



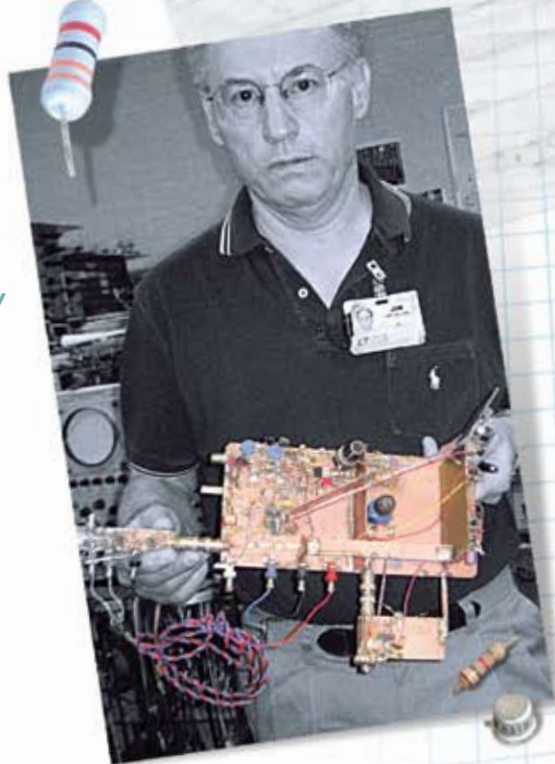
PROTOTYPING TECHNIQUES: THINGS TO KNOW BEFORE PULLING THE TRIGGER

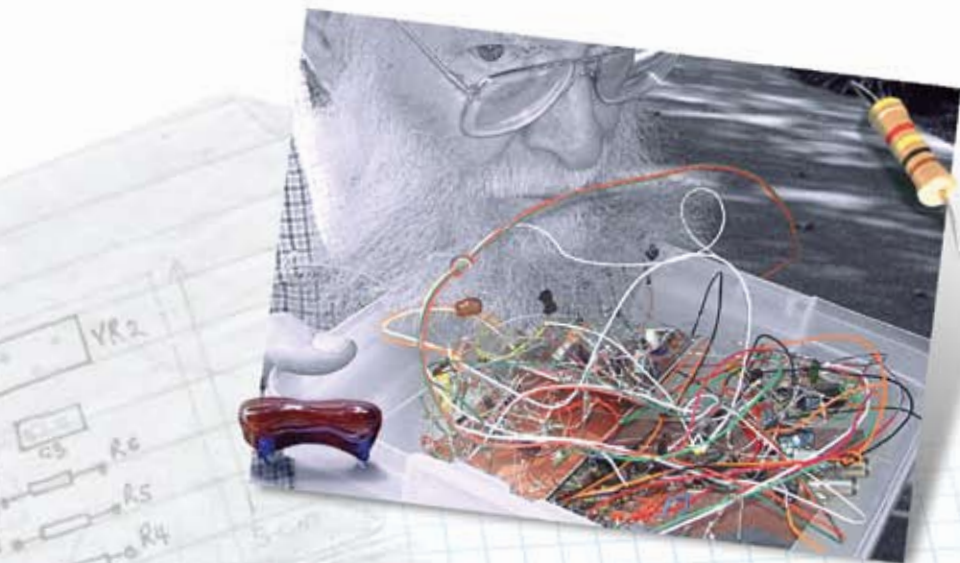
BY PAUL RAKO • TECHNICAL EDITOR

Prototyping has become far more difficult than it was in the old days. For one thing, electronic components have gotten smaller; an IC can now be the size of a peppercorn or a grain of sand. As a result, you must take measures to illuminate, observe, and handle these tiny parts (see sidebar “Prototyping: Think small and work smart” with the Web version of this article at www.edn.com/081205cs). Making things even more difficult is the fact that many modern circuits operate at high frequencies, so you can no longer just solder wires between components; you must connect those circuits with controlled impedances using traces. Thermal management brings more challenges.

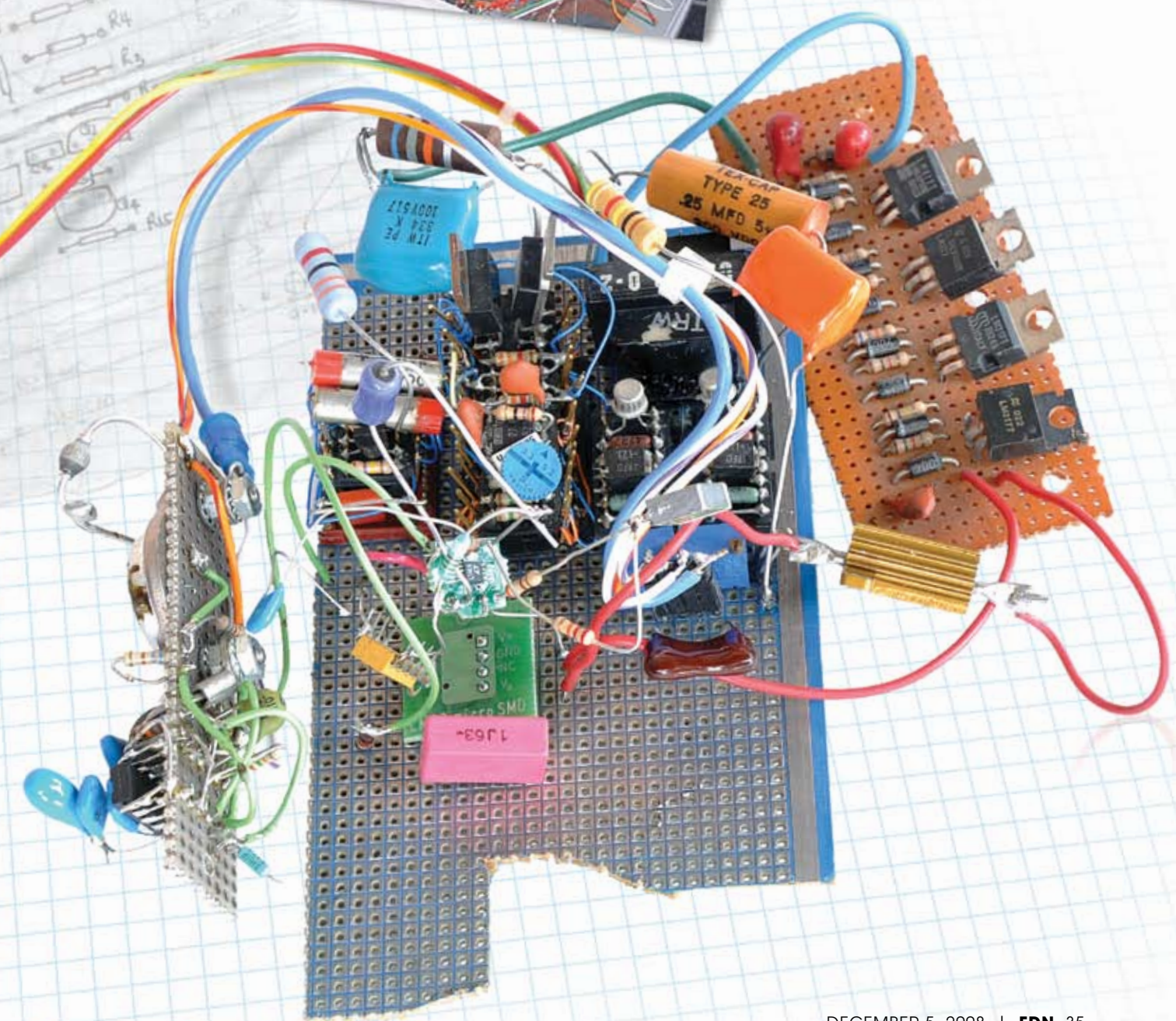
To prototype a PCB (printed-circuit board), you need lamps, tweezers, magnifiers, microscopes, and solder stations. Once you have collected this equipment, you are ready to build your prototype. Remember to take special precautions in the design and layout of analog boards, however (Reference 1).

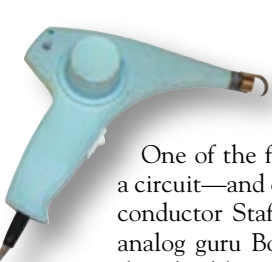
**TINY COMPONENTS
AND HIGH-FREQUENCY
CIRCUITS DEMAND
RIGOROUS PROTO-
TYPING METHODS.**





Even if you write your ideas on a cocktail napkin, you have a number of prototyping tools, including heat guns, at your disposal. Bob Pease (left) whips up an “air ball” of components to quickly prove out his designs, and Jim Williams (lower left) carves up copper-clad material to prototype his analog circuits (courtesy National Semiconductor and Linear Technology, respectively).





One of the fastest ways to prototype a circuit—and one that National Semiconductor Staff Scientist and resident analog guru Bob Pease champions—is the “dead-bug” technique, so called because the finished prototype resembles an insect lying on its back with its legs in the air. The technique can use a solid, copper-clad board as a ground plane. You solder the ground pins of the ICs directly to the plane and wire together the other components above the plane. Because the circuit nodes are suspended in the air—hence, the technique’s other nickname, “air-ball” prototyping—the stray capacitance is lower than it would be if the nodes were on a board. The disadvantage of this approach is that it makes it difficult to wire together fast circuits with controlled impedances, although you can use twisted-pair and coaxial cable to connect the parts. For tiny IC packages, you can use a converter board from Digi-Key, Mouser, Newark, Allied, Jameco, or another distributor (Figure 1). Although Pease’s counterpart at Linear Technology, Jim Williams, sometimes uses the dead-bug technique, he prefers to use copper-clad PCB material, cutting off the copper with an X-Acto knife to make the connections.

Another prototyping method is to use perforated Vectorbord, which Vector Electronics introduced decades ago. Start with a perforated board with solder pads (Figure 2) or with both solder pads and plated-through holes. Although this material costs hundreds of dollars per sheet, it allows you to make solid



Figure 1 A huge selection of adapter boards spread out tiny packages to 100-mil spacing.

AT A GLANCE

- Modern prototyping requires a full complement of magnifiers, microscopes, lamps, and tweezers.
- “Dead-bug”- and copper-clad-prototype techniques are fast but lack documentation.
- A wire-wrapped prototype is suitable for low-frequency designs.
- Acquiring a board from a fab house gives you a head start.
- Assembly houses can provide quick board turnaround.
- Your job is to deliver a proven documentation package.

solder joints that can withstand a lot of mechanical stress.

A more traditional prototyping technique, wire wrapping, works well for digital designs; you can employ it for analog designs only if there are no fast signals on the board. With the technique, a process that Cooper Hand Tools developed (Figure 3), you need to be aware of undershoot problems on the clock. In one case, an engineer spent two weeks trying to figure out why his Z80 processor wouldn’t work. He found that the wire-wrapped board had caused 10% undershoot in his design’s 4-MHz clock. To fix the problem, he placed a 33Ω resistor in series with the clock circuit to damp out the undershoot. For moderately sized boards, it is worth buying an electric or a manual squeeze-wrap tool rather than spinning the wraps on with a cheap hexagonal barrel tool. You place the unwrapped side of the barrel tool into a drill and use it to unwrap a large number of pins.

Note that, as with all things analog, no one method always works; you may have to combine techniques to get a working design. For example, you might combine sections of copper-clad wiring with areas of dead-bug wiring along with premade demo boards and wire-wrapped areas. You need not use any of these techniques if you can obtain a demo PCB from an IC manufacturer or a reference design from a distributor, such as Avnet.

If the board has more than a dozen traces, you cannot carve it into copper-clad material. However, copper-clad boards with preapplied photoresist are

available from many companies. With photoresist, a light-sensitive material for forming a patterned coating on a surface, you can make films for building two-sided PCBs. The only other tools you need are a darkroom and a laser printer. Be sure to flip the image in your computer so that when you place the art over the photoresist, the laser-toner side presses directly against the photoresist; this approach produces crisper lines. Many companies, including Injectorall, offer both precoated boards and photoresist that you apply to boards yourself.

Another method of prototyping requires no photoresist. You use a laser printer to apply the artwork toner to a special plastic film and press the toner side of this film to a copper-clad board. This toner-transfer method requires heat from an iron or a hot plate; the heat from this equipment transfers the toner on the plastic-film artwork to the board. You can rub a ferric-chloride-soaked sponge over the copper to etch off the exposed copper adhering to the board—a faster copper-removal method than agitated tanks can achieve.

The inventor of this rubbing process, Frank Miller, founder of PCB-fabrica-



Figure 2 Soldering discrete wires to parts on a perforated board is an acceptable way of prototyping circuits.

tion-supply company Pulsar, also added a step to the photoresist process in which a second transfer seals the material. “The key is a hot-roller laminator and the second application of TRF [toner-reactive foil], which adds strength to the toner and allows for direct rub in the etch process,” says Wayne Yamaguchi, owner of Yamaguchi Consulting and a proponent of Miller’s method. “The etch time with direct rubbing is now a minute or two.”

PROVING OUT THE PROTOTYPE

Making boards with CAD (computer-aided-design) files proves the validity of your files early in the design process, so fewer problems emerge when your circuit enters production. The toner-transfer method requires etching and hand drilling. You can also use milling techniques to make your board. LPKF, for example, produces mills such as the one in **Figure 4** that can prove out your Gerber and drill files. Prices for the device start at \$11,900, including software that converts conventional Gerber files into milling-machine-tool paths that isolate the traces from each other. Running the mill tool through those parts of your board with 100-mil (0.001-in.) spacing between traces reduces those spaces to 10 mils. For RF designs in which the shape and proximity of the copper are important, you can set the mill software to precisely replicate the trace isolations. In this mode, the mill operates in raster fashion—systematically sampling a grid pattern of pixel spaces to represent an overall image—over any large areas of copper that the mill removes. This process takes longer than it does to



Figure 4 This LPKF mill can route out prototype boards in less than an hour (courtesy National Semiconductor).



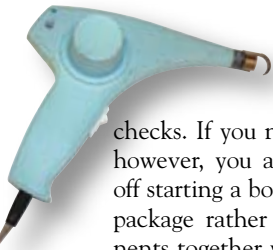
Figure 3 Because of low operating frequencies, wire-wrap techniques work well in this toy-watch-recorder prototype.

simply isolate the nets from each other in one pass.

The mill can in a matter of hours put a complex prototype into your hands. Using a mill, you make further cuts until a trace achieves the proper impedance. Note, however, that drilling between the layers cannot make a via connection; you must solder a small wire in the via’s holes to achieve connectivity. LPKF offers a package that fills the vias with conductive epoxy and supplies small plating tanks for creating multi-layer boards. Find out whether your employer requires vent hoods or hazardous-materials handling of these materials. If your company requires a million-dollar facilities investment for a small plating tank, you will need to deal with the conductive epoxy or soldering wires in the vias.

INTO THE BOARDING HOUSE

After you use some or many of these prototype techniques, a board will eventually emerge. Using dead-bug components and X-Acto knives on copper-clad materials provides proof-of-concept



checks. If you need to make a product, however, you are almost always better off starting a board design in your CAD package rather than soldering components together without documentation. John Massa, consultant at Datadog Systems, points out that, in his 47 years of experience, every project has always come down to the PCB layout and prototyping. “There is rarely a downside to paying for quick-turnaround boards,” he says. “You always end up with a better product.” Your job is not just to get the circuit working but also to provide a documentation package to manufacturing—whether that manufacturer is your own company or a contract manufacturer in China. Toner-transfer prototypes check out your Gerber files but not your drill files. Milling machines prove out your drill files but do not replicate the silk-screen process. Mills have trouble with micro-SMD (surface-mount-device) packages and other CSPs (chip-scale packages). To make a usable board, you must have a perfectly set up mill that is in good condition.

All these constraints make a good case for sending your board design to a PCB-fabrication house (Figure 5). PCB fabrication has in 10 years gone from taking weeks and thousands of dollars to taking 24 hours and hundreds of dollars. Dozens of reputable PCB shops can in a few days turn your CAD files into a board. However, a few companies stand out in their efforts to serve engineers. One that does, Sierra Proto Express, makes two or three boards from your files, delivers them in a few days, and charges less than \$200. According to Sierra’s owner, Ken Bahl, a need for fast-turnaround prototypes emerged in the early 1990s. By 1996, the managers at PCB-fabrication house Advanced Circuits also realized the benefits of quick-turnaround prototypes. Another pioneer, Sunstone Circuits, had previously been the exclusive board producer for Tektronix. These companies can all provide two-layer boards in a day, as well as multilayer boards in a few days, but each emphasizes a different aspect of the prototype service.

Advanced Circuits touts on-time delivery. “We maintain redundant machinery for the entire process,” says Larry McQuinn, vice president of sales and marketing. “If one machine breaks,



Figure 5 Only a board house can make small boards with fine trace widths and spacings.

we can still get you your order on time.” Advanced also provides online, real-time DRC (design-rule checking). You can upload your files to a server, and, within hours, you will receive a report detailing shortages or manufacturing problems. This approach prevents your losing a day or more when the fab house finds a problem and has to halt work while you make a change. Advanced also offers the free PCB Artist layout tool, which creates all the files you need for single-layer and multilayer boards.

Sierra Proto Express has taken another tack: extending the technology in its prototype boards (Figure 6). The company can routinely produce boards with lines as narrow as 3 mils. The company also takes on jobs with 2-mil lines and spaces, buried and blind vias, and laser drilling that allows the buried and blind vias to reside on arbitrary and overlapping layers. Although the company provides copper layers as thick as 6 oz, sensible engineers know that they can’t expect 2-mil spacing and 6-oz-thick copper on the same board. Sierra can also provide 62-mil-thick, 14-layer boards and thicker boards with as many as 30 layers. Features can include 1-mil-diam-



Figure 6 A PCB-fabrication shop expends huge efforts to electrically and optically inspect your board (courtesy Sierra Proto Express).

eter test pads and state-of-the-art finishes and laminates.

Sunstone is also far from being a bare-bones single-layer-fab shop. The company offers quick-turnaround prototypes and the free PCB123 CAD tool. The company has also developed DFM (design-for-manufacturing) plug-ins for popular layout packages, such as Altium's Designer and Cadsoft's Eagle, that you can set for the appropriate Sunstone design rules. These packages flag any mistakes you make as you are designing the board. Sunstone also works with Screaming Circuits to assemble your board. The company's CAD package can do price checks from Digi-Key, and Sunstone can save shipping and logistics by overseeing the assembly of your boards.

Although these three US companies meet all local, state, and federal standards for pollution control, you can also find responsible PCB fabrication offshore, such as from PCB-Pool, an Irish company that has been making prototype boards since 1994. The company charges by the panel and does not charge for routing, so if you have many small boards requiring routing, PCB-Pool is a good option. The company does charge for the silk-screening and solder-masking options, but the prices are competitive.

ON TO ASSEMBLY

Now that you have a prototype board, you must assemble it or contract with an assembly house to do so (see **sidebar** "PCB assembly: home-brew or send out?" with the Web version of this article at www.edn.com/081205cs). During your career, you should endeavor to send out at least one prototype board to a fab

and have a contract manufacturer build it. The experience you gain will help you understand the vicissitudes and exigencies of manufacturing. The fab houses are ready to help, even if you drew your design on a cocktail napkin, and you will be able to lower design costs because you have an understanding of the manufacturing process. "Having a good board shop is part of your being an innovative company," says Amit Bahl, director of marketing for Sierra Proto Express. Even if you have four months

to make a prototype, it is well worth it to have a fab house provide a three-day turnaround on your boards. In that way, your marketing and sales organization will have time to review the product and perhaps get one of your prototypes into the hands of a customer. You will then have the opportunity to make improvements and produce a better version of that board in less than a week. By the time the product comes out, you will have a solid design that does even more than your customer expected. As your competitors are trying to catch up, you can be moving to lower cost or improved performance, staying ahead of the pack all the while. **EDN**

REFERENCE

1 "Analog Breadboarding," Analog Devices, www.analog.com/static/imported-files/rarely_asked_questions/moreInfo_raq_analogBreadboarding.html.

⊕ Go to www.edn.com/081205cs and click on Feedback Loop to post a comment on this article.

⊕ For more information on the companies and resources this article covers, go to www.edn.com/081205cs.

⊕ For more technical articles, go to www.edn.com/features.

You can reach Technical Editor Paul Rako at 1-408-745-1994 and paul.rako@reedbusiness.com.

