

FEATURE ARTICLE

Stuart Ball

Breadboarding

With this offering, Stuart pumps new life into the dying technique of prototyping. For electronics experimenters, the cost of making a circuit board is too high for building only one. There is another choice. One-of-a-kind prototypes can be produced without having to lay down the cash for printed circuit boards. Hand-wiring may seem laborious, but you'll end up with an effective low-cost alternative.



Sometimes the best way to test a new circuit is to prototype it. If you are planning to build only one, the prototype may be the only one you ever put together. Unfortunately, prototyping is slowly dying out in the electronics industry. Modern CAD tools let you produce a circuit board layout in a few hours. And, rapid prototype circuit board shops can get you boards faster than you can get the parts to populate them. Compared to the time and labor involved in hand-wiring a prototype, making a circuit board is inexpensive for most companies in the electronics business.

However, the electronics experimenter has a different economic scale. Paying \$600 or more for tools for something you will build only one of is a difficult decision to justify.

In this article, I will look at a way to produce one-of-a-kind prototypes without resorting to printed circuit boards. This method is suitable for high-speed logic and microprocessor

circuits and is best suited for building digital circuits. These techniques can also be applied to analog or RF designs.

The basic prototyping material is perfboard, such as that sold at Radio Shack. This material has holes on 0.1² (2.54-mm) centers and is available in several sizes. The holes match the pin spacing on DIP ICs and many other components.

You can get perfboard with plated-through pads on every hole, but I prefer the type without pads. It's cheaper and, as you'll see, more flexible when using the techniques I'm going to tell you about.

GROUNDING

A key area where many digital and microprocessor designs have problems is grounding. Early digital designs often used two-layer printed circuit boards, with power and ground traces mixed with the signal traces on the top and bottom layers.

As clock speeds and edge rates go up, simple grounds such as this are less effective. A modern production digital design will typically use one or more ground planes between the signal layers to get a low-impedance ground. It is difficult to duplicate this with most prototyping techniques. Ground wires connecting the ground leads of the parts work for low-frequency designs, but the overall impedance using this technique is just too high for many fast components such as ACT logic.

The heart of my prototyping technique involves the use of adhesive copper tape to make a ground plane, running the copper strips between the IC leads and around the edge of the board. This arrangement effectively makes a copper grid. The copper tape makes a low-impedance connection even at high frequencies.

Figure 1 shows two sketches of a prototype board using this technique. The cutaway view shows 8- and 14-pin

DIPs mounted on a perfboard with copper tape on the bottom of the board run between the IC leads and around the edge of the board.

The vertical ground strips are soldered to the horizontal strips where they cross at the edges of the board. The ground pins on the ICs are connected directly under the IC by running a bare wire from the IC lead to the copper strip.

The full drawing in Figure 1 shows a complete board with several DIP ICs and a few discrete components mounted on the board. The copper tape attached to the bottom of the board is shown with dashed lines.

Bypass capacitors are mounted near the ICs, and one lead can be connected directly to the copper ground strip, just like the IC grounds. Any discrete component leads that are connected to ground can be bent over and soldered to the ground strip.

PLACING COPPER STRIPS

When you place the copper ground strips, you should burnish them with a hard, round object such as the back of a spoon to force the adhesive into contact with the board. Don't use anything with an edge or you will tear the copper, and don't apply so much soldering heat that the adhesive lets go from the board. Be careful when working with the copper; it works like tape, but the edges of the foil can be sharp.

This technique is effective and produces a low-noise ground that is suitable for most digital/microprocessor projects you're likely to encounter. 3M makes copper tape intended for EMI shielding that will work (also available

through Newark Electronics). This tape (3M's 1181 and 1194 series) comes on a roll with a peel-off liner. The tape is available in widths 0.25" and wider. 3M also makes a tinned copper tape, the 1345 series.

Many hobby stores sell copper tape with the metalworking supplies. The 3M tape doesn't seem to tarnish as much as the hobby store tape, but the hobby store tape is considerably less expensive. Whichever tape you use will need to be cut to the proper width. You can do this with a sharp knife or scissors.

Don't try to use the copper tape to hold a lead on the board. The adhesive will eventually let go. If you have a component with some stress on it, punch a lead through the perfboard and bend the wire around the top, as shown in Figure 1.

Don't try to use this grounding

technique with aluminum or stainless steel tape. Solder won't stick to it.

For a low-impedance ground, you can cover the entire board with tape on one side, then cut out slots where component leads will go. This is more tedious and requires more care to avoid shorting, but it is nearly as good as a ground plane.

Although the sketches so far have shown vertical copper strips connected by horizontal ones along the edges of the board, other layouts are possible. If you are working with a lot of logic on a large board, you might want to add more horizontal strips for lower ground impedance. One of the advantages of this method is that you can put the grounds wherever you need them.

POWER WIRING AND SUPPLY BYPASSING

In most digital designs, there are one or two digital supplies that need to be low-impedance. Usually, the power leads can be wired between the ICs using 26- or 28-gauge insulated wire. Bypass capacitors are connected between the supply leads and ground strips. Typical bypass capacitor values for digital/microprocessor circuits are 0.001 to 0.1 μF . I usually use red wire for the logic power and yellow or orange wire for other supplies. This makes it easy to trace the wiring later.

For low-impedance power distribution, copper strips can be run on the top of the board, between the IC leads and directly over the copper ground strips. This provides a quality power connection, but it is more difficult to wire and you have to be sure you don't short the upper and lower ground strips simultaneously.

In some cases, you

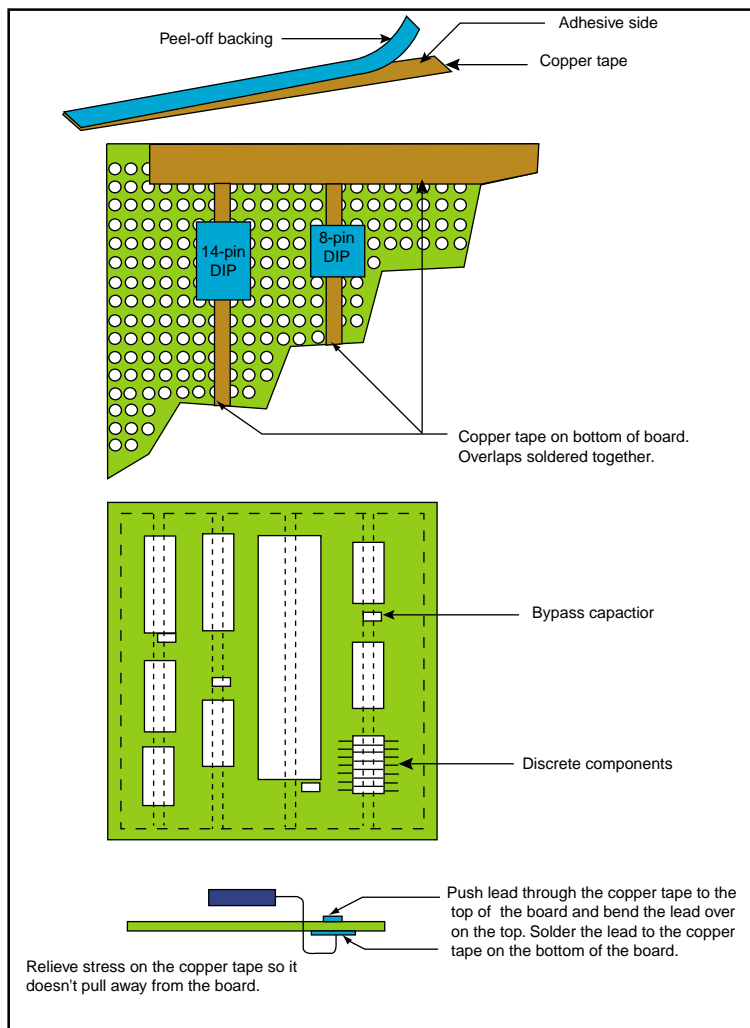


Figure 1—Copper tape grounding provides low-impedance ground connections.

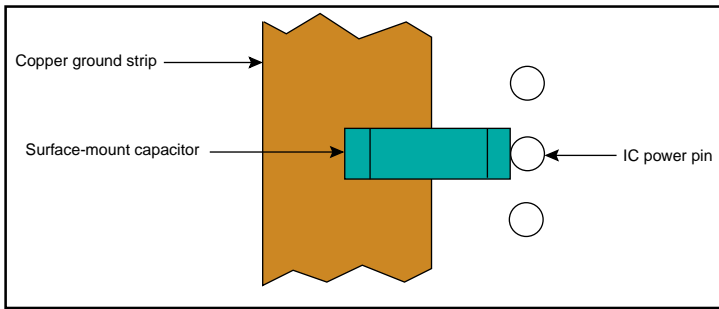


Figure 2—Here, a surface-mount capacitor is used to get bypassing close to an IC power lead. 1206-size capacitors are easiest to work with.

want to locate a bypass capacitor close to the IC power lead, with no intervening wire. An example of this would be a processor or other component running at a high clock rate or generating fast (read, high switching noise) edges.

Figure 2 shows a way to handle these cases. In this example, a surface-mount capacitor is soldered directly to the IC lead and copper ground strip. You will typically use a 0805- or 1206-size surface-mount capacitor. Be careful not to apply so much soldering heat that the end caps detach from the capacitor.

LEAD WIRING

I like to mount the ICs with a couple rows of holes between them (see Figure 3). The signal wires are then run on top of the board using 30-gauge wire-wrap wire. The wire is passed through the holes next to the IC or component leads, and about 0.25" is stripped and wrapped around the lead. This is then soldered. The holes in most perfboards are large enough to accommodate two wires.

The wires are run in the channels between the ICs. Placing the wires on the top of the board in this way reduces the chance of accidentally breaking wires off the board, something that is always a problem with point-to-point wires routed on the bottom of the board. For areas where a lot of wires have to be run (such as a micro-processor data/address bus), I like to leave three or four rows of holes between the ICs.

There is often a temptation for new builders to pull the wires tight when making these connections. It is better to leave a little slack in the wires to prevent wire breakage and avoid warping the board. It's really frustrating to check out everything then have it quit

working because you broke a too-tight wire when you bolted the board into the case.

It's a good idea to socket the ICs on any prototype. The cost of the sockets raises the cost of the prototype somewhat, but sockets permit you to solder the parts without danger of damaging the ICs and then plug the ICs in when you're done.

TEST POINTS

When debugging a breadboard, you are always looking for a place to ground your oscilloscope or logic analyzer probes. One advantage of the copper tape grounding scheme is that you can punch through the tape with a sharp tool and solder component leads directly to the copper.

You can add ground test points in the same way. You can use a piece of resistor lead, bend it into a "U" shape, push it through the copper, and solder it. For a more rugged test point, use the Keystone 5000 series parts. The leads on these parts can be squeezed to fit through the holes on the perfboard, and they come with plastic standoffs in different colors so you can make the ground test points black.

SURFACE-MOUNT PARTS

The electronics world is moving to surface-mount parts for most products. Many parts are still available in both DIP and surface-mount packages, but many newer parts are available only in surface-mount packaging.

The easiest way to handle surface-mount packaging is to convert it to 0.1" spacing. If you can get your parts in plastic PLCC packages, you can get sockets with holes on 0.1" centers that will accommodate them.

For dual in-line SMT packages such as SOICs, you can get adapters that consist of a SOIC pattern on top and 0.1" spaced pins on bottom. You solder the SOIC to the top and plug the pins into a standard IC socket. These adapters are expensive, so you won't want to fill an entire board with them. Aries makes a typical line of these parts, series 35000x.

Finally, you can get perfboard with holes on 0.05" centers instead of 0.1" centers. This will let you prototype with SMT devices, as long as the leads are no closer than 0.05". I have found that the best way to wire this perfboard is to bend the IC legs so they fit into the holes, insert the wire-wrap wire leads into the same hole as the IC leg, and solder them in place. However, it is difficult to do because you need a small soldering iron tip and a steady hand.

There is no doubt that surface-mount parts are harder to breadboard with, and many of the smaller ones are simply impractical to use without a printed circuit board.

OTHER TIPS

Sometimes you have to wire a connector such as an IDC header, where there are two rows of pins but all the wires come off the same side. With a

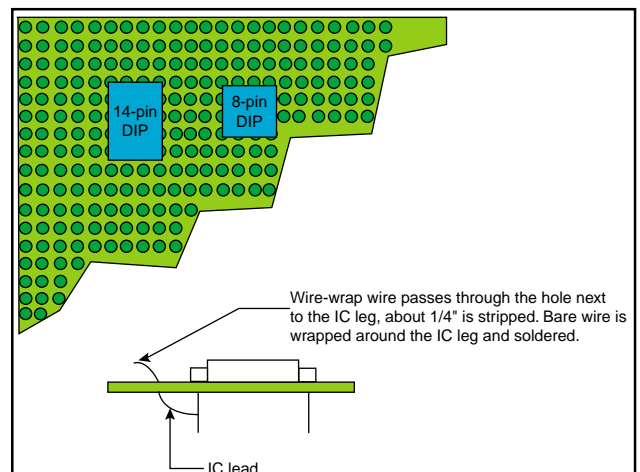


Figure 3—Insulated wires are run through the board, stripped, and soldered to the component leads.

PLCC or PGA socket, you have a similar situation—inner and outer rows of pins, with the wires of the inner pins having to pass through the outer pins.

Figure 4 shows the best way to handle such situations. Wire the outer rows first and solder the connections. Then wire the inner rows, passing the wires between the pins of the outer rows. This way, you don't damage the insulation on the inner wires while soldering the outer pins.

Also shown in Figure 4 is a tip on wiring discrete component leads such as resistors and transistors. When wiring these parts, wrap the wire around the component lead close to the board. If you leave a lot of component lead protruding above the perfboard, the leads are more prone to bend and adjacent leads may short together.

Because the perfboard doesn't have any pads or traces, you can drill holes without worrying about shorting anything. Power devices, such as TO-220 regulators, can be bolted directly to the board.

The copper tape is about the same thickness as 1-oz. copper on a PC board, and you can use it in much the same way. If you are using small surface-mount power packages (such as DPAK and D²PAK), you can solder their power tabs directly to the copper for heatsinking. The only thing to remember is that the tape is stuck to

the board with adhesive, not bonded. Too much heat will soften the adhesive, and the tape will let go. Any mechanical stress will eventually lift the tape unless it is clinched to the board (as described earlier).

Because the copper tape can be placed wherever you want it, you can have two independent ground grids. For example, you can have separate digital and analog grounds connected where power comes into the board.

PUTTING IT ALL TOGETHER

The procedure that I follow when building a breadboard using this method is to first design the circuit. This seems obvious, but you can get into trouble if you start assuming placement of components you know you'll be using before you've designed the rest of the circuit.

Next, lay out parts on the perfboard. Be sure to leave sufficient room for bundles of wires to run between the ICs and for the discrete components. Leave extra space for things like microprocessor data and address buses.

After laying out the parts, drill mounting holes. If you leave this for later, you may find that you've placed wires or copper where you want the mounting holes to be.

Then, place the copper ground tape, soldering the intersections. Remember,

be careful not to cut your fingers on the edges of the tape.

And finally, place the IC sockets and connectors (soldering the ground pins to the copper tape), wire the IC power connections (be sure to wire the bypass capacitors), and wire the remaining connections.

OTHER TECHNIQUES

Wire-wrap is a similar prototyping technique, using IC sockets with long pins and wrapping stripped wire-wrap wire around the pins. Wire-wrap can be used with the copper ground strip technique, but the wire-wrap sockets are relatively expensive. They often cost more than the ICs themselves!

Wire-wrap has an added disadvantage in that the long wire-wrap pins make it impossible to do any kind of low-profile mounting. Any wire-wrap project will have a project board with a thickness of at least 1". In addition, many components (such as connectors) are not available in wire-wrap versions.

Many experimenters make their own circuit boards for projects. There are drawbacks to these methods. The simplest way to make a circuit board is to manually draw the circuit pattern on a blank PC board with an etch resist pen or dry-transfer patterns (both available at Radio Shack). The problems with this approach are circuit density, registration, through-board connections, and circuit quality.

First of all, you can only get so dense with those traces, especially because the circuit layers have to be shared with the power and ground connections. There is no way to get more than two layers with any manual process

Secondly, a double-sided board is difficult to keep registered, unless you drill all the holes first and place the pads/traces second.

Also, any connection that has to be made through the board requires that a wire be run through the hole and soldered on both sides. Component leads with traces connected to the top and bottom pads have to be soldered on both sides of the board to make the connection.

And finally, whether using an etch-

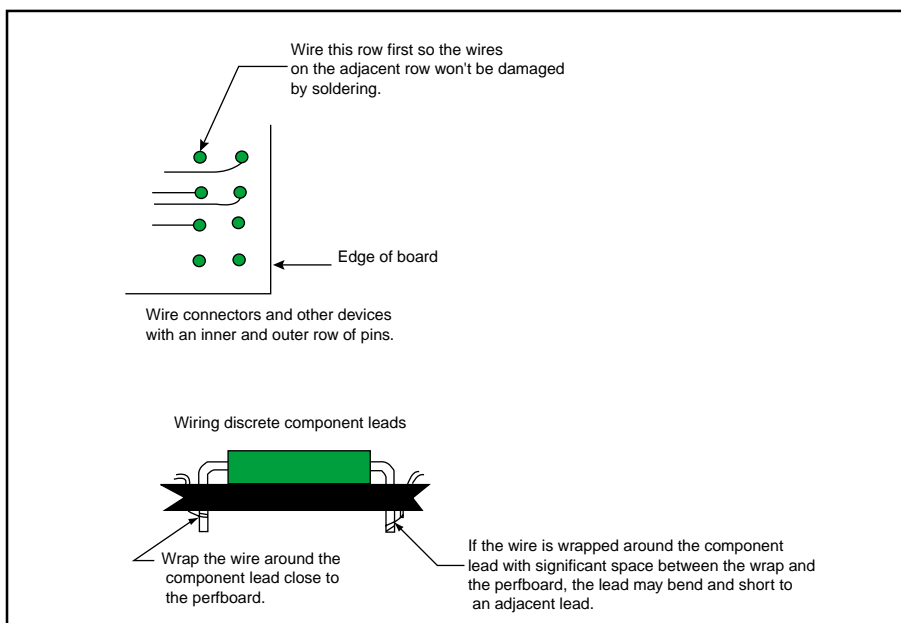


Figure 4—Here are some wiring tips for devices with two rows of pins and for wiring discrete components.

resist pen or dry-transfer resist, it is difficult to make the board without some breaks in the resist. These result in open traces that have to be located and fixed.

Some experimenters use photoetching, which usually eliminates the open-trace problem but still leaves all the others.

Clearly the best way to make printed circuit boards is to use a CAD package and send the resulting Gerber files to a board shop. This takes care of registration, through-hole plating, and allows multiple layers to be used. The drawback, of course, is the cost. Most circuit board houses have high initial setup fees and a minimum order requirement. However, there are services on the Internet that cater specifically to the hobbyist by reducing the setup fees and providing a bare-bones two-layer process.

Although you wouldn't want to hand wire a 500-MHz Pentium processor, the copper-tape and wire method of prototyping allows you to experiment with higher speed devices and parts than traditional prototyping techniques allow. ☐

Stuart Ball, P.E., is an electrical engineer with 20 years of experience. He has written numerous articles and three books about microprocessor-based designs, all published by Butterworth-Heinemann (www.bhusa.com). You may reach Stuart at sball 85964@aol.com

ExpressPCB
Engineering Express
www.expresspcb.com

SOURCES

EMI copper tape

3M

http://www.3m.com/ehpd/emi_shielding/emi_shielding_tape.html

Adapters and sockets

Aries Electronics, Inc.
(908) 996-6841
Fax: (908) 996-3891
www.arieselec.com

Newark Electronics

(800) 463-0275
(773) 784-5100
Fax: (773) 907-5339
www.newark.com

Circuit Cellar, the Magazine for Computer Applications. Reprinted by permission. For subscription information, call (860) 875-2199, subscribe@circuitcellar.com or www.circuitcellar.com/subscribe.htm.