

Making Two-Sided Circuit Boards by the Photoetching Process

Neat High-Accuracy Alternative to the Paint-and-Etch Method

BY MICHAEL RATHBUN,* WA0PZI

SOME months ago I began a project using 7400-series digital ICs, which had become inexpensive and readily available from surplus distributors. With some 25 circuit packages involved, the prospect of making printed-circuit boards by the paint-and-etch method I had used previously was not very heartening. The complexity of the circuit made two-sided boards desirable, if not mandatory. With some ICs mounted quite close together, the copper traces had to be thin, straight, and of predictable and uniform width. The "dead bug" method (ICs fastened on a board, pins up, and connected by lengths of wire) was rejected for obvious reasons.

KØHYG suggested photoetching, but mental pictures of lights, etching tanks, and expensive optical equipment made me tend to reject this method as too expensive and troublesome. However, some research and experiments convinced me that it could be practical, even for a limited budget and restricted work space. In recent months I've made a number of circuit boards with methods described herein. In accordance with one of the more obvious implications of Murphy's Law, "Experience is directly proportional to material ruined," the knowledge gained was a bit costly. The information below is offered in the hope that readers may avoid some of my mistakes.

How Photoetching Works

In the other process most often used by amateurs, the etch-resist pattern is applied to the copper surface of the board with brush or pen. The board is then immersed in an etching solution, to remove the unwanted copper not covered by the coating. In the photo process, the metal surface is

* 4200 Mercier, Kansas City, MO 64111.

completely covered with a photosensitive resist. This lacquer-like substance can be removed with certain solvents, usually trichlorethylene, until it is exposed to ultraviolet light (UV), after which the resist becomes relatively insoluble.

The pattern of the circuit is transferred by exposing the sensitized board to UV through a photographic negative of the circuit pattern. On this negative image, areas corresponding to those from which the copper is to be removed are opaque. The photoresist receives no light on these areas, so it can be washed away with solvent after exposure for the proper time, leaving a pattern of resist on the copper surface. The unwanted copper can then be etched away in the normal manner.

The negative of the circuit pattern can be produced in several ways. The simplest is probably the *mechanical negative*,¹ a transparent sheet with a coating that is opaque to UV. Portions of the coating are removed with a sharp tool, to produce lines and connector pads.

In another method the pattern is drawn in dark ink on semitranslucent paper,² and the negative produced by the contact-print process. This is useful in duplicating a project for which a full-size circuit-etch layout is provided. Trace the pattern from the article, or make a Xerox type of copy (if the foil pattern alone is shown), and make a contact negative of it.

A more versatile method involves making tape-artwork transparencies in three steps: the layout, pattern transparencies with graphic-art aids, and contact negatives of these transparencies. Advantages include precise control of line width and path, ease of making corrections and design changes, uniform size and shape of conductor terminals, and ease and precision in fabrication of two-sided boards. This method is covered in detail here.

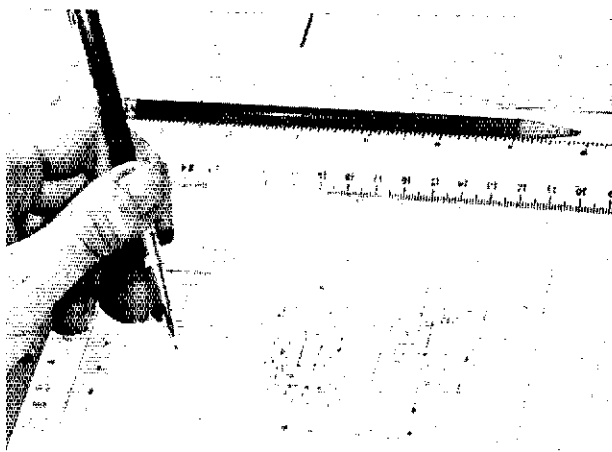
Preliminaries

It is assumed that you are building from a

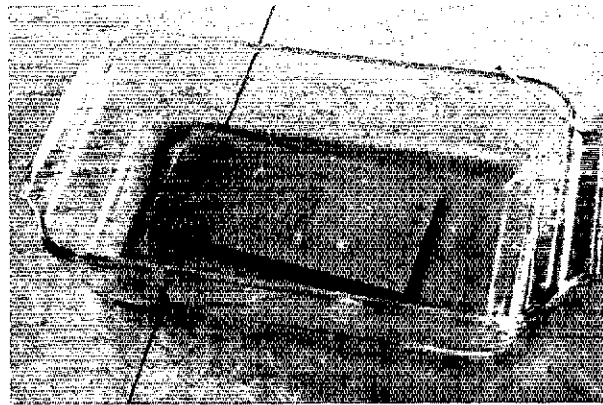
¹ Available from Kepro Circuit Systems, 3630 Scarlet Oak Blvd., St. Louis, MO 63122.

² Kit for this process is available from Lester Kisorsy, 10 Aiyer Street, Newark, NJ 07105.

Step one in production of a double-sided board in preparation of a complete circuit diagram in full-scale form. All leads are shown on this drawing, made on grid-graph paper. Connections on the top of the board are drawn in red. Those to be on the bottom are drawn in blue. Connections on both sides of the board are done in black.



Etched board can be tin plated easily, for improved ease in soldering. Here the board is shown, bottom side up, in the tin-plating solution.



design known to be workable with the parts on hand, and that major decisions have been made on dimensions of the circuit assembly, placement of it relative to other parts, type and placement of mounting hardware, ventilation of circuitry, and so on. Following is a list of minimal supplies, in addition to the recommended tool and supply list given in the *Handbook*:

Drawing board. A kitchen cutting board at least 10 by 14 inches will do

Sharp, fine-pointed knife for use with drafting aids
Red and blue colored pencils

Pressure-sensitive drafting aids (see below)

Transparent plastic sheets of proper size (see below)

Drafting or masking tape

Darkroom supplies, including:

1) Safelight (red Christmas tree bulb in night-light fixture will do nicely)

2) Several plastic clothespins

3) Darkroom tongs (the wood variety is desirable for use during the etch-resist development stage — the developer is an excellent solvent for many plastics)

4) Two or more glass trays at least 20 percent larger than circuit boards to be fabricated

5) Sheet of glass approximately 20 percent larger than circuit board; may be clear Plexiglas

6) No. 2 photoflood bulb with suitable (heat resistant!) fixture

7) Seven-watt incandescent bulb with suitable fixture, suspended about 3-4 feet above work area

8) Kodalith developer or equivalent

10) Photographic fixer (sodium thiosulphate)

11) Circuit board developer (trichlorethylene)

12) Electroless tin plating solution (optional) available from Kepro Circuit Systems, Inc., 3630 Scarlet Oak Blvd, St. Louis, MO 63122, Catalog No. ITP-1PT

13) Presensitized copper-clad circuit boards of proper size. Available from Kepro, or through Burstein-Applebee, 3199 Mercier, Kansas City, MO 64111

14) Ferric chloride etchant solution.

The work area used for the photochemical processes should be near a source of running water, if possible, and should have a work surface at least 2 feet x 2 feet (0.61 x 0.61 m). When this work was done I lived in a basement apartment with very limited space, and was able to carry out these processes by using as a work space the top of my washing machine, which is located in a combination bathroom-laundry room. The room must be sufficiently dark during negative preparation that you cannot read the print on this page with dark-adapted eyes. Kodalith film has roughly the same speed as contact printing paper, so the total darkness needed when handling regular con-

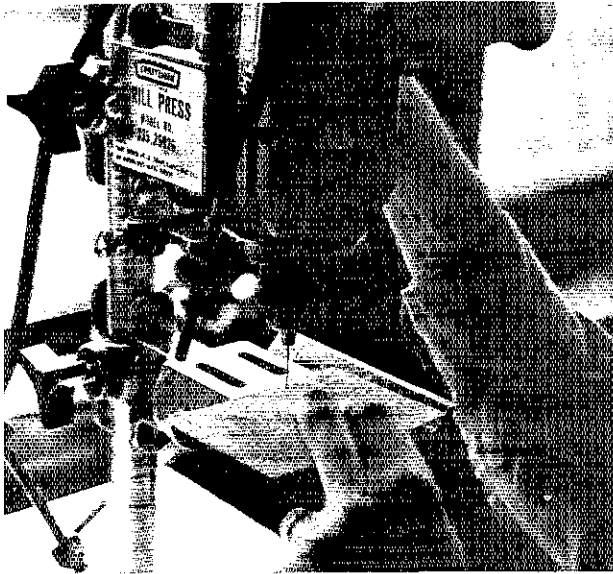
tinuous-tone film is not necessary, although it would be desirable.

Sensitized circuit boards are relatively insensitive to visible light, and may be handled in semidark conditions. Substituting the 7-watt bulb for the Christmas tree safelight during handling of this material has worked quite well. The work area must be well ventilated during etch-resist development, as trichlorethylene fumes are toxic.

Layout

Begin by taping a piece of drawing paper to the surface of the drawing board. If dual-in-line packages will be used in the circuit, 0.1-inch or 0.2-inch grid graph paper is very useful for this purpose, as the pins are spaced at intervals of 0.1 inch, with 0.3-inch spacing between lines of pins. A ruler graduated in tenths of an inch is also a great help in this type of layout. Lightly draw in a full-sized outline of the circuit board as seen from the top side, i.e., the side on which the components will be mounted. The placement of parts can now in large measure be determined by arranging the actual parts on the paper. Keep in mind the location of off-board power and signal connections while arranging the parts. Try to minimize the length of the circuit paths, but avoid crowding the components. The practical minimum distance between ICs in dual-in-line packages is about 0.3 inch. Mark the location of each terminal in the exact place it should appear on the finished assembly. The package outline drawings in semiconductor handbooks are very useful for plotting the spacing of leads.

When the locations of terminals have been plotted, consider the placement of the interconnecting traces. Use of colored pencils is desirable if two-sided circuit boards are to be produced — plot the top-side conductors in red, the bottom-side conductors in blue. Features which appear on both sides of the board can be plotted in black. Experience has shown that the fewer connections made directly to components by traces on the upper side of the board the better. The copper traces can lift from the surface of the board if it becomes necessary to remove a component. This is less likely to occur if component connections are on the opposite side of the board. When the plotting of the conductor paths is complete, it is advisable to recheck the circuitry at least once.



In drilling a finished board, the connector pads provide indications of hole locations.

Transparencies

The finished layout is now duplicated on sheets of acetate, using pressure-sensitive drafting aids. The acetate sheets I have been using were cut from transparent three-ring document protectors purchased at a local stationery store. The pressure-sensitive drafting aids, manufactured by Chartpak Rotex, 2620 S. Susan St., Santa Ana, CA 92704, were obtained through a local drafting supply store. They are die-cut crepe in the shape of component "doughnut" pads, which are placed at terminal connection points, and uniform-width tape, which is used to form the conductor traces. Also useful are dry transfer letters, which serve neatly to identify components, signal points, and so on. The pads are available, in a wide variety of sizes, in rolls of 250, at about one dollar per roll. I have used the .065-inch pads for IC and transistor connections, and other sizes up to 0.1 inch for larger components and off-board terminals.³

The minimum recommended tape width for conductor traces is 1/32 inch (0.8 mm). Maximum recommended current through a conductor of this size is one ampere. Kepro literature recommends no more than 5 amperes per 1/16-inch (1.6-mm) width. I tend to be more conservative, but have never tested etched circuitry at high current levels. The advantage of using the widest possible line is very apparent at high radio frequencies, where conductor inductance begins to be critical.

Wherever practical, the use of a ground plane on the component side of the board is recommended. Advantages of this practice include higher noise immunity and lower ground impedance. Chartpak has available rolls of tape in widths from .015 inch to 2 inches. The larger widths are useful in blacking out ground-plane areas on the transparency. The minimum recommended spacing between conductors is 1/32 inch (0.8 mm).

³EDITOR'S NOTE: Multipad configurations for various pin arrangements of integrated circuits are also available from numerous suppliers, such as those of Bishop Graphics, distributed by dealers who handle G.C. Electronics products.]

To begin making the transparencies, tape a piece of plastic sheet over the layout drawing. Ensure that it is firmly in place, as it may have a tendency to move while pressure-sensitive materials are being laid down and trimmed. First lay down registration marks, so that the finished negative can be reliably placed when exposing the board. My practice has been to place tape exactly over the upper left and lower right corners of the board outline on the layout drawing. Next, place pads over each mark on the layout drawing, indicating a terminal to which a connection will be made on the *upper* (component) side of the board, remembering that there may be some to which connections will be made on both sides of the board. These will appear on both transparencies. The proper technique for laying down pads is something best learned by experience. I recommend practice on a scrap piece of plastic sheet before attempting it on a layout transparency. Absolute precision in laying the pads down from the roll is not vital, as they are easily nudged into the proper place with a pencil point or knife blade.

At this point, tape may be laid down over the red lines on the layout drawing, forming the conductor pattern which is to appear on the upper surface of the board. The tape is flexible, and is easily laid down in straight and curved paths. For sharp bends with the wider tape sizes, it may be necessary to cut the tape and make an angular intersection. Precut elbow bends are available on rolls, but it is unlikely that the average builder would use enough of them to make the cost (250 for \$2.00) justifiable. If identifying letters or numbers are desired, dry transfer sheets may be used to provide uniform, sharp characters.

When the first transparency has been completed and checked, remove it and replace it with a second sheet of similar size. Place identical registration marks on this sheet, and lay down pads for all connections to be made on the *lower* side of the board, and also for any unused pins on IC packages, for which holes will need to be drilled. The pads in this case will serve as drill guides, and if desired, may then be soldered to provide extra mechanical anchorage for the device to be mounted. Next, lay tape over the blue lines on the layout, to form the conductor pattern for the lower side of the board. Be sure to keep in mind that the pattern at this point is a mirror image of the actual pattern, since in a sense you are looking at it *through* the board, from the top. Consequently, if any letters are to be laid down on this transparency, they must be applied to the reverse side, or they will appear backwards on the finished board.

Negatives

When the transparencies are complete, set up the darkroom. The developer and fixer solutions should be prepared according to the directions provided by the packager. I prepare the developer in small quantities by placing approximately one ounce by weight of part A and part B powders in separate dishes, and dissolving them in eight ounces each of tap water. The solutions thus formed are mixed in one of the glass trays just prior to use. This resultant solution will deteriorate within a few days whether or not it is used. A second tray should be placed next to the developer tray, containing either water or a commercially prepared stop bath. I have not found a stop bath to be necessary. A third tray contains the fixer solution. This is reasonably stable and may be reused a moderate number of times. Care must be taken to set up the work area for minimum chance of dripping one chemical into another, especially fixer into developer. If possible, the chemical trays should be located away from the area where exposures are made, to prevent contamination of unprocessed film by droplets of developer or fixer. The area should have the 7-watt bulb suspended three or four feet directly above the work, and the safelight nearby, preferably no closer than three feet. A string on which the wet negatives may be hung to dry should be stretched across a corner of the work area, above head level.

When these preparations are complete, darken the room, and turn on the safelight. When your eyes have become reasonably dark-adapted, open the film package, take out a sheet of film, and carefully reclose the package. The film has a shiny (base) side, and a dull (emulsion) side. To make an exposure, place the sheet of film, emulsion side up, on the work surface directly under the 7-watt lamp (which is OFF at this time!) Ensure that the transparencies and the glass or plastic cover sheet are free from dust, as this will show up as "pinholes" in the dark areas of the negative, and, if large enough in area, as spots of copper on the finished board. Place the first transparency, tape side down, on top of the film. When you are satisfied that it is correctly placed, cover these sheets with a piece of glass.

Now expose the film by turning on the 7-watt lamp for 90 seconds. Turn off the lamp, take the film from under the glass and, holding it with the darkroom tongs, slide it into the developer solution emulsion side down. Move the negative back and forth rapidly in the solution for a few seconds, to

dislodge any air bubbles which may be clinging to the surface. Turn the film over in the solution. An image should begin to form within 15 to 30 seconds, becoming dense black in areas which were exposed to light. Sufficient development is indicated when the image on the base side is as densely black as the image on the emulsion side. When this occurs, remove the negative from the developer, and allow excess developer to drop back into the tray for a few seconds. Then insert the film into the next tray, containing the water or stop bath. Agitate. After 10 to 15 seconds, transfer the film to the fixer solution. Sufficient fixing is indicated when the unexposed (white) areas of the image have become completely transparent. At this time, room lights may be turned on. After fixing, wash the film for 15 to 30 minutes in a tray or sink of clean water. A sink is desirable in that the drain can be adjusted to leak slightly, and the tap left on sufficiently to maintain the water level. After washing, the negative may be hung up to dry with the plastic clothespins.

The second transparency is copied in the same manner, except that it is placed on the unexposed film tape side up. This is done to ensure that, when the negatives are placed over the sensitized board for exposure, the film emulsion will be next to the board. This provides the clearest possible image.

Board Exposure and Etching

When the negatives have dried, the circuit board can be prepared for exposure. The 7-watt lamp may be used for a safelight at this time, provided that it is shaded from the work area. To avoid confusion in handling the board between exposures of the two sides, I mark, with a dot, the upper left hand corner of what will become the component side of the board. A small felt-tip marker is used. Line up the registration marks of the first negative with the upper left and lower right corners of the board. Secure the glass sheet on top of the board and negative. It is advisable to clamp or weight the glass in place, as the negative has a tendency to buckle somewhat from the heat developed by the exposure light source. Check the registration of the negative after placing the glass, and then fix the No. 2 photoflood lamp ten inches above the board and negative. Expose for six to eight minutes. Remove the glass, turn the board over left to right, and line up the registration marks of the second negative. Bear in mind that when the proper side of

(Continued on page 149)

Finished product, in this case a control logic board for a counter, shown top side up.

