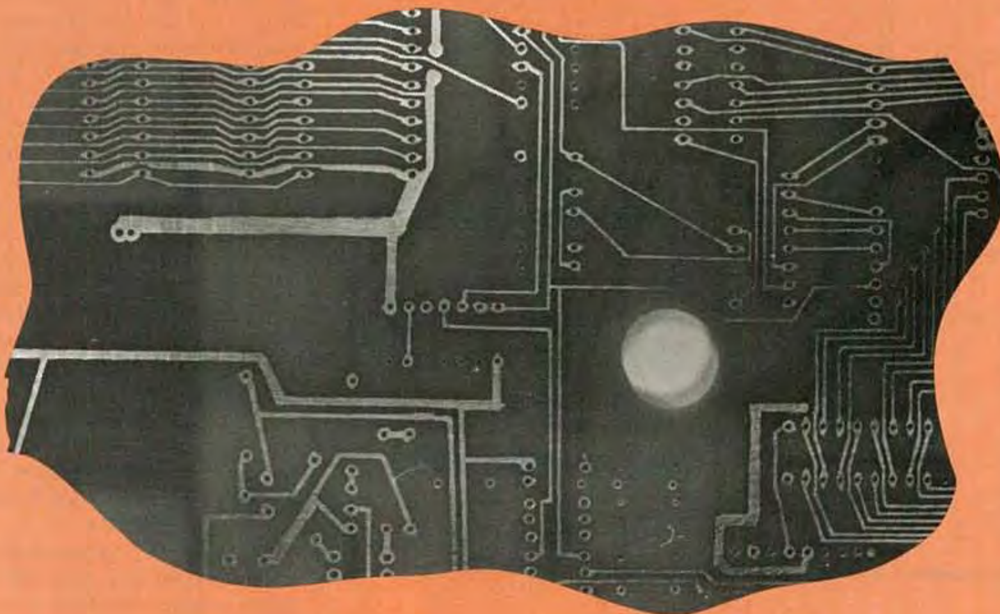
HOW TO



ETCH YOUR OWN PC BOARDS

Even the most complicated printed-circuit boards are easy to make with this simple technique. No exotic materials are required.

NEIL R. DAVIS

ONE OF THE MORE DIFFICULT PROBLEMS YOU MAY RUN INTO when you build a complicated project is laying out and etching the circuit boards. There are several techniques that can be used, but all of them seem to have one drawback or another. For small boards, the usual methods include dry transfers, drafting tape, or etch-resist pens.

The dry transfers produce clean, sharp lines, but are difficult to use where there is a lot of fine work or where there are bends in the pattern. In addition, they don't always adhere properly throughout the etching process; you're likely to find the transfer floating in the bath while the foil disappears in the etching solution. The same problem occurs with narrow drafting tape, although the wider strips work reasonably well. Etch-resist pens are a good idea, but often don't protect the copper adequately.

Large boards, or microprocessor boards with complex buses, are difficult to make without using photographic techniques. Unfortunately, the photographic equipment and drafting aids needed make this technique somewhat impractical for the low-budget hobbyist.

FIG. 1—TINTED SHELLAC, a *Rapidograph* or similar drafting pen, a high-speed rotary drill, and drill bits are the only tools you need to make high-quality PC boards.

The easy way

The method described here uses a different etch-resist—ordinary shellac—that is laid down using a drafting pen such as the Koh-I-Noor *Rapidograph*, or similar pens made by Keuffel & Esser and Staedler-Mars. The tools that you'll need (including a PC-board drill) are shown in Fig. 1. The shellac should be tinted with a paint pigment (blue makes for good contrast) so that it is visible on the copper. Mix the shellac and the pigment thoroughly, and use the mixture undiluted.

To lay down a trace, simply draw a line on the copper, and go over it again if necessary, judging the thickness of the trace by the density of the pigment. For very fine work, it may help to dab the pen on the copper like a pogo stick to build up an even layer. Be careful that you don't leave an "empty" area in the center of a line.

What if you make a mistake? For smaller errors, scrape off the dried shellac with a small knife-blade, then clean the area with an ink eraser before redrawing. Larger mistakes can be removed easily with a cotton swab dipped in shellac thinner (methanol). Use the same solvent to clean the shellac off the board after etching, and to clean out the pen—it won't hurt the plastic parts. As a matter of fact, the shellac can be left in the pen for several days or more without drying up or even getting gummy.

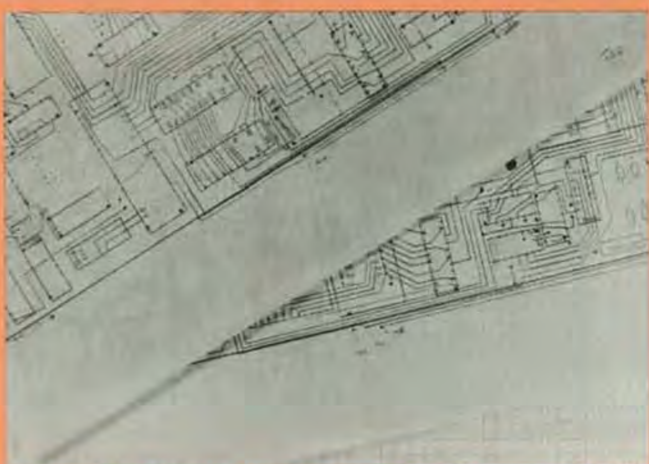


FIG. 2—DRAW YOUR PATTERN on a sheet of tracing paper that has been attached to a sheet of graph paper. The graph paper should have 0.1-inch spacing. For double-sided boards use a second sheet of tracing paper.

The technique is just as useful for small boards as it is for large ones. For small boards, simply copy the artwork from an article or draw your own on the copper. For larger boards, you will generally want to lay out your artwork on paper before working in shellac (see Fig. 2). The artwork can be done much more quickly if you use tracing paper or onionskin typing paper stapled or taped to a sheet of graph paper that has 0.1-inch spacing (the same spacing as IC pins).

For double-sided boards, add a second sheet of tracing paper for the traces on the other side of the board. As a general rule, put the most complicated traces (data bus, address bus, etc.) on the bottom side of the board, and use the top side of the board to "cross over" the traces on the bottom. Remember that the holes will not be plated through so allow for feedthrough jumpers where necessary. Use No. 28 or 30 wire-wrap wire for the jumpers and solder them on both sides of the board.

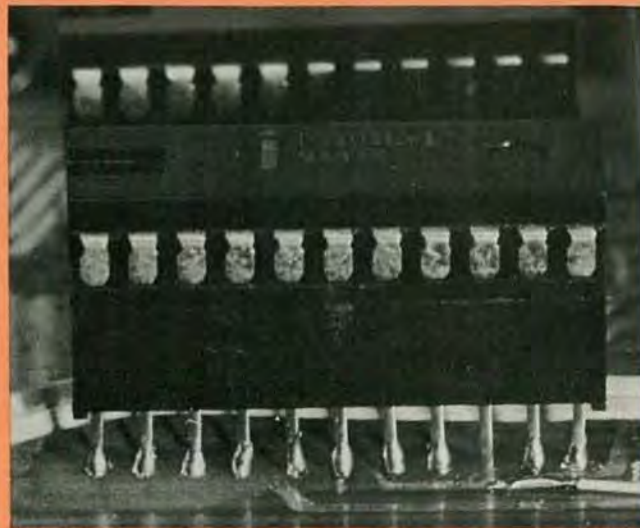


FIG. 3—SINCE THE HOLES ARE NOT PLATED THROUGH, wire-wrap sockets should be raised about 3/16 inch off the board so that the pins can be soldered on both sides.

For complicated boards, it might be a good idea to use a hybrid wire-wrap/PC construction technique: Use the bottom side of the board for most of the circuitry; use the top side of the board for the power and ground, and wire-wrap sockets and wire-wrapping for the runs that won't fit on the board. The wire-wrap sockets should be raised approximately $^3/_{16}$ inch above the board to allow them to be soldered to both the top and bottom traces, as shown in Fig. 3.

Another approach for very complicated boards is to use both sides of the board for the circuit traces, and to hard-wire the ground and power lines later using jumpers. That is what was done in the board shown in Fig. 4: a microprocessor board with keyboard and display interface, a serial data interface, a total of 52 ports, and a battery-powered back-up memory.

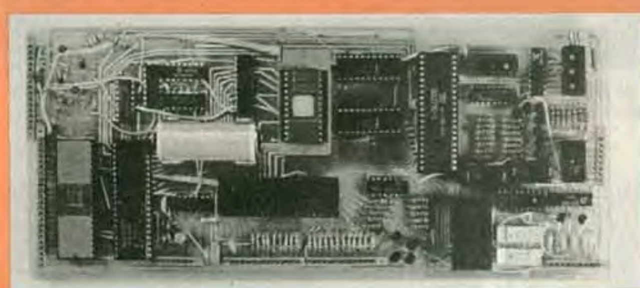


FIG. 4—THIS MICROPROCESSOR BOARD was built using the hybrid wire-wrap/PC construction discussed in the text. The board has a keyboard and display interface, a serial data interface, a total of 52 ports, and a battery-power memory back-up.

Step-by-step

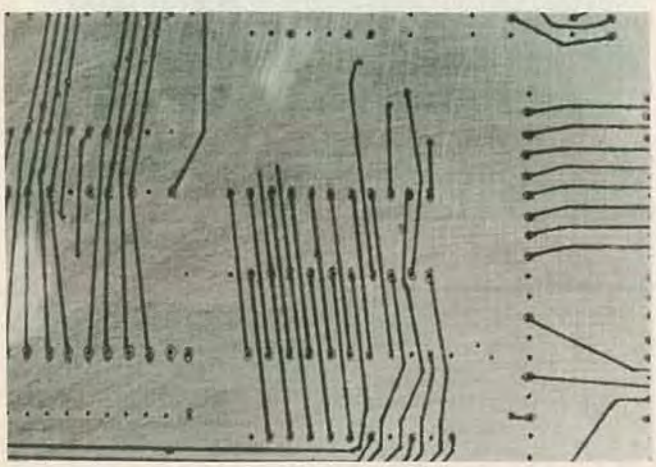
The first step in making a board is drilling the holes. That is best done with a Dremel or similar high-speed rotary drill. Make a photocopy of your completed layout for use as a drilling guide, tape it to the board, and mark all of the holes with a center punch or scratch awl. For small boards, a piece of perforated construction board using 0.1-inch spacing makes an excellent drilling guide (see Fig. 5).

Use a No. 62 or No. 60 drill bit for resistor and capacitor solder pads, and for IC solder pads where traces do not run between the pads; a No. 65 bit where the traces *do* run between the IC solder pads, and a No. 74 bit for feedthrough holes.

After you've finished drilling, clean off all the burrs with a fine file, sand the board thoroughly with No. 400 sandpaper, and then clean the board thoroughly by scrubbing the board with cleanser and water.

FIG. 5—WHEN DRILLING A SMALL BOARD, a piece of perforated construction board with 0.1-inch spacing makes an excellent drilling guide.

FIG. 6—PLASTIC KITCHEN WRAP is used to protect sections of the board that you are not working on from skin oils and smudging.



CLOSE-UP VIEW OF A PC BOARD under construction showing the shellac resist in place.

When that has been done, you can start drawing the pattern on the board. Complete one side of the board before starting the other, and begin by drawing the IC pads with the tinted shellac. Then, draw the larger traces using a drafting pen with a No. 2½ tip; use a No. 1 tip for traces that go between the IC solder pads and for labeling the board. Use a straightedge to guide the pen, and plan your work so that you will not have to lay the straightedge on wet shellac. Cover the portions of the board that you are not working on with plastic kitchen wrap, as shown in Fig. 6, to prevent contaminating the copper with skin oils and salts, and to keep from smudging the board.

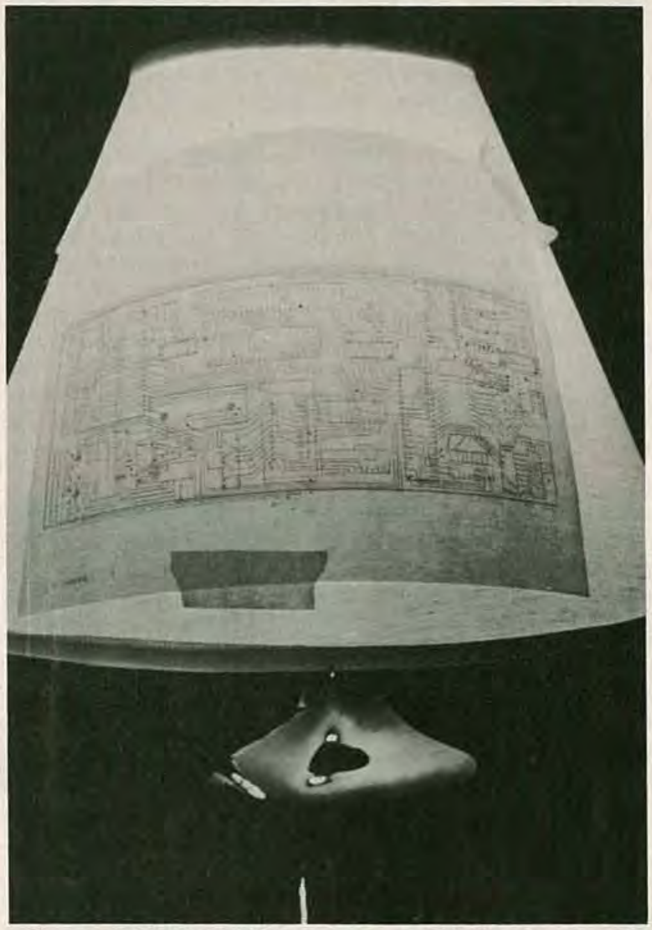


FIG. 7—TO OBTAIN A MIRROR image copy of a foil pattern, tape the art to a lampshade and trace it.

To draw the traces on the top side of the board, simply copy your artwork. For the bottom side of the board, you will have to turn your artwork over and hold it up to a light (taping the drawing to a lamp shade as shown in Fig. 7 is one approach that will make the job easier) before copying the artwork onto the copper.

After etching, clean the board thoroughly with methanol to remove the shellac. If you have followed the instructions, and have been careful, the result will be a clean and sharply-detailed foil pattern.

Be sure to solder all feedthrough jumpers on the top and bottom sides of the board before installing the sockets or IC's. Try to use sockets for the larger or more expensive IC's, or where you anticipate circuit changes. Soldering the IC leads is usually safe, but it is a good idea to use a temperature-controlled iron with a fine tip to avoid any damage resulting from excessive heat.

When the board is completed, clean off all solder residue with lacquer thinner or carburetor-cleaner spray. As a final step, coat the copper traces with a varnish or lacquer to prevent corrosion. **R-E**