possible to minimize any series inductance and resistance. Because these components are surface-mount, minimum stray inductance and resistance exist in the path to the ground plane. The 10-µF tantalum capacitors shunt the lower frequency transient currents to ground.

The input and output signal traces of the AD8001 evaluation board are 50 Ohm microstrip transmission lines. A considerable amount of continuous ground-plane area exists on both sides of the pc board. Plated-through holes connect the top and bottom ground planes at several points to maintain the continuity of this low-impedance ground at high frequencies.

Author's Biography

Walt Kester is a corporate staff application engineer at Analog Devices Inc in Greensboro, NC. During more than 25 years at Analog Devices, he has designed, developed, and given application support for high-speed ADCs, DACs, S/H amplifiers, op amps, and multiplexers. Besides writing many papers and articles, he has prepared and edited six major application books that form the basis of the Analog Devices worldwide technical seminar series. Kester has a BSEE from North Carolina State University, Raleigh, NC, and an MSEE from Duke University, Durham, NC. He enjoys carpentry and *travel.*

References

1. Williams, Jim, "High Speed Amplifier Techniques," Application Note 47, Linear Technology Corp, August 1991.

2. Pease, Robert A,*Troubleshooting Analog Circuits*, Butterworth-Heinemann, Stoneham, MA, 1991.

3. Johnson, Howard W, and Martin Graham,*High-Speed Digital Design*, PTR Prentice-Hall Inc, Englewood Cliffs, NJ, 1993.

4. Antognetti, Paolo, and Guiseppe Massobrio,*Semiconductor Device Modeling with Spice*, McGraw-Hill, New York, NY, 1988.

5. "Amplifier Applications Guide," Section 13, Analog Devices Inc, Norwood, MA, 1992.

6. Boyle, GR, et al, "Macromodeling of Integrated Circuit Operational Amplifiers,"*IEEE Journal of Solid-State Circuits*, SC-9, December 1974.

7. "Practical Analog Design Techniques," Analog Devices Inc, Norwood, MA, 1995.

| EDN Access | feedback | **subscribe to EDN!** |
| design features | out in front | design ideas | departments | products |
