Breadboarding and Prototyping Techniques

LIMITATIONS OF ANALOG CIRCUIT SIMULATION

As discussed in Tutorial MT-099, there has been much pressure placed on system designers to verify their designs with computer simulations before committing to actual printed circuit board layouts and hardware. Simulating complex digital designs is extremely beneficial, and very often, the prototype phase can be eliminated entirely. However, bypassing the prototype phase in high-speed/high-performance analog or mixed-signal circuit designs can be risky for a number of reasons.

The macromodels discussed in MT-099 are only *approximations* to the actual circuit, and parasitic effects such as package capacitance and inductance and PC board layout are rarely included. The models are simple enough so that circuits using multiple ICs can be simulated in a reasonable amount of computation time and with good certainty of convergence. Consequently, SPICE modeling does not always reproduce the exact performance of a circuit and should always be verified experimentally using a carefully built prototype.

Finally, there are certain mixed-signal ICs such as A/D and D/A converters which have *no* SPICE models, or if they exist, the models do not simulate dynamic performance (signal-to-noise ratio, effective bits, SFDR, etc.). However, recent advances in software (ADIsimADC™ or VisualAnalog™) provide accurate behavioral models which can be used to predict ADC dynamic performance under user conditions without the need for hardware.

PROTOTYPING TECHNIQUES

A basic principle of a breadboard or a prototype structure is that it is a *temporary* one, designed to test the performance of an electronic circuit or system. By definition it must therefore be easy to modify, particularly so for a breadboard.

There are many commercial prototyping systems, but unfortunately for the analog designer, almost all of them are designed for prototyping *digital* systems. In such environments, noise immunities are hundreds of millivolts or more. Prototyping methods commonly used include non copper-clad Matrix board, non-clad Vectorboard®, wire-wrap, and plug-in breadboard systems. Quite simply, these all are unsuitable for high performance or high frequency analog prototyping, because of their excessively high parasitic resistance, inductance, and capacitance levels. Even the use of standard IC sockets is inadvisable in many prototyping applications (more on this, below).

Figure 1 summarizes a number of key points on selecting a useful analog breadboard and/or prototyping system, which are further discussed below.

- **Always Use a Large Area Ground Plane for Precision or High Frequency Circuits**
- **Minimize Parasitic Resistance, Capacitance, and Inductance**
- **If Sockets Are Required, Use "Pin Sockets" ("Cage Jacks")**
- **Pay Equal Attention to Signal Routing, Component Placement, Grounding, and Decoupling in Both the Prototype and the Final Design**
- **Popular Prototyping Techniques:**
    - **Freehand "Deadbug" Using Point-to-point Wiring**
    - **Milled PC Board From CAD Layout**
    - **Multilayer Boards: Double-sided With Additional Point-to-point Wiring**
- **Modern Surface-Mount ICs in Small Packages Require Special Techniques—Usually a Preliminary Multilayer PC Board Layout**

Figure 1: A Summary of Analog Prototyping System Key Points

One of the more important considerations in selecting a prototyping method is the requirement for a large-area ground plane. This is required for high frequency circuits as well as low speed precision circuits, especially when prototyping circuits involving 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 (>10 MHz) with rise and fall times of less than a few nanoseconds, while the effective throughput rate of the converters may be less than 100 kSPS. Successful prototyping of these circuits requires that equal (and thorough) attention be given to good high-speed and high-precision circuit techniques.

Years ago, many ICs were offered in both DIP and surface-mount packages, so breadboarding and prototyping could be done using the user-friendly DIP package. Today, however, most high-performance data converters are not available in DIP packages—and if they were, the added package parasitics would limit performance in many cases.

Breadboarding and prototyping in today's environment is especially difficult, because modern surface-mounted ICs in small packages can be extremely difficult to solder into any type of PC board using manual techniques. Ball grid array (BGA) packages are nearly impossible to solder manually. Sockets—very expensive if available—are generally out of the question because of added parasitics, so in many cases, an actual multilayer PC board must be designed and fabricated. This trend has placed an even greater responsibility on the IC manufacturer to supply a variety of high quality well documented evaluation boards to assist in the initial design phases of a project.

"DEADBUG" PROTOTYPING

A simple technique for analog prototyping where DIP ICs are available uses a solid copper-clad board as a ground plane (see References 1 and 2). In this method, the ground pins of the ICs 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. Connection wires should be located close to the surface of the board to minimize the possibility of stray inductive coupling. In most cases, 18-gauge or larger insulated wire should be used. Parallel runs should not be "bundled" because of possible coupling. Ideally the layout (at least the relative placement of the components on the board) should be similar to the layout to be used on the final PCB. This approach is often referred to as *deadbug* prototyping, because the ICs are often mounted upside down with their leads up in the air (with the exception of the ground pins, which are bent over and soldered directly to the ground plane). The upside-down ICs look like deceased insects, hence the name.

Figure 2 shows a hand-wired "deadbug" analog breadboard. This circuit uses two high speed op amps, and in fact gives excellent performance in spite of its lack of esthetic appeal. The IC op amps are mounted upside down on the copper board with the leads bent over. The signals are connected with short point-to-point wiring. The characteristic impedance of a wire over a ground plane is about 120 Ω, although this may vary as much as ±40% depending on the distance from the plane. The decoupling capacitors are connected directly from the op amp power pins to the copper-clad ground plane. When working at frequencies of several hundred MHz, it is a good idea to use only one side of the board for ground. Many people drill holes in the board and connect the sides together by soldering short pieces of wire. If care isn't taken, this may result in unexpected ground loops between the two sides of the board, especially at RF frequencies.

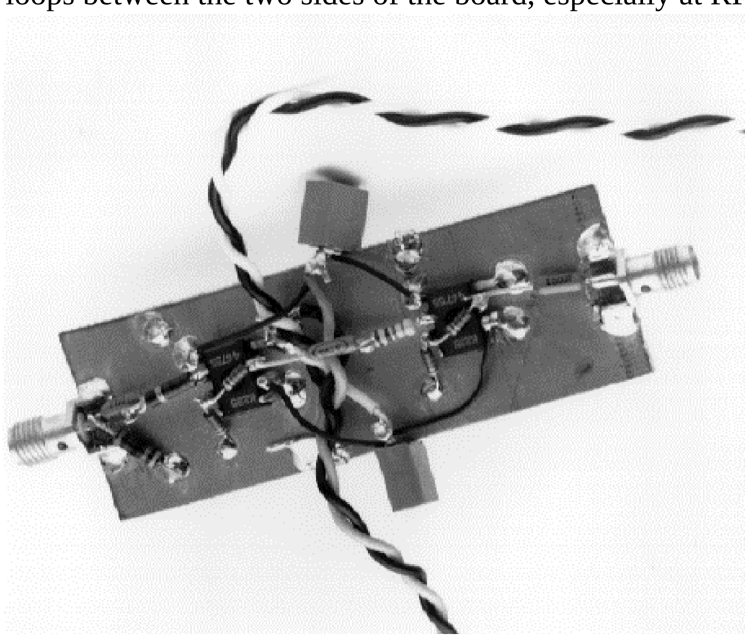


Figure 2: A "Deadbug" Analog Breadboard

Pieces of copper-clad board 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 through-hole connections) with the board itself providing screening. For this, the board will need corner standoffs to protect underside components from being crushed.

When the components of a breadboard of this type are wired point-to-point in the air (a type of construction strongly advocated by Bob Pease (see Reference 3) 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 easily be modified (this of course assumes the person doing the modifications is adept with soldering techniques).

Another prototype breadboard variation is shown in Figure 3. Here the single-sided copper-clad board has pre-drilled holes on 0.1" centers (see Reference 4). Power busses are used at the top and bottom of the board. The decoupling capacitors are used on the power pins of each IC. Because of the loss of copper area due to the pre-drilled holes, this technique does not provide as low a ground impedance as a completely covered copper-clad board of Figure 2, so be forewarned.

In a variation of this technique, the ICs and other components are mounted on the non-copper-clad side of the board. The holes are used as vias, and the point-to-point wiring is done on the copper-clad side of the board. Note that the copper surrounding each hole used for a via must be drilled out, to prevent shorting. This approach requires that all IC pins be on 0.1" centers. For low frequency circuits, low profile sockets can be used, and the socket pins then will allow easy point-to-point wiring.

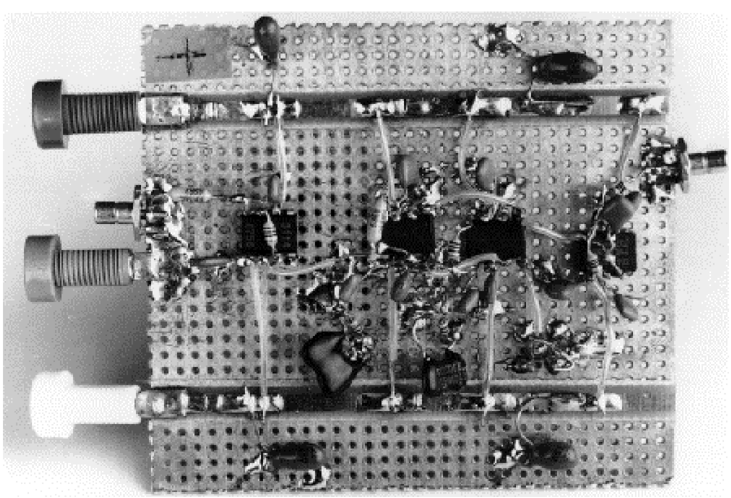


Figure 3: "A Deadbug" Prototype Using 0.1" Pre-Drilled Single-Sided, Copper-Clad Printed Board Material

MILLED PCB PROTOTYPING

The "deadbug" prototypes become tedious for complex analog circuits, and larger circuits are better prototyped using more formal layout techniques.

There is a prototyping approach that is but one step removed from conventional PCB construction, described as follows. This is to actually lay out a double-sided board, using conventional CAD techniques. PC-based software layout packages offer ease of layout as well as schematic capture to verify connections (see Reference 5). Although most layout software has some degree of auto-routing capability, this feature is best left to digital designs. The analog traces and component placements should be done by hand, following the rules discussed elsewhere in this chapter. After the board layout is complete, the software verifies the connections per the schematic diagram net list.

Many designers find that they can make use of CAD techniques to lay out simple boards. The result is a pattern-generation tape (or Gerber file) which would normally be sent to a PCB manufacturing facility where the final board is made.

Rather than use a PCB manufacturer, however, automatic drilling and milling machines are available which accept the PG tape directly (see References 6 and 7). An example of such a prototype circuit board is shown in Figure 4 (top view).

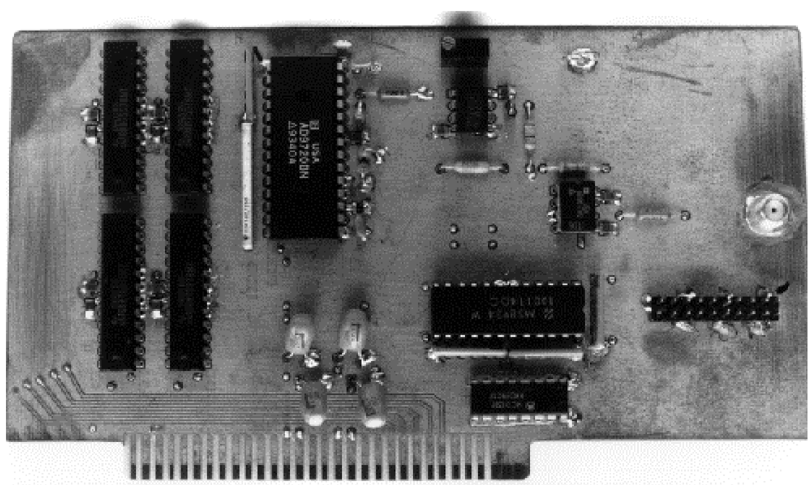


Figure 4: A Milled Circuit Construction Prototype Board (Top View)

These systems produce either single or double-sided circuit boards directly, by drilling all holes and using a milling technique to remove conductive copper, thus creating the required insulation paths, and finally, the finished prototype circuit board. The result can be a board functionally quite similar to a final manufactured double-sided PCB.

However, it should be noted that a chief caveat of this method is that there is no "plated-through" hole capability. Because of this, any conductive "vias" required between the two layers of the board must be manually wired and soldered on both sides.

Minimum trace widths of 25 mils (1 mil = 0.001") and 12 mil spacing between traces are standard, although smaller trace widths can be achieved with care. The minimum spacing between lines is dictated by the size of the milling bit used, typically 10 to 12 mils.

A bottom-side view of this same milled prototype circuit board is shown in Figure 5. The accessible nature of the copper pattern allows access to the traces for modifications.

Perhaps the greatest single advantage of the milled circuit type of prototype circuit board is that it approaches the format of the final PCB design most closely. By its very nature however, it is basically limited to only single or double-sided boards—rendering it virtually useless for surface-mount designs.

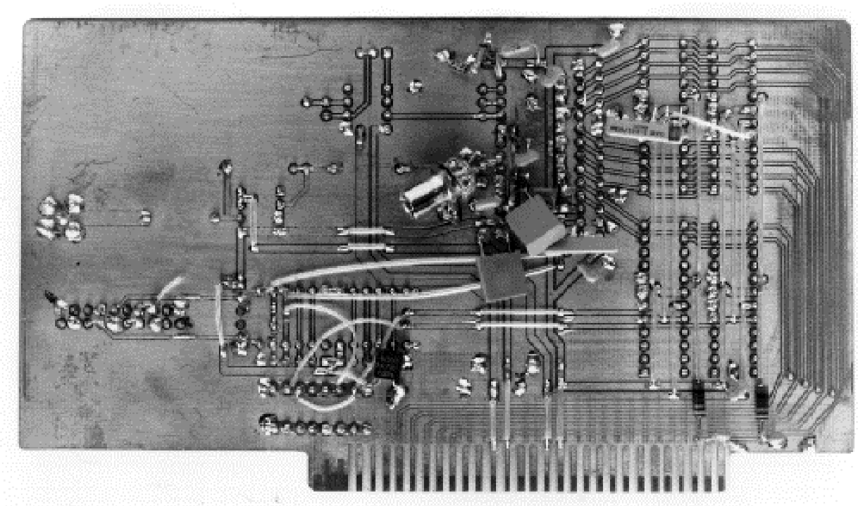


Figure 5: A Milled Circuit Construction Prototype Board (Bottom View)

BEWARE OF SOCKETS!

IC sockets can degrade the performance of high speed or high precision analog ICs. Although they make prototyping easier, even *low-profile* sockets often introduce enough parasitic capacitance and inductance to degrade the performance of a high speed circuit. If sockets must be used, a socket made of individual *pin sockets* (sometimes called *cage jacks*) mounted in the ground plane board may be acceptable, as in Figure 6.

To use this technique, clear the copper (on both sides of the board) for about 0.5-mm around each ungrounded pin socket, Then solder the grounded socket pins to ground, on both sides of the board.

Both capped and uncapped versions of these pin sockets are available (AMP part numbers 5-330808-3, and 5-330808-6, respectively). The pin sockets protrude through the board far enough to allow point-to-point wiring interconnections.

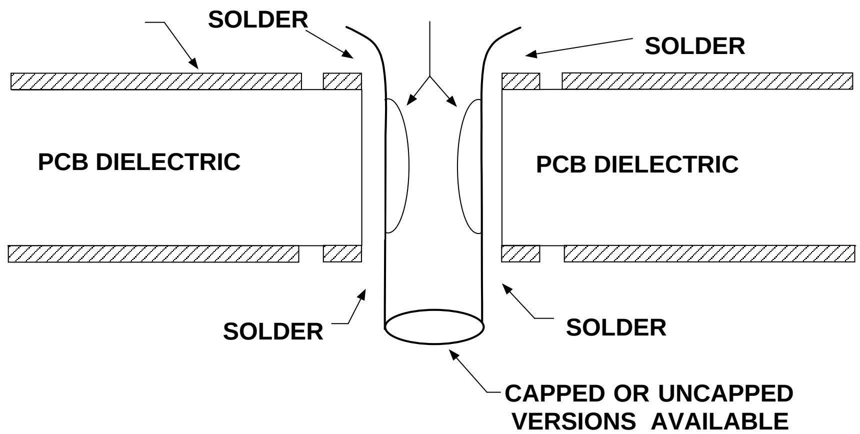


Figure 6: When Necessary, Use Pin Sockets for Minimal Parasitic Effects

Because of the spring-loaded gold-plated contacts within the pin socket, there is good electrical and mechanical connection to the IC pins. Multiple insertions, however, may degrade the performance of the pin socket, so this factor should be kept in mind.

Note also that the uncapped versions allow the IC pins to extend out the bottom of the socket. This feature leads to an additional useful function. Once a prototype using the pin sockets is working and no further changes are to be made the IC pins can be soldered directly to the bottom of the socket. This establishes a rugged, permanent connection.

PROTOTYPING SMALL SURFACE-MOUNT ICs USING ADAPTER BOARDS

When small surface-mount packages made their debut, there were usually DIP versions of the devices available which could be used for prototyping. Today, however, this is not usually the case, and prototyping these small ICs is a real challenge. Typical surface mount IC packages are shown in Figure 7.

The solution (short of actually laying out a PC board) is to mount the small surface-mount IC on an adapter board which then is soldered onto the copper-clad prototype board. Connections are then more easily made to the larger pads and traces on the adapter board. Adapter boards are available from a number of manufacturers (see References 9 and 10, for example), and a selection of them is shown in Figures 8 and 9.

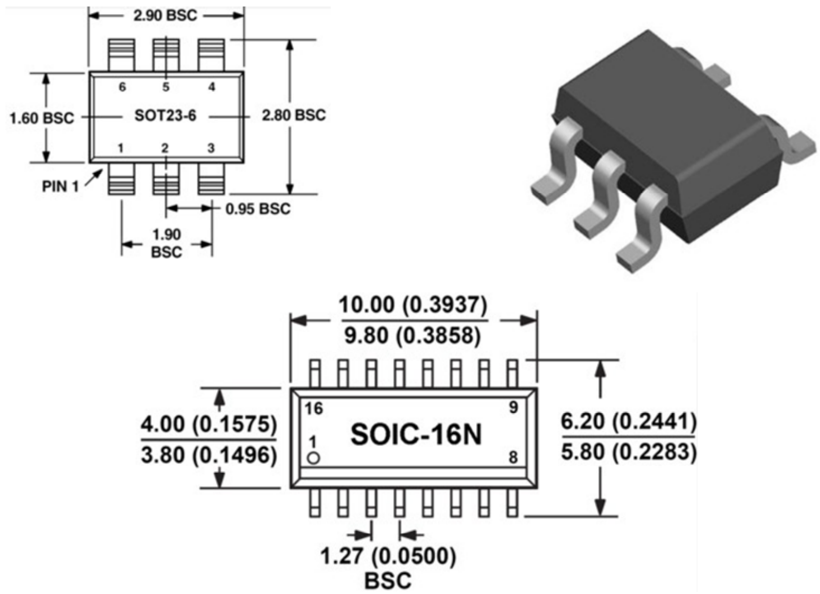
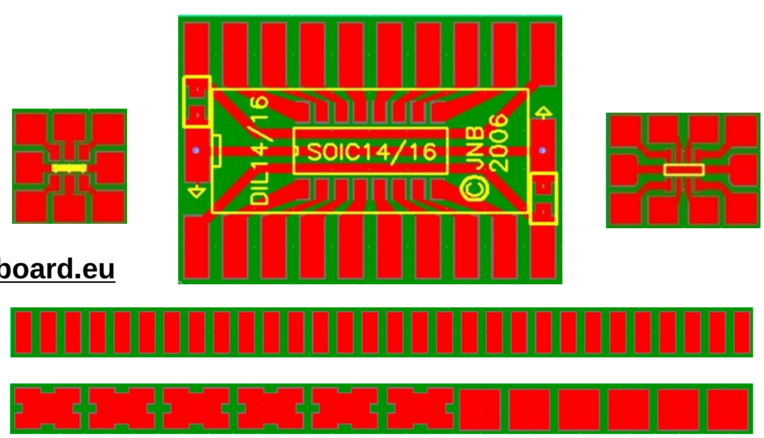


Figure 7: The Problem: Building Breadboards to Work with Tiny Surface-Mount ICs

- **Small PC adapter boards which hold SMDs and have pads (at 0.1" [2.54 mm] spacing) which are large enough to mount larger wired components.**
- **The boards, some of which can also carry DIL ICs, (usually) have provision for supply decoupling and can be mounted on a PCB ground plane by soldering their back plane to it (use a HOT iron!).**
- **There are also strips of pads for mounting other components.**
- **You can then breadboard just as you always do.**



www.sm-breadboard.eu

Figure 8: The Solution: Small Adapter Boards

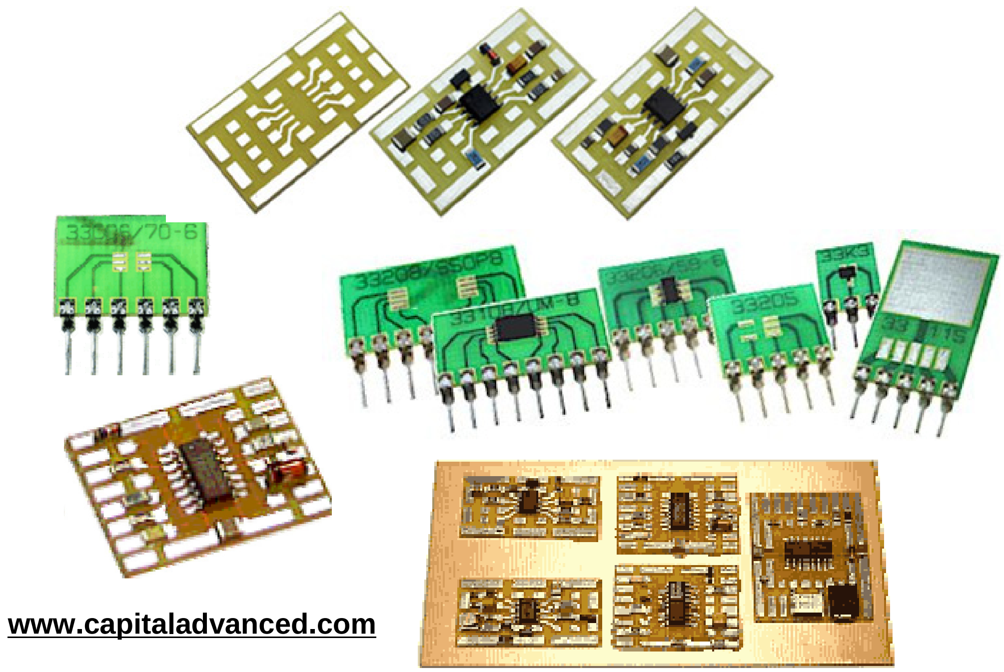


Figure 9: Surfboards® from Capital Advanced Technologies, Inc.

Proper soldering techniques are critical to the success of breadboards using small surface-mount ICs. The most important thing is to use a thermostatically controlled soldering iron with the proper size solder-well tip as shown in Figure 10.

- **You need a good thermostatic soldering iron with a solder-well tip.**

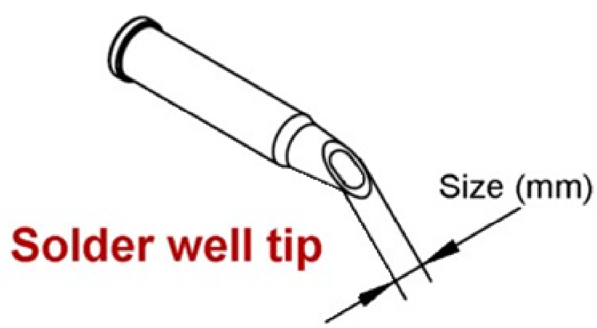


Figure 10: Another Problem:
How do I mount the ICs on the little PCBs?

A detailed procedure for soldering the small ICs is shown in Figure 11, courtesy of www.sm-breadboard.eu™.

- Set the tip temperature to the temperature appropriate to the solder alloy being used. (Leaded solder is cleaner and the EU lead-free laws do not apply to laboratory work.)
- Place component and fix two opposing corner pins.
- Apply flux liberally to all the pins of the IC.
- Clean the solder-well tip on a sponge.
- Fill the concave portion of the tip with solder, to slightly above the rim. Do not overfill!
- Holding your soldering iron VERY LIGHTLY in your hand, set the filled tip, with the solder-well side parallel to the PCB, down onto the flat exterior portion of the pins. The iron and tip should be parallel to the body of the SMD. Slowly pull it across the pins towards you.
- Repeat steps four and five for the remaining sides of the SMD.
- Remove flux residue if necessary.
- Note: A suitable iron, available with solder-well tips, is the ERSA i-CON
  - ERSA GmbH • Leonhard-Karl-Straße 24 • 97877 Wertheim • Germany • www.ersa.com

Figure 11: Soldering Procedure
(Courtesy www.sm-breadboard.eu™)

SOME ADDITIONAL PROTOTYPING POINTS

The prototyping techniques discussed so far have been limited to single or double-sided PCBs. Multilayer PCBs do not easily lend themselves to standard prototyping techniques. If multilayer board prototyping is required, one side of a double-sided board can be used for ground and the other side for power and signals. Point-to-point wiring can be used for additional runs which would normally be placed on the additional layers provided by a multi-layer board. However, it is difficult to control the impedance of the point-to-point wiring runs, and the high frequency performance of a circuit prototyped in this manner may differ significantly from the final multilayer board.

Other difficulties in prototyping may occur with op amps or other linear devices having bandwidths greater than a few hundred megahertz. Small variations in parasitic capacitance (< 1 pF) between the prototype and the final board can cause subtle differences in bandwidth and settling time.

Sometimes, prototyping is done with DIP packages (if available), when the final production package is an SOIC. *This is not recommended!* At high frequencies, small package-related parasitic differences can account for different performance, between prototype and final PCB. To minimize this effect, always prototype with the final packages.

Parasitic differences can also affect high frequency performance of prototypes made using the small "adapter boards" previously discussed.

EVALUATION BOARDS

Most manufacturers of analog ICs provide *evaluation boards* usually at a nominal cost. These boards allow customers to evaluate ICs without constructing their own prototypes. Regardless of the product, the manufacturer has taken proper precautions regarding grounding, layout, and decoupling to ensure optimum device performance. Where applicable, the evaluation PCB artwork is usually made available free of charge, should a customer wish to copy the layout directly or make modifications to suit an application.

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

The AD8001 high speed current-feedback amplifier is a case in point, and a dedicated evaluation board is available for it. A bottom side view of this SOIC board is shown in Figure 12. A triple decoupling scheme was chosen, to ensure a low impedance ground path at all transient frequencies. Highest frequency transients are shunted to ground by dual 1000-pF/0.01-µF ceramic chip capacitors, located as close to the power supply pins as possible to minimize series inductance and resistance. With these surface mount components, there is minimum stray inductance and resistance in the ground plane path. Lower frequency transient currents are shunted by the larger 10-µF tantalum capacitors.

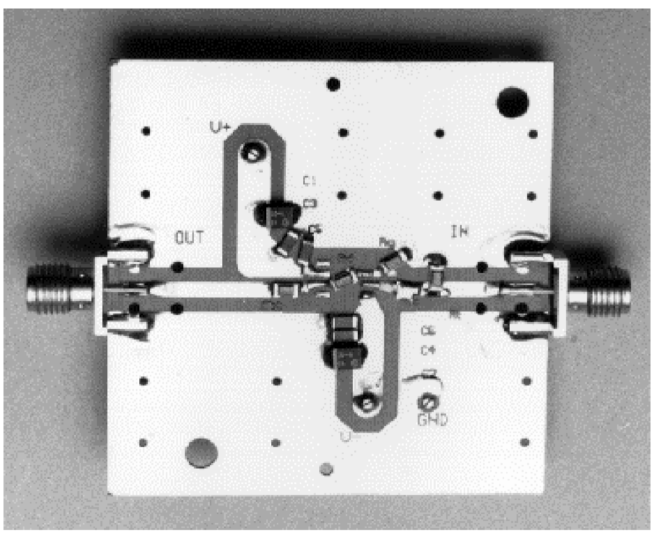


Figure 12: A High Speed Op Amp Such as the AD8001 Requires a Dedicated Evaluation Board With Suitable Ground Planes and Decoupling (Bottom View)

The input and output signal traces of this board are 50-Ω microstrip transmission lines, as can be noted towards the right and left. Gain-set resistors are chip-style film resistors, which have low parasitic inductance. These can be seen in the center of the photo, mounted at a slight diagonal.

Note also that there is considerable continuous ground plane area on both sides of the PCB. Plated-through holes connect the top and bottom side ground planes at several points, in order to maintain lowest possible impedance and best high frequency ground continuity.

Input and output connections to the card are provided via the SMA connectors as shown, which terminate the input/output signal transmission lines. The board’s power connection from external lab supplies is made via solder terminals, which are seen at the ends of the broad supply line traces.

Some of these points are more easily seen in a topside view of the same card, which is shown in Figure 13. This AD8001 evaluation board is a non-inverting signal gain stage, optimized for lowest parasitic capacitance. The cutaway area around the SOIC outline of the AD8001 provides lowest stray capacitance, as can be noted in this view.

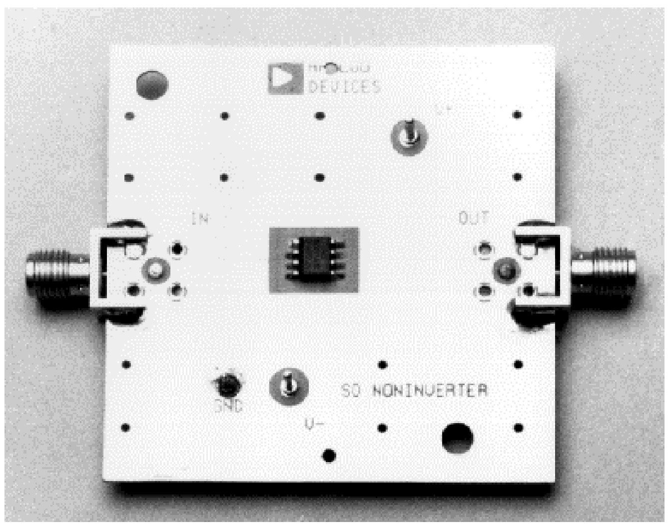


Figure 13: The AD8001 Evaluation Board Uses a Large Area Ground Plane as well as Minimal Parasitic Capacitance (Top View)

In this view is also seen the virtually continuous ground plane and the multiple vias, connecting the top/bottom planes.

DATA CONVERTER EVALUATION BOARDS

A well designed manufacturers' evaluation board is a powerful tool that can greatly simplify the integration of an ADC or DAC into a system. Probably the best feature of an evaluation board is that its layout is designed to optimize the performance of the data converter. Analog Devices provides a complete electrical schematic and parts list as well as a PC board layout of its evaluation boards on the data sheet for most ADCs and DACs. Each layer of the multilayer board is also shown, and Analog Devices will supply the CAD layout files (Gerber format) for the board if needed. Many system level problems related to layout can be avoided simply by studying the evaluation board layout and using it as a guide in the system board layout—perhaps even copying critical parts of the layout directly if needed.

Evaluation boards typically have input/output connectors for the analog, digital, and power interfaces to facilitate interfacing with external test equipment. Any required support circuitry such as voltage references, crystal oscillators for clock generation, etc., are generally included as part of the board.

Many modern data converters have a considerable amount of on-chip digital logic for controlling various modes of operation, including gain, offset, calibration, data transfer, etc. These options are set by loading the appropriate words into internal control registers, usually via a serial port. In some converters, especially sigma-delta ADCs, just setting the basic options requires considerable knowledge of the internal control registers and the interface. For this reason, most ADC/DAC evaluation boards have interfaces (either parallel, serial, or USB) and software to allow easy menu-driven control of the various internal options from an external PC. In many cases, configuration files created in the evaluation software can be downloaded into the final system design.

Figure 14 shows an example of one of an Analog Devices' High-Speed Analog-to-Digital Converter Data Capture Evaluation Kit which interfaces to a variety of high-speed ADC evaluation boards, such as the AD9430 12-bit, 210-MSPS ADC shown here. The evaluation kit includes a memory board (FIFO or FPAG-based) to capture blocks of data from the ADC as well as ADC evaluation software.

The data capture board can be connected to the USB port of a PC and used with the analysis software to quickly evaluate the performance of the high speed ADC. The FIFO-based board contains two 32K, 16-bit wide FIFOs, and data can be captured at clock rates up to 133 MSPS on each channel. It operates with ADIsimADC™ software. The FPGA-based data capture board has a 64K memory and operates up to 644 MSPS SDR or 800 MSPS DDR on each channel. It operates with VisualAnalog™ software.

These boards can be used single or dual ADCs, or ADCs with demultiplexed digital outputs. Users can view the FFT output and analyze SNR, SINAD, SFDR, THD, and harmonic distortion information.

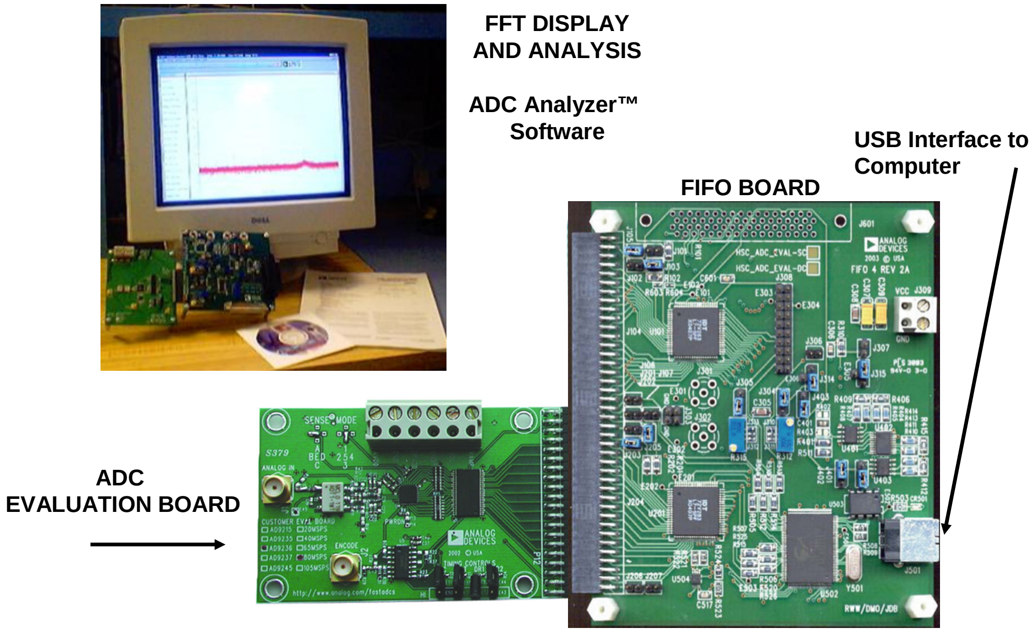


Figure 14: Analog Devices' High Speed ADC Data Capture Evaluation Kit

SUMMARY

The prototyping techniques described earlier in this section are quite useful for ICs which are in DIP packages, and it is well worth the effort to prototype at least some of the critical analog circuitry before going to a final board layout. However, modern high performance ADCs and DACs are often provided in small surface mount packages which do not lend themselves to simple prototyping techniques. In the system, multilayer PC boards are required which further complicates the prototyping process.

In many cases, the only effective prototype for high performance analog systems is an actual PC board layout, especially if a multilayer is required in the final design. Special soldering skills may be required to assemble PCBs with small surface mount devices. Various websites, including References 8 and 9, can be helpful.

Manufacturer's evaluation boards are not only useful in the initial evaluation phases, but their layouts can be used as guides for the actual system board layout.

REFERENCES:

1. Jim Williams, "High Speed Amplifier Techniques," Linear Technology AN-47, August, 1991.

2. Jim Williams, *The Art and Science of Analog Circuit Design (EDN Series for Design Engineers)*, Butterworth-Heinemann, 1995, ISBN-10: 0750695056, ISBN-13: 978-0750695053.

3. Robert A. Pease, *Troubleshooting Analog Circuits*, Butterworth-Heinemann, 1991, ISBN 0-7506-9184-0.

4. Vector Electronics and Technology, Inc., 11115 Vanowen St., North Hollywood, CA 91605, USA. Phone: (818) 985-8208. http://www.vectorelect.com/

5. PADS Software, Mentor Graphics, 8005 S.W. Boeckman Rd., Wilsonville, OR 97070, Tel: (503) 685-7000, http://www.mentor.com/index

6. LPKF Laser & Electronics, 28220 SW Boberg Rd., Wilsonville, OR 97020, 800-345-LPKF or (503) 454-4200, http://www.lpkfcadcam.com

7. T-Tech, Inc., 5591-B New Peachtree Road, Atlanta, GA, 34341, 800 370-1530 or (770) 455-0676, http://www.T-Tech.com

8. SparkFun Electronics, 6175 Longbow Drive, Suite 200, Boulder, CO, 80301. Tutorials on soldering surface mount ICs for prototypes http://www.sparkfun.com/commerce/tutorials.php

9. www.sm-breadboard.eu

10. Capital Advanced Technologies, Inc., 309 Village Drive Carol Stream, Illinois. 60188, 630-690-1696

11. Paul Rako, "Prototyping Techniques: Things to Know Before Pulling the Trigger", *EDN*, 12-05-2008.

12. Hank Zumbahlen, *Basic Linear Design*, Analog Devices, 2006, ISBN: 0-915550-28-1. Also available as *Linear Circuit Design Handbook*, Elsevier-Newnes, 2008, ISBN-10: 0750687037, ISBN-13: 978-0750687034. Chapter 12

13. Walter G. Jung, *Op Amp Applications*, Analog Devices, 2002, ISBN 0-916550-26-5, Chapter 7. Also available as *Op Amp Applications Handbook*, Elsevier/Newnes, 2005, ISBN 0-7506-7844-5. Chapter 7.

14. Walt Kester, *High Speed System Applications*, Analog Devices, 2006, ISBN-10: 1-56619-909-3, ISBN-13: 978-1-56619-909-4, Part 4.