

Making Printed-Circuit Boards The Old-Fashioned Way

Using photographic techniques without a camera or darkroom can provide repeatable professional results at low cost

By Anthony J. Caristi

Many readers fabricate circuits on printed-circuit boards for their personal construction projects, often using etching-and-drilling guides such as ones printed in *Modern Electronics*. There are a variety of ways to create such circuits, including dry transfer, resist tape, photographic, and so on. Each method has its advantages and disadvantages.

The photographic method, which is the subject of this article, is often desirable to achieve professional-quality results when the circuit is more complex than a very simple one or when more than one identical board has to be produced. The term, "photographic," is really a misnomer here since a photolab with camera equipment and a darkroom is not required. Instead, it simply means that light-sensitive material is used that must not be exposed to bright light, a negative of sorts is produced, and the circuit board is developed in a solution.

Following the techniques to be described, you will soon be making pc boards that can be virtually indistinguishable from professionally-made boards.

The Artwork Materials

In printed-circuit work, the master layout of the desired copper-trace pattern that is to remain on the board after it has been etched is known as the "artwork." The artwork is laid out

on translucent or transparent film, usually Mylar in composition. The pattern for the copper traces is laid down on this film using self-adhering tapes, "donuts," IC and transistor pads, and other specialized patterns.

Basic to the artwork materials line-up are donut pads and tapes. These are available in a number of colors and "scales" to suit different layout needs. Black is the color most popularly used, though red and blue and sometimes other colors are used on artwork for very complex, multiple-layer boards.

Typical of the scales available are $\times 1$, which is actual finished size; $\times 2$, which is twice actual size; and $\times 4$, which is four times actual size. Larger-than-actual-size and colors other than black artwork materials are usually used where pc patterns are extremely complex and usually call for two or more layers of copper patterns on a single circuit-board assembly.

For our purposes, we'll concentrate on the use of actual-size artwork composed with commonly available black materials since we want to avoid the need for an expensive photography lab or the additional cost of having the artwork generated photographically sized as needed by an outside source.

The materials you'll be using are sold through some electronics supply houses and graphics art supply stores under several brand names. Kits of pc materials and individual items are made by DATAK, Bishop Graphics,

Kepro, GC Electronics and Vector Electronic, among others.

The artwork materials with which you'll be working are simple. These include 10-box-per-inch grid paper or film; transparent or translucent Mylar drafting film; an assortment of donut sizes and tape widths; and DIP IC pad patterns. The only "tools" you need are a hobby knife with spare blades or single-edge utility razor blades and a roll of masking tape.

Based on the types of circuits you generally make into projects, choose an assortment of different tape widths and donut diameters. Typically, tape widths of 0.020, 0.040, 0.080, 0.100 and 0.125 inch will be sufficient for any but high-power circuit pc layouts. If your projects are exclusively low-power digital designs, all you may ever need are the first two tape widths and perhaps the 0.100-inch width for heavy power bus and ground runs. For most work, donuts in diameters of 0.080, 0.100 and 0.125 inch will suffice and perhaps 0.050 inch for the feed-through holes required on double-sided board artwork designs.

Though there are available a variety of different DIP IC pad patterns that have anywhere from 14 to 48 pads in two rows that are 0.30 or 0.60 inch apart, you really don't need every one of these. Actually, if you make a large number of IC-intensive pc guides throughout a year, the best and most economical IC-pad artwork configuration to use has long

strips of single-row pads that can be cut to length as needed and be individually placed on the Mylar film. These strips are available in a number of pad configurations. If you anticipate a lot of work in which conductors will have to pass between pads, use the configuration that has narrow pads that will permit this.

When laying out the actual master artwork, it's a good idea to set the IC-pads, donuts, edge connectors, etc., in place first. Burnish them solidly down. Then make the tape runs that interconnect the various pads and patterns. Don't stretch the tape as you place it down on the film; if you do, it will lift or "creep" toward other tapes or pads in time, creating unwanted short circuits in the artwork. Also, though the tapes are flexible, avoid making sharp turns.

Layout of the master artwork begins with taping a sheet of the transparent or translucent film onto grid paper. If you're copying directly from a printed etching-and-drilling guide like those provided for many of the projects in *Modern Electronics*, tape the film over the published artwork—not a photocopy of it, which may introduce size distortion—and work directly from that. Size the film as needed for the pc layout, leaving an inch or so extra all around.

To generate a pc artwork "master" for an original circuit design, the first step is to make a rough layout of the pc pattern that will be used. The simplest way to go about this is to tape a sheet of tracing paper over the circuit and another over a sheet of grid paper; the latter can have any number of boxes per inch, though 10 boxes per inch will make it more convenient to directly copy the rough.

Draw whatever DIP integrated-circuit pads are needed on the tracing paper laid over the grid paper. Then make each connecting line in turn, drawing in the pads for the discrete components as you go along. Draw each connecting wire run also on the tracing paper laid over the sche-

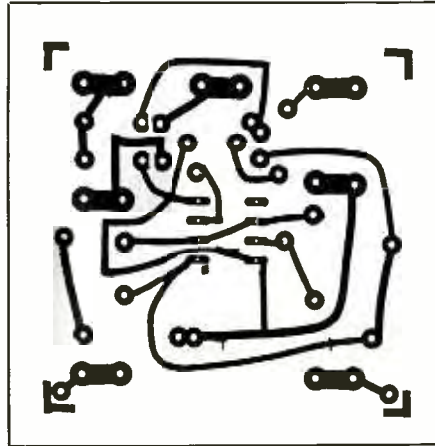


Fig. 1. A typical artwork pattern composed on drafting film using DIP IC pads, donuts and three widths of tape. This view is of the copper side of the board.

matic to keep track of what you've already done and what still remains to be done.

As you come to the end of each tape run, place the edge of the hobby knife's blade or the razor blade against the terminal point of the tape and snap the tape sharply to make a clean edge. Do not use a sawing or slicing action that may cut through the donut or pad at the terminating point. Firmly burnish each tape run to solidly fix it in place on the Mylar film as it is being made. A typical example of a printed-circuit master artwork is shown in Fig. 1.

When making the master artwork for a double-sided board, compose the solder side view first. Flip over the composed artwork and tape to it a second sheet of Mylar and compose the component side view, using the first artwork master as a guide to assure perfect registration (see Fig. 2). If you find the tape runs and patterns on the bottom artwork to be confusing, slide a sheet of tracing paper between the two sheets of Mylar. You'll still be able to clearly see the bottom pattern, but it won't be nearly so obtrusive as you work on the top pattern.

When both original master art-

work patterns have been composed, label the first "solder side" and the second "component side." Also place labels on each to identify the projects for which they were composed. Figure 3 illustrates the artwork masters for a simple double-sided pc board.

Your artwork "masters" aren't meant to be used directly. They're simply the "positives" from which "negative" exposure masks are made. The latter are what are actually used to expose the copper-clad blanks that will become the printed-circuit boards for your projects.

Whenever you finish making an original piece of master artwork, carefully check it against the published artwork or the rough artwork from which it was made. Immediately correct any errors that were made during composition.

Making the Exposure Mask

It's almost a maxim nowadays that you'll be using "B neg" resist for printed-circuit work. With this type of resist, exposure to ultraviolet light hardens the exposed areas. Recall that your master artwork "positive" has black patterns on a clear or translucent background. If this were to be used as is to expose a presensitized pc blank, the UV light would harden the resist over the areas from which you wish copper to be removed. Consequently, you need a *negative* of the master artwork.

We'll discuss use of readily available materials from 3M to generate a negative from the positive master artwork to obtain clear lines and patterns on an orange background that's opaque to UV energy. (Other "reversing" films are available as well from suppliers of professional pc materials. These might have black, dark brown or other base color. The following procedure will work equally well with reversing films other than those from 3M.)

Materials required for the photo-reversal operation, are 3M's No.

77-9802-9269 transparent negative color proofing film and No. 77-9800-7992-3 color proofing film developer. These materials are available from many photo graphics supply outlets and from the source given in the box at the end of this article.

In addition to the film and developer, you need some kind of contact print frame or box, such as those used by photographic hobbyists. Best are commercial models, of course, but you can easily use a sheet of glass and a perfectly flat piece of plywood. The "contact box" will be used to expose the photographic film to a source of ultraviolet light, using the original master artwork as the "positive" from which your "negative" exposure mask will be made.

Your source of ultraviolet energy can be the sun itself, assuming you can depend on bright sunny days, with the sun's rays directly striking your exposure setup. If you don't want to have to depend on the whims of nature for exposing your negatives and presensitized pc blanks, consider investing in a commercial UV light.

Preliminary to setting up the exposure arrangement, determine which side of the reversing film has the emulsion (coating) on it. This can be done by any of several methods. One is to visually inspect both sides of the film to identify the dull or emulsion side. This must be done in subdued lighting or under safe lighting conditions. It also may prove to be a bit difficult because there is very little difference in reflectivity between the two sides.

Another way to determine which side is which is to try to write on both sides near the edge of the film with a soft lead pencil. The side to which the lead more readily takes is the emulsion side.

To make absolutely certain that you've properly identified the two sides of the film, there is another, better, test you can make. That is to gently scrape one surface near the edge of the film with a sharp hobby

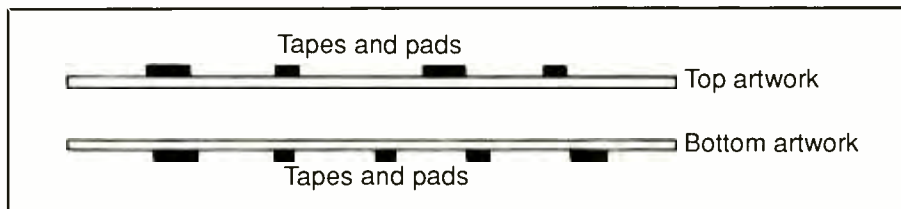


Fig. 2. Drawing illustrates orientations of the solder-side (bottom) and component-side (top) artwork masters during composition.

knife or razor blade. If after a couple of strokes the clear plastic of the base doesn't show through, flip over the film and scrape the second side. This time, when the clear film shows through, you know that you're scraping on the emulsion side.

When you're absolutely certain that you've identified the film's two sides, place a small self-stick label on the emulsion side near a corner. Write on this the words "emulsion side" for future reference.

Conduct all tests on the reversing film under subdued or "safe" lighting conditions. To obtain a safe lighting environment, use a common yellow "bug light" in your work area as the source of illumination. This light can be quite bright without having an effect on the film. Do *not* work in sunlight, which is a robust source of

ultraviolet radiation. If you work in an incandescent-lighted area, do not expose the film directly to the light and keep the brightness of the illumination as low as possible and work as quickly as you can.

The procedure for producing a negative exposure mask from the positive master artwork is fairly simple:

(1) Cut a piece of the reversing film to about 1/2 inch larger in both dimensions than the original master artwork. If the artwork is very small—say, less than 1 inch in a given dimension, add a bit to the reversing film to facilitate easy handling.

(2) Referring to Fig. 4, place the cut piece of film emulsion side down on the wood plate of your contact box and follow up with the original master artwork. Position the art-

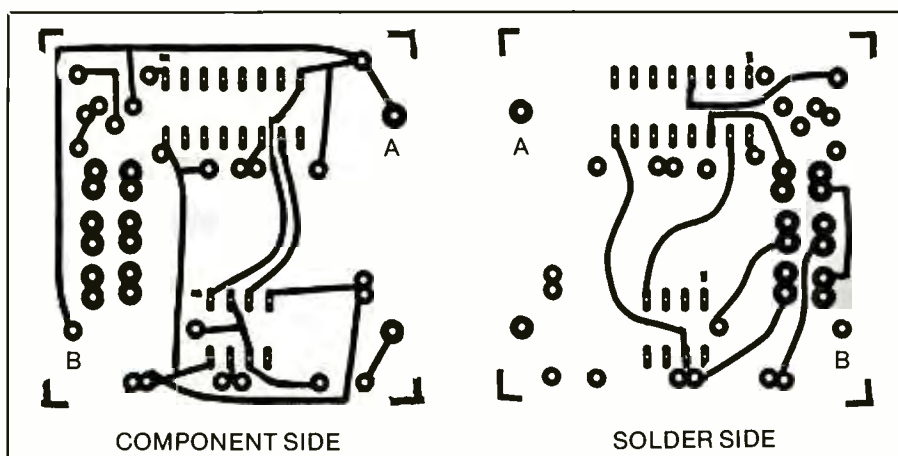


Fig. 3. Solder-side (left) and component-side (right) artwork masters for a double-sided pc board. Arbitrarily selected holes "A" and "B" are predrilled in the pc blank prior to sensitizing to permit exact registration of each negative exposure mask during the exposure operation.

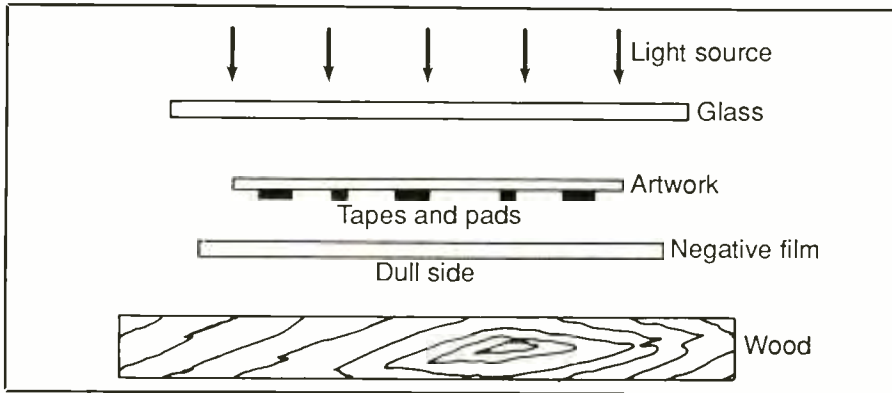


Fig. 4. Shown here are details for making a negative of the original artwork master. A glass sheet holds the artwork master and reversing film in intimate contact with each other during exposure.

work so that it is centered on the film and has its tape-and-pattern side down. Place over this the sheet of glass, the weight of which will force the artwork into intimate contact with the reversing film during the exposure process.

It is *very* important that the artwork and film be maintained in intimate contact during exposure to prevent unwanted light from bleeding under the black pattern of the artwork. If such bleeding should occur, you will end up with a poor quality negative. If you're careful to keep the exposure setup tight, you will end up with a crisp, sharp negative of the original artwork.

(3) Expose the assembly to a source of ultraviolet light or direct sunlight for as long as necessary to "set" the emulsion on the film in the selected areas. Exact exposure time is impossible to give here, since it depends on the strength and purity of the UV energy and other factors. In bright direct sunlight, the exposure time will average about 60 seconds, while for UV lamps it can vary from less than half this time upward.

To determine how long to expose the reversing film for the specific UV energy source you're using, run a series of test exposures on a strip cut from the film. Expose 1/2-inch-wide sections of the reversing-film strip,

emulsion side down, in increments of 10 seconds. Use a permanent marker to write the exposure time of each section of the film and expose through the lettering. (If you're using a UV lamp, maintain the same distance between it and the exposure setup throughout the test exposures.)

After test exposing the film, place it emulsion side up and pour onto it a sufficient amount of developer to soak the emulsion. Work in subdued lighting or under safe lighting conditions. Spread the developer over the entire emulsion-covered surface and allow it to soak in for 15 seconds or so. Use a very soft tissue or cloth dampened with additional developer to gently rub the developer into the film until all softened portions of the emulsion wash away. Then thoroughly rinse the strip of film under running tap water to remove all developer. Examining the result, the proper exposure time for the UV source you're using will be the time marked on the film that appears in sharp contrast against a transparent film background.

(4) After exposing the film through your original master artwork, immediately return to safe lighting conditions. Place the exposed film emulsion side up and "develop" it as you did for the test strip above. Work slowly and carefully and use plenty

of developer when rubbing away the unwanted portions of the emulsion. If the exposure was correctly made, the conductor pattern represented by the tape-and-dot layout on the master artwork will begin to appear as clear against the orange of the reversing film. Continue rubbing until the entire pattern is visible.

Bear in mind that with the developer on the film, the emulsion may be fairly soft at the end of the rubbing operation and can easily be damaged by mishandling. Therefore, handle the developed film with care. You can now work under normal lighting conditions. Thoroughly rinse off every trace of developer under cold running tap water. Then allow the film to thoroughly air dry. Do *not* pat or rub it dry with a cloth or paper towel or tissue.

(5) When it has dried, examine the negative under fairly bright light, comparing it against the original master artwork. If the negative does not have clear, sharp lines in accordance with the artwork, your exposure was too long. If some of the orange background has washed away, the exposure time was too short. If your negative is almost perfect, you can gently scrape away unwanted parts of the orange background, using a sharp razor blade or hobby knife, and touch up with photographic opaquing ink any spots that should be opaque.

(6) If you wish to make more than one printed-circuit board at a time, make as many negatives of the original master artwork as needed. When you're done and all negatives are completely dry, trim them for minimal pc blank wastage and arrange them together with magic or clear tape to make a single multiple-pattern exposure mask. Figure 5 shows the result of taping together of the same pieces of negative artwork for making four identical pc boards at the same time.

(7) Make certain that you identify the side of the negative that repre-

sents the copper-side view of the pc board for which it will be used. Since this will invariably be the shiny side of the film, use a permanent marker to label the film accordingly. While you're about labeling, write on the film the name of the circuit or project for which the artwork was prepared and indicate whether it is for a single- or a double-sided board.

Preparing the PC Blank

Before you can expose the printed-circuit blank for exposure through the negative, the blank must be coated with a light-sensitive photoresist. Of course, presensitized blanks can be purchased from pc materials suppliers at a premium cost. However, if you're planning to fabricate a number of boards, either of the same type or of different patterns for different projects, it makes better economic sense to purchase copper-clad pc blanks and spray-on or paint-on photoresist. You can prepare blanks as they're needed at less cost for each and less risk of the deterioration a presensitized blank is subject to as it sits on a shelf for a long period of time.

To sensitize a pc blank, you need a can of photoresist and compatible

photoresist developer, such as KTI Chemicals' No. 175-2074 resist and No. 175-3572 developer. These items are available from most graphics supply outlets or the source given in the box at the end of this article. Of course, the KTI Chemicals items specified are just an example of similar materials available from a number of other suppliers. Some brands of photoresist are paint-on, others are spray-on. Which you use will be dictated as much by availability as by your personal preferences. Either will work effectively.

Sensitize a copper-clad pc blank as follows:

(1) Cut to a piece of pc blank to a size approximately $\frac{1}{2}$ inch longer and wider than needed for the size of the pc artwork negative to allow for final trimming and drilling of mounting holes for the board.

(2) Thoroughly clean the pc blank, removing all traces of dirt, oil, oxidation and other contaminants from the copper surfaces. Use scouring powder and an abrasive pad or fine steel wool during this operation. If you're preparing a double-sided pc blank, scour *both* copper surfaces. Holding the cleaned blank by its edges only, thoroughly rinse away all

traces of the scouring powder under warm running water. Examine the scrubbed copper. It should be bright and shiny; if not, repeat scrubbing until it is. The blank must be squeaky clean when you're finished. This will be indicated by a smooth coat of water flowing over the copper surface. If it isn't thoroughly clean, there will be defective areas in the exposed blank.

(3) Be sure to handle the board only by its edges from here on. Thoroughly dry the cleaned blank. Do *not* use a towel or other material that might contaminate the copper surface. Instead, allow the blank to air dry, dry it in a warm oven or force dry it with a blow dryer. If the copper cladding discolors slightly, it won't affect the process, as long as the blank is clean.

(4) When the blank is completely dry, take it to an area where the lighting is subdued or you're using a yellow bug lamp. Place the blank on a meticulously clean surface, copper-clad surface up. If you're using a liquid photoresist, pour a small amount of it on the copper-clad surface and use a clean Q-Tip or similar spreader to paint the entire copper surface with an even layer of resist. If necessary, add more resist to assure complete and uniform coverage. Alternatively, pour the resist into a glass—*not* plastic—tray that is large enough to accommodate the pc blank and submerge the blank in the resist.

If you're using a spray-on photoresist, place the blank inside a box to contain over-spray and spray the copper surface with a uniform layer of resist. Make the coating quite wet and free of visible imperfections.

Whichever type of photoresist is used, prop the coated blank on edge to allow excess photoresist to drain off. Leaving the coated blank propped almost vertically, allow the resist to thoroughly air dry under light-tight conditions. The drying process will take two or more hours.

It's absolutely essential that no light be allowed to strike the pc blank

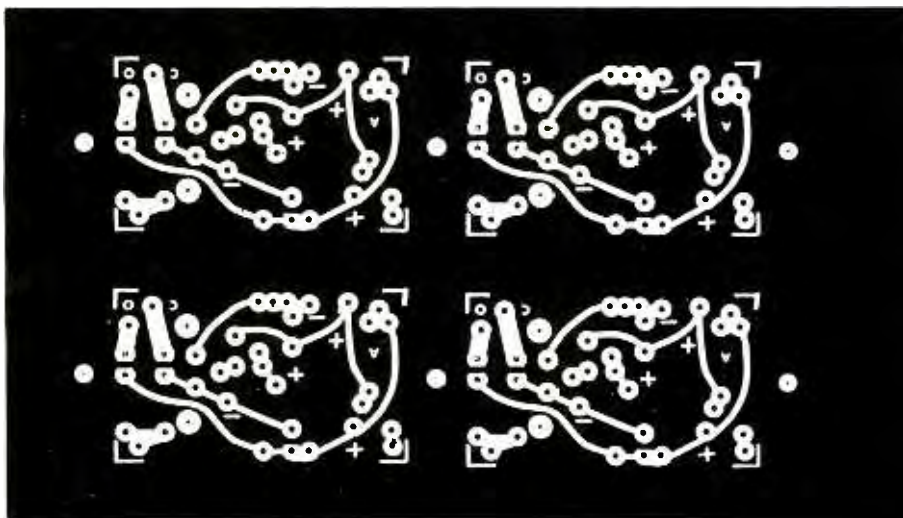


Fig. 5. The original artwork converted into a negative that will then be used to expose a presensitized pc blank. This step-and-repeat pattern permits four identical pc boards to be made simultaneously.

from the moment it's coated with resist. Otherwise, the blank will have to be stripped of resist and thoroughly scrubbed and be resensitized. Since the resist-coated blank is highly light-sensitive, sensitizing more blanks than will be used in a single day isn't recommended.

(5) When coating with resist a double-sided blank, the procedure is the same, except coat the second side after the first has thoroughly dried. Of course, if you use the alternate immersion method, the whole operation can be performed in one pass.

(6) Keep the sensitized pc blank inside its light-tight enclosure until you're ready to use it. When you are ready to use it, place the sensitized blank inside a light-tight box or envelope—such as the plastic envelopes computer boards are usually supplied in—while transporting it to your working location.

Exposing the PC Blank

Expose the presensitized pc blank to ultraviolet light in exactly the same procedure as the reversing film was exposed through the master artwork. In this case, though, the negative is used as the "master." Use the same contact print box or frame or separate plywood board and sheet of glass, as illustrated in Fig. 6. The procedure is as follows:

(1) In subdued light, place the sensitized pc blank copper side up and lay over it your exposure mask, the marked "copper side" up. Make certain you orient the exposure mask properly; if you place it upside-down on the pc blank and proceed to expose, develop and etch the blank, you'll end up with a useless board. Center the mask on the blank and place the sheet of glass over the two. Figure 6 illustrates this arrangement.

(2) Place the exposure mask and blank inside your contact print box so that the two are held firmly in intimate contact with each other. Any space between the exposure mask

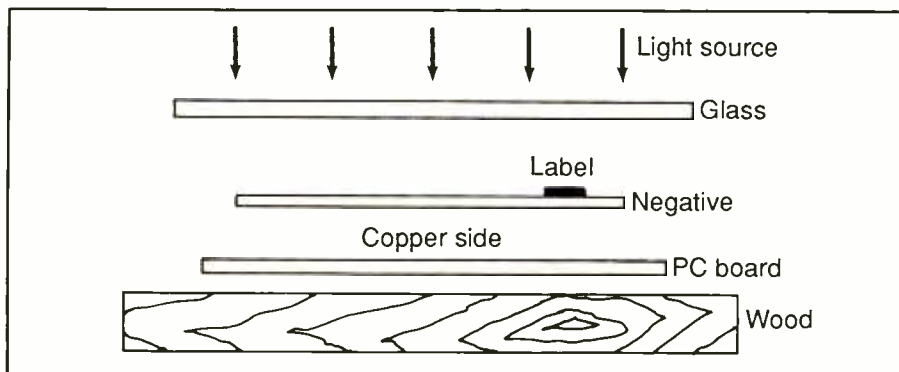


Fig. 6. Illustration of placement of the negative exposure mask and sensitized pc blank for exposure. The glass sheet holds the exposure mask in intimate contact with the pc blank.

and pc blank will result in a blurry printed-circuit pattern.

(3) Expose the pc blank through the mask to a source of ultraviolet light or the sun itself. Again, as was the case in preparing the negative from your artwork positive, exposure time will have to be determined experimentally in much the same manner as before. Typically, it will be twice as long as was required to properly expose the reversing film. Therefore, a good starting point would be 2 minutes with test exposures at 20-second intervals.

(4) Once you've determined how long an exposure is required, expose the pc blank. Meanwhile, pour the board-developing solution into a glass—not plastic!—tray that is large enough to handle the pc blank to a depth of about ¼ inch.

(5) When the pc blank's exposure period has lapsed, turn off the UV source (or shield the setup from the sun) and arrange safe lighting conditions. Remove and set aside the sheet of glass and exposure mask. Then, handling the exposed pc blank only by its edges, transfer it to the tray containing the developer solution. If the blank is single sided, place it copper side up in the solution. Allow the blank to sit in the developer for 2 minutes, at which time, the blank will no longer be sensitive to light and you

can finish working under brighter ambient lighting conditions.

(6) Handling the pc blank only by its edges, thoroughly rinse it under cold running water to wash away all traces of the developer. As you're doing this, avoid touching the copper surface because the remaining resist pattern is likely to be quite soft and relatively easy to damage. To "set" the resist pattern and dry it at the same time, it's a good idea to force dry the pc blank with a blow dryer. Otherwise, allow the blank to dry naturally by evaporation.

(7) Careful examination of the pc blank should reveal the pc pattern formed by the remaining resist. If you note any area or areas in which the pattern has washed away to excess or entirely, your exposure time was too short or you improperly presensitized the blank with photoresist. If the resist hasn't been removed sufficiently from certain areas, your exposure time was too long or the presensitized blank was exposed to light before it was placed in the contact box. In either event, you'll have to strip all resist from the blank and start over again from scratch.

During your examination, if you note any minor defects in the resist pattern, you can correct them by scraping away the resist in areas where it didn't wash off or add resist

A Dry-Transfer Alternative

There are a number of alternative ways to make printed-circuit boards. A popular method is the dry-transfer one where etch-resist is applied to a printed-circuit board, which can be done in various ways. For example, DATAK's systems are popularly used to make one-time artwork and single boards.

The company's ER-4 Photo-Etch Kit (\$37) contains all the material needed to reproduce artwork published, say, in *Modern Electronics*. It also includes the company's Direct-Etch system to make

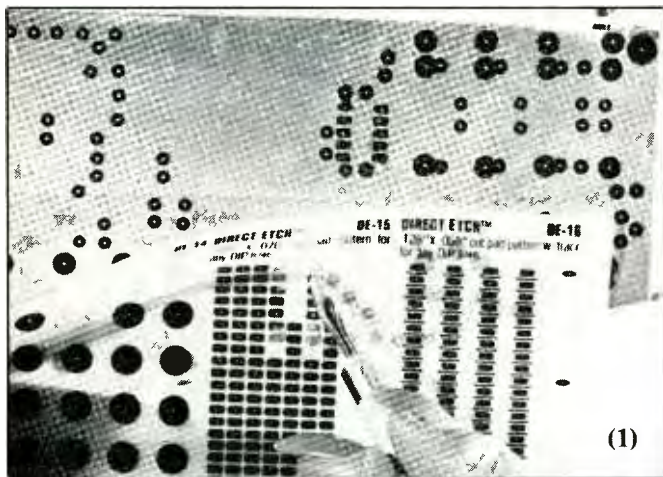
single pc circuits without artwork, which simplifies trying out different circuit layouts.

The latter system uses plastic etch-resist patterns that are transferred by pressure onto a copper-clad circuit board. A set of 69 different sheets of patterns—DIP's, TO's, surface-mount, edge card contacts, straight-line connections, polarity symbols, etc.—is available as DE-973 for \$34.95, while refill sets of two sheets cost \$2.

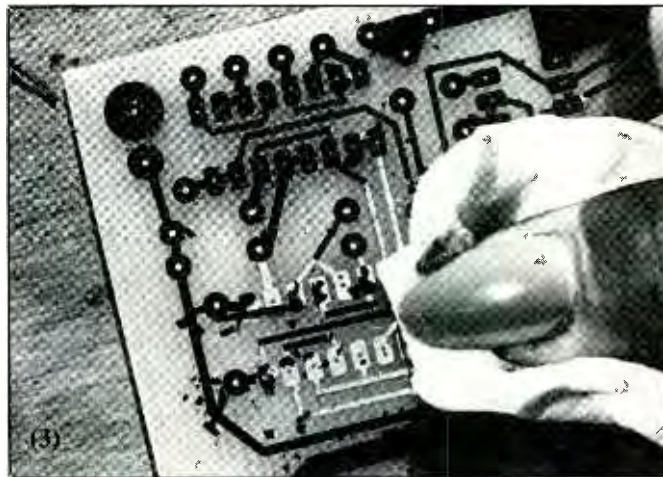
To create a circuit on a copper blank,

one simply lays down a selected pattern on the sheet and rubs it with a ball point or spoon burnisher. Then connecting traces from the sheet are cut to the desired length with a razor or knife and similarly transferred to the board. The completed circuit is then sprayed or tank etched, followed by removing the resist by soaking the board in mineral spirits and rubbing clean with a soft cloth.

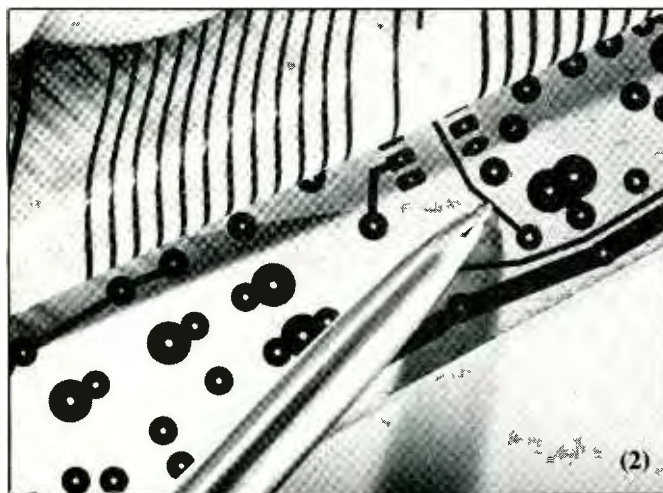
As you can see, there are many ways to skin the cat.



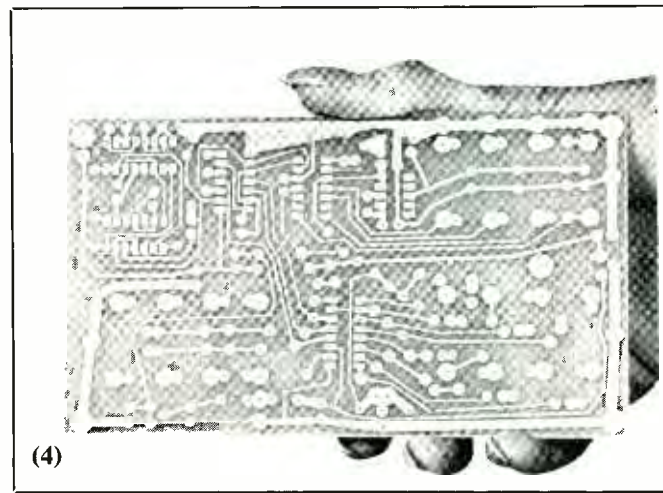
A Direct-Etch Dual-In-Line-Pin (DIP) pattern is transferred to a copper-clad pc board by laying the pattern down where it's to be positioned and rubbing it with a burnishing tool.



After etching, the dry transfer etch resist is easily removed by rubbing it with a cloth that's impregnated with mineral spirits or rubber-cement thinner.



Printed-circuit connecting traces are first cut to length and then transferred between pattern points, as shown.



The finished printed-circuit board, displaying clean edge definition, is pictured here.

with an artist's paintbrush where it washed off but should not have. If you add resist, you must expose it to UV light to set it prior to etching the pc blank.

(8) Whatever developer remains in the tray can be reused. Pour it back into its container and seal tightly.

If you're making a double-sided printed-circuit board, your exposure task will require a few extra steps. To begin with, of course, you must expose the pc blank twice, once for each side.

Alignment of the two exposure masks—technically known as “registration”—is critical. The simplest way to assure this is to align the two exposure masks with the sides labeled “solder side” and “component side” facing out and punch pinholes in two or more strategically selected component-hole locations through both masks (see points “A” and “B” in Fig. 3). Place “solder side” mask copper side up on a double-sided pc blank that has been previously cut to size and gently center-punch the hole locations onto the copper surface of the blank. Remove the mask and drill No. 60 holes at each marked location.

After drilling the holes, double-check alignment between the blank and two masks. Then clean and sensitize the blank as described above. When you're ready to expose the blank, place it on a corrugated cardboard box. Orient one mask on it with its punched holes aligned with the holes in the blank and push straight pins through from the mask side in each hole location and completely into the cardboard.

Place the glass over the assembly and check for intimate contact between the mask and pc blank. If you note any lifting of the mask, due to the heads of the pins, place thin pieces of cardboard on top of the mask and weight the assembly down with the sheet of glass. (Don't forget to do all this under subdued or safe lighting conditions!) Expose the blank through the mask to UV light.

Then repeat the entire procedure for the other side of the blank and remaining exposure mask.

Etching the Blank

Etching is a chemical process by which the portions of the pc blank not protected by the resist pattern are attacked and eaten away by a chemical solution. In effect, the unwanted copper is washed away, leaving behind only enough copper to make up the desired copper-trace pattern for the printed-circuit board.

Two etching agents are popularly used by both industry and home experimenters and hobbyists. These are ferric chloride and ammonium persulfate. Of the two, ferric chloride is much the faster etchant, requiring about 10 to 30 minutes, depending on solution temperature and agitation activity, to eat away the unwanted copper from a typical size pc blank. Ammonium persulfate may take up to 2 hours to do the same job.

Ammonium persulfate is a graceful etchant compared to ferric chloride. Both chemicals are poisonous and, therefore, must be kept out of the reach of children. They must never come in contact with eating utensils. This being the case, you should purchase a glass baking dish and use it exclusively for pc work. In fact, buy two or three dishes, saving each for exclusive use of only one chemical (etchant, developer, etc.).

A word of caution about ferric chloride is highly in order here. This is a particularly nasty chemical in that it will attack almost any metal and will stain anything else it comes into contact with. Be very careful not to spill or splash this chemical. Wash it immediately off anything that may accidentally come in contact with it. Also, if you pour ferric chloride into a sink, it will stain the surface.

Ferric chloride comes premixed and ready to use. Ammonium persulfate, on the other hand, is supplied in a crystal form and must be mixed with water at the time it is to be used.

Both chemicals work faster at elevated temperatures and are at their optimum at about 100 degrees Fahrenheit. You can warm ferric chloride to 110 or even 120 degrees by placing it in its unopened container in a pail of very hot hot water until it becomes hot to the touch before actually using it. With ammonium persulfate, simply use hot water in which to dissolve the crystals.

Pour the etchant into a glass baking dish to a depth of $\frac{3}{8}$ to $\frac{1}{2}$ inch. Then place the pc blank in the etchant. As the etchant begins to work, gently rock the dish back and forth to agitate the etchant and wash away the released copper. Continue etching the blank, rocking the dish every couple of minutes or so, until all copper has been etched away from the areas not protected by resist. Take care to avoid splashing or spilling the etchant.

During the etching process, periodically lift the board out of the etching solution to inspect the progress. Use only plastic tongs to lift the board from the solution. *Never* use metal tongs; the etchant will immediately begin attacking them if you do.

Don't over-etch the pc blank. If you do, you may find that the etchant will begin to attack the desired copper pattern. When etching is done, remove the board from the glass baking dish and thoroughly rinse it under running tap water. Then carefully examine the board to verify that it has been properly etched. If there are still traces of unwanted copper, return the board to the etching solution until they've been dissolved and once again thoroughly clean the board.

Once the etching process is truly complete and all traces of the etchant have been rinsed from the board, remove the resist that remains on the copper pattern to make it possible to solder to the copper pads. Use fine steel wool or very fine emery cloth to remove the resist.

If you use ammonium persulfate etchant, be advised that it is not reus-

able. Upon completion of etching a board, this chemical should be disposed of according to the rules for such disposal in your community.

You'll know when ferric-chloride etchant has become saturated with dissolved copper by the fact that its copper-removing action will be noticeably slower or cease altogether. When this occurs, it's time to discard the etchant. The best way to do this is to pour the etchant into a container of dry sand, which will soak it up like a sponge, and then dispose of it with your normal trash pick-up. Before doing the latter, however, check with your local environment protection agency if this method of disposal is legal; if not, ask what must be done to get rid of the etchant.

At this point, you're ready to finalize your new pc board. If the board is to take an odd shape—such as an "L" or a triangle—or must have cut-outs or notches, do whatever cutting is required *before* drilling any holes. Then drill all holes, starting with the

Materials Availability

The printed-circuit artwork and board materials mentioned in this article, as well as others that are not mentioned, are available from a number of sources. However, to make it more convenient for you to obtain small quantities of the supplies needed to make professional-quality pc boards, the following items are available from A. Caristi, 69 White Pond Rd., Waldwick, NJ 07463:

- An artwork kit consisting of an 8½ × 11-inch sheet of drafting film, roll of 0.030-inch-wide tape; 192 0.100-inch-diameter donut pads; and 10 DIP IC pad patterns, \$16.95.
- 3M negative film (5 × inches), \$7.95, and 3M film developer (2 ounces), \$3.95.
- Paint-on-type photoresist (1 ounce), \$6.95, and developer (4 ounces), \$4.95.
- Ammonium persulfate crystals to make 1 quart, \$4.95.

Add \$5 per order to cover P&H. New Jersey residents, add state sales tax.

largest and working down to the smallest. If the centers of any donut or IC pads have no copper removed from them, carefully dimple them using a sharply pointed center punch and a gentle tap from a hammer.

Your printed-circuit board is now ready to be populated with components. If you followed the routines outlined here, the board—and, by

extension, the wired circuit-board assembly—should look every bit as professional as any you can get from commercial sources.

Once you master the “art” of fabricating pc boards, you’ll discover a whole new dimension of the electronics hobby. Given professional-looking pc assemblies, you’ll undoubtedly want to show off your projects. **ME**
