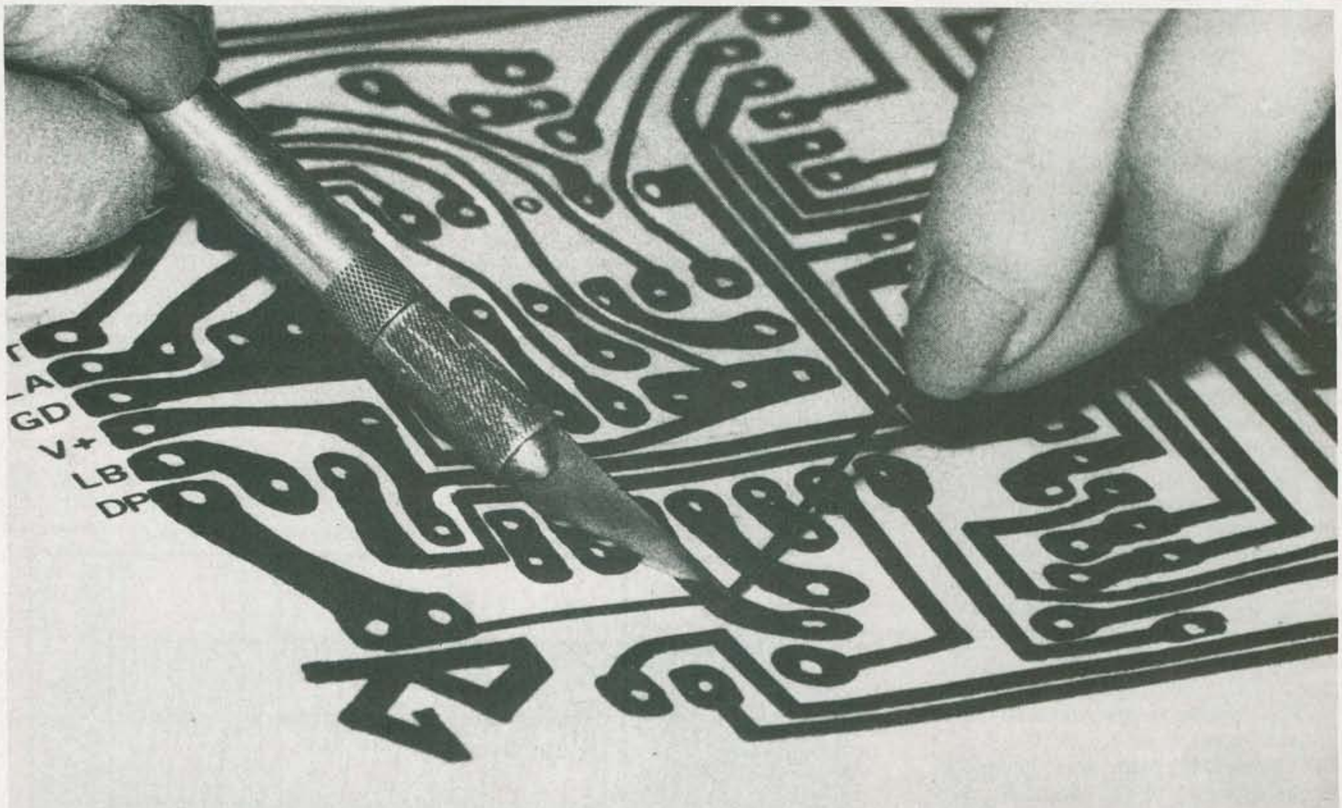

ETCH YOUR OWN PC BOARDS

There are lots of ways to make printed-circuit boards, but if the pattern is complicated, nothing beats doing it photographically.

ROBERT GROSSBLATT

ELECTRONICS HOBBYISTS HAVE NEVER had it as good as they do today. The modern home workshop can now be equipped with laboratory-grade instruments at consumer prices and can produce projects of industrial quality. It's rather astounding to consider that the decision whether to build or buy is more a function of convenience than complexity. Ever since the IC manufacturers discovered the profits to be made in the consumer market, the components needed for virtually any project are rarely more than a postage stamp away.

So why haven't we seen a tremendous increase in the number of sophisticated projects built and developed on a backroom bench? Obviously there's been a problem, but the biggest stumbling block to home construction doesn't have anything to do with design or complexity—nine times out of ten it's the printed-circuit board. A hobbyist who wouldn't think twice about designing a circuit for bouncing signals off Neptune often cringes when he considers producing printed-circuit boards. While electronics technology gets more sophisticated almost by the minute, the process of making PC boards has remained just about the same for the last thirty years. And the reason why it hasn't changed is that the process is really quite simple.

No kidding, it's really easy to do. There's been a big push among manufacturers to develop wire wrapping as some sort of technology all on its own. If you're into wirewrapping, don't bother reading any farther because all I think wire wrapping is good for is baling hay. But making even the most complicated double-sided board isn't difficult—it takes time and requires attention to detail, but it's not hard. Let's take a look at the basic procedure.

Copper-clad board is coated with a light-sensitive chemical and allowed to dry. A mask is made photographically (the foil pattern) and contact-printed on the board. The light in the printing process causes a change in the chemical coating so that the unmasked areas become resistant to the developer. What happens then when the board is placed in the developer is that the chemical over the unmasked areas (the traces) remains, while the chemical over the rest of the board is removed. The result is a board ready to be dumped into the etchant. That is the basic procedure for so-called negative photoresist; there's also a positive resist but the idea is the same. As you can see, there are several steps in the process and we'll go over them completely later on in this article. By the time we finish, however, you'll be wondering why you thought it was difficult in the first place.

Layout

Before we explore the mysteries of board making, there's something that comes first—laying out the pattern. By the time you're ready to consider making PC boards you've probably pulled out a lot of hair in breadboarding. You've rearranged circuitry and substituted different values to make sure everything is working exactly the way you want it. When you reach that point—when you decide that you're looking at the final design—you're ready to move on.

The first step is to draw a schematic of your breadboarded circuit. The chances are that you've already done that, but if you're anything like me the drawings are a mess. When I'm at that point, I usually have several pages filled with false starts and things that blew up. That's to say nothing about the most important sections, which I usually scribble on the back of an envelope. It's really important to stop and redraw everything. Make a clean drawing of the circuit, lay it out in a coherent fashion, assign numbers to the components, and carefully label everything on the page. Also, make sure you include the pin numbers of the IC's you are using. Use graph paper to help keep the lines straight and do as many versions as you need in order to make the drawing neat. Keep on checking your drawing against the breadboard as mistakes are easy to correct at this stage—it's much simpler to erase a line than to correct a trace. Make a couple of copies of the finished drawing in case you spill coffee on the original.

A completed drawing is a valuable tool because it will give you your first overview of your entire circuit. Don't be surpried if you wind up combining things or simplifying the design. Unused gates can be combined to eliminate whole IC's and a little reorganization can allow you to simplify complicated sections of your design. I usually wind up with several stages of "final" drawings until I get the point where any further changes are profitless. When you've reached that stage you are ready to begin the layout of the board.

At this point you have the schematic and the breadboarded circuit. **Whatever you do, don't fool around with the breadboarded circuit!** Not only will you still be checking the drawing against it, but later on, when the printed-circuit board is complete but perhaps doesn't work, you'll have something to compare it to. Remember an all-important rule of original design: the breadboarded circuit is the only one in the world that does what you designed it to do. The last step in creating a working circuit on a PC board, *the very last step*, is to pull the breadboard apart.

Special considerations

Examine the breadboard and the drawing. Are there sections of your design that have to receive special treatment? If certain sections need shielding, or have unusual power requirements, they should be physically close together on the board. Should the circuit be broken into several boards to conserve space? Is the printed-circuit board going to provide structural strength to the finished project? Does the board have to have a particular shape because of the case? Those are the sorts of questions you have to consider when you begin to do the layout. The answers will determine the materials you use, the size and number of the boards, how compact the layout has to be, and so on.

Before you start planning the printed-circuit pattern make sure that you have all the components that go on the board. Resistors, capacitors, and other components come in different shapes and sizes—and you're doing a layout for components of a particular size. If you plan on making three boards, make sure you already have three of everything you'll need. There are few things sadder than going through the trouble to make a complicated board and then finding out that certain small electrolytics aren't available any longer. It's always best to stick with standard, readily available parts.

The best way to work out your layout is to do it on graph paper, but the choice of paper is important. What you want is a light-blue grid measuring ten squares to the inch. Get a large pad of at least 20 by 24 inches and tell the people in the art supply store that you want it to be non-reproducible blue. That is a particular shade of blue that won't register on the film you'll use to photograph the final drawings. The pencils you use for the initial drawing should be the same shade of blue and can also be obtained at an art supply store. The reason you want such a large pad is that you'll be doing the drawing double size and reducing it later on. Leave yourself lots of room to work and make sure that the paper you use is at least four times larger than the intended size of your board.

Drafting

Draw a horizontal line across the middle of the graph paper. You should be able to fit a double-size outline of your board's

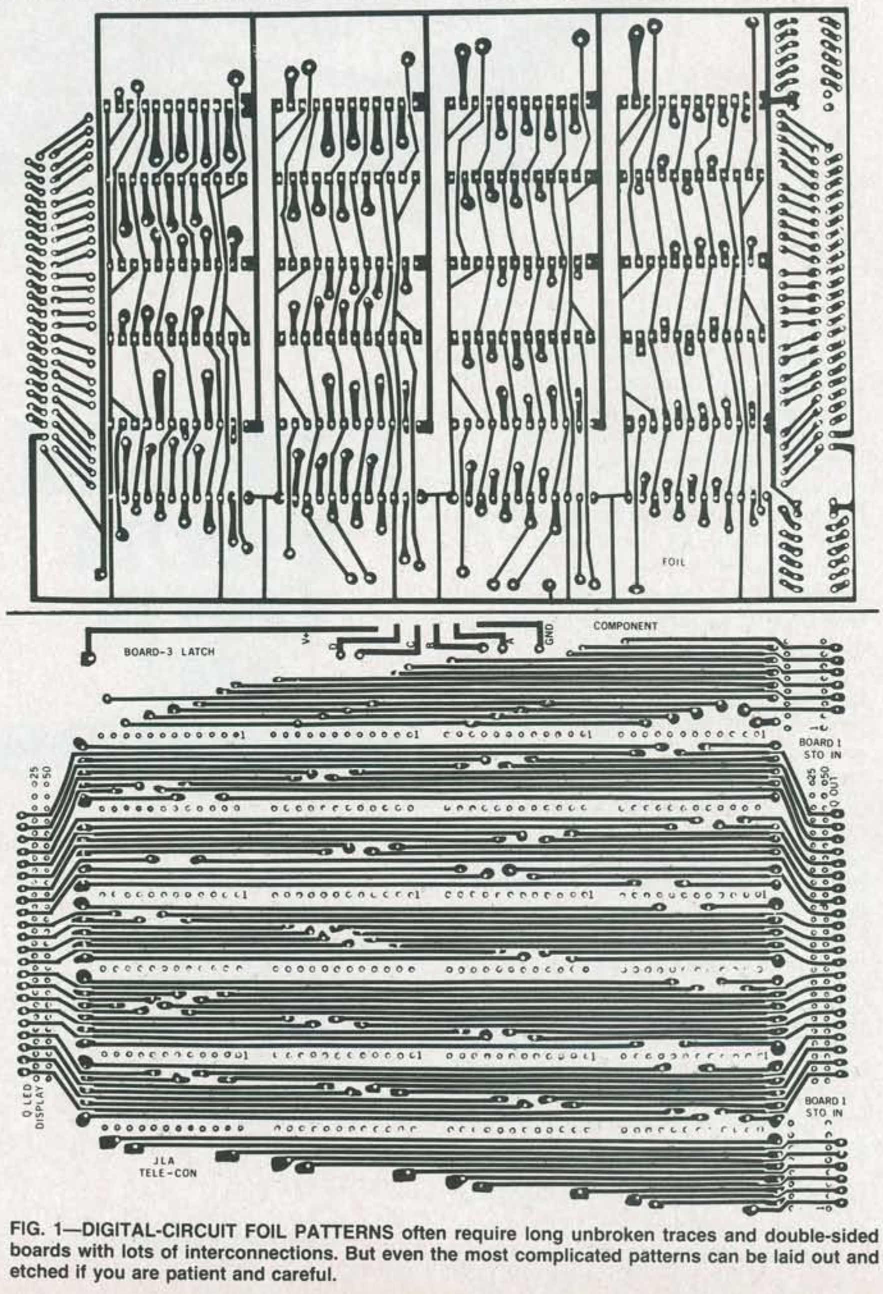

FIG. 1—DIGITAL-CIRCUIT FOIL PATTERNS often require long unbroken traces and double-sided boards with lots of interconnections. But even the most complicated patterns can be laid out and etched if you are patient and careful.

shape on both the lower and upper halves of the paper. If you can't, get larger paper. The reason for dividing it in half is that you'll be doing the foil pattern and the component layout at the same time. Not only that, but you'll be able to tell whether your board has to be single or double sided. Draw the outlines of the board in black using a fine-point magic marker. All the rest of your preliminary work will be done with blue pencils. This is the time to draw in the board extensions for edge connectors, or any other odd shape you need. Anything that has to be in a particular place on the board should be done in black. These are the "givens" around which the rest of the components have to fit. They are usually due to particular case requirements.

There are lots of schemes for doing a layout. Probably the best way to start is to draw the pins for the IC's. A good way to start the layout is to look at how the IC's were placed on your breadboard. Remember that you're working twice actual size so adjoining pins should be two squares apart. Since the physical dimensions of IC's are designed around 0.1-inch spacing, you'll find that everything fits neatly onto the paper. Place the IC's in neat rows and columns. That will make things easier when you start drawing in the traces and will make redundant connections obvious. Common connections such as power, ground, clocks, and so on will be much easier to lay in if the IC's are logically spread out on the paper. Make sure to indicate the IC pins on both halves of the graph paper and label pin number 1 on both halves to help keep everything straight in your mind.

Remember that you're not reversing anything—the bottom and top halves of the paper are being drawn as if you had laid both sides out flat and were looking down at them. Some layout schemes call for drawing the two sides of the board on top of each other in different colored inks. Not only does that cause extra steps later on, but left and right get confused and it's really easy to make mistakes.

I wish I could give you some foolproof way of routing the leads but I don't know any other than trial and error. Be prepared to do a lot of erasing and redrawing because it takes a while to solve the topological problems caused by your circuit—there's no teacher other than experience. When you find that you absolutely, positively have to cross traces, break one of them and draw doughnuts at the ends. Locate the **exact same** positions on the other side of the board, draw two more doughnuts there, and draw a trace connecting them.

As you draw the connections for your circuit and put in the components, try to avoid using the component side of the board because single-sided boards are much easier to make. Unfortunately, however, digital circuitry often requires a double-sided board. If you accept that in the beginning, you can save a lot of grief by first laying out the address and data lines. The chances are that those traces will require a number of connections between both sides of the board. It's not at all unusual to have a trace jump back and forth from one side of the board to the other eight or ten times—if you can make the connection that way, do it.

The complexity of the foil pattern you generate will depend on a number of things. If the component density is high, the foil pattern will be busy. Obviously, the more connections you have to make, the more complicated the foil pattern will be. Interestingly enough, if your circuit has lots of passive components, you'll have an easier time figuring out a foil pattern. You'll wind up with lots of short traces and you'll be able to route the longer power and ground connections around them. Digital circuitry is tough because lots of direct connections have to be made between the IC's. That requires long, unbroken traces and, unfortunately, lots of connections between both sides of the board (see Fig. 1).

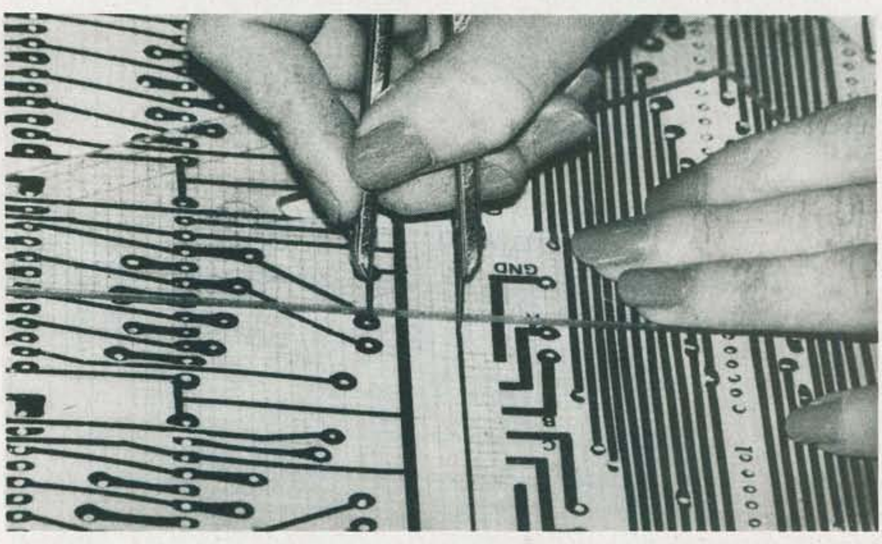

FIG. 2—WHEN LAYING OUT a double-sided pattern, one of the most important steps is to make sure that both sides line up properly.

Double-sided boards

Sometimes you can simplify things by rearranging the IC's but if you're dealing with power, ground, clock, address, and data lines, there's no way you're going to be able to get them all on a single-sided board. Don't throw in the towel, however, as double-sided boards *aren't* that much harder to make. Piggybacking components and using loads of jumpers is a poor alternative to making a double-sided board.

You have to be really careful when you're working back and forth on the two sides of the drawing—make sure that the holes and traces line up properly. Use a straightedge and calipers, as shown in Fig. 2, and don't feel silly about counting squares—I've made a lot of double-sided boards and I do it all the time.

Once you've decided to make your board double sided, take advantage of the decision. There's as much copper on the component side of the board as there is on the foil side. It's rather strange to jam as many traces as you can on the foil side and leave only a few on the component side. Not only that, but the more evenly you distribute your traces, the more likely it is that the board will etch evenly. If you have 90% of your pattern on the foil side and just a few traces on the component side, you're running a serious risk of undercutting the traces on the foil side during etching. What's more, having lots of traces on the foil side means that some of them are going to be thin. While you're waiting for a veritable sea of copper to be etched on the component side, the thin traces on the foil side will disappear.

If you plan things carefully, the connection from one side of the board to the other can be made using the leads of the components. It's a simple matter to solder the leads on both sides of the board. For those traces that wander from one side of the board to the other, you'll just have to leave doughnuts and connect them with small pieces of hookup wire soldered to both sides.

The big problem comes when you have to deal with IC's. There are a few options: For one thing, you can solder the IC's directly to the board and make the feedthrough connections by soldering the IC legs on both sides—but **don't do it—** always use IC sockets. You can mount the socket so that it is raised about an eighth of an inch or so above the board and solder the the socket legs on both sides of the board; of course, you'll need wire-wrap sockets to do that. I don't like that method because wire-wrap sockets are more expensive and physically larger than others. Also, having the socket sit above the board leaves the plastic part of the socket free to move and puts an undue strain on the legs. All things considered,

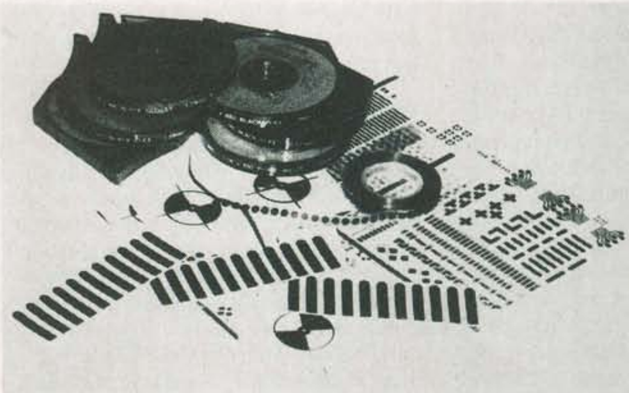

FIG. 3—DRAFTING AIDS, such as the ones shown here, can be very useful in completing your layout.

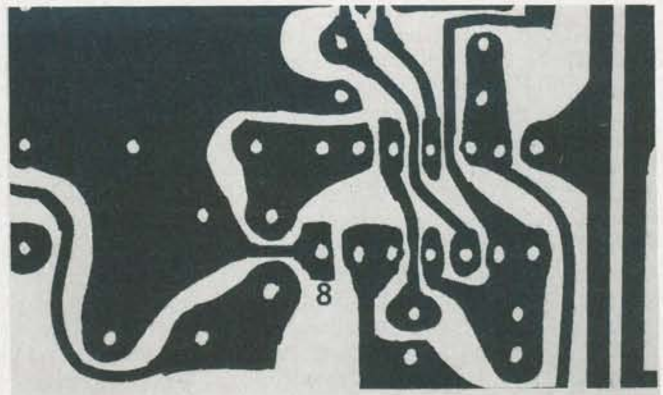

FIG. 4—IF TRACES MUST BE ROUTED between IC pads, elongate the pads to make soldering the components easier later on.

if your circuit is complex enough to require a double-sided board, you want to eliminate any potential problems, and mounting sockets above the board is a possible source of trouble.

What I do is to extend the trace on the component side and use small doughnuts on both sides of the board to make the connection from one side to the other. It's neater, and makes the board more secure and compact. After the board is etched, I thread hook-up wire through all the feedthroughs and then solder and snip them in one shot. Any other method leaves room for error and if you have a board with lots of connections, it's much too easy to forget the IC legs hidden under the sockets and then go crazy trying to figure out why the circuit won't work.

Further considerations

Whenever you add a component to your drawing, show the doughnuts on both sides of the board. Not only does that make the geography of the board easier to understand, but it will be the basis for a parts-placement diagram later on. When you have your drawing completed, draw the components in (remember you're working twice actual size) and make sure that there's enough room to fit them on the board. Count up the number of traces on the component side to see whether a double-sided board is justified. If there aren't too many, consider the idea of jumpers, instead. If you feel, though, that a double-sided board is necessary, make sure that you leave at least two squares between the traces and any component leads. That is particularly important around the IC's because you don't want the IC sockets to hide the traces on the component side. It's almost impossible to locate a small break in a trace if it's hidden under a socket, and it's even harder to correct it.

Once you have the drawing finished and have checked the hole alignment, make sure that the connections agree with the schematic. Take your time doing that because corrections are a lot easier to make at this stage than they will be later on.

Final artwork preparation

After everything has been verified, the next step is to prepare the board for photography. All that means is that you have to ink in all the doughnuts and traces you've done in blue pencil. Many of the commercially available drafting aids can come in handy here (see Fig. 3). Long runs can be made using black drafting tape and several companies sell IC pads and doughnuts in double-size patterns. When working double size, try to keep the traces at least 1/16-inch wide and separated by at least 1/32-inch. That's the absolute minimum, though—tiny traces have a nasty habit of disappearing in the etchant and small separations may not show up in the final foil pattern.

The only exception is when you find it necessary to route leads through the pins of IC's (which you should avoid like the plague—especially if you're doing a double-sided board). If you find that you absolutely have to route traces between IC pads, make the pads, rather than the trace, thinner, and elongate the pads parallel to the traces as shown in Fig. 4 to be sure you'll have enough copper to solder the pins to.

When you've finished that part of the layout make sure there aren't any breaks in the pattern. If you find any, color them in with a fine-point felt-tipped pen as shown in Fig. 5. Check the pattern carefully and if any of the traces or corners look thin, use the pen to fatten them. If you make a mistake, cover it over with typewriter correction-fluid and redraw it.

Make sure your lines are as black as possible and are completely filled in—every bit of black you add now will become copper on the board. Also, try to anticipate any special problems. Will you be soldering any nuts on the board to mount a meter or power transistor? If so, blacken in an area large enough to provide copper for the nut. There are no hard and fast rules to tell you how much copper to leave on the board and everything depends on your application, and your patience. Just remember that the more copper you keep, the less you have to etch. Once I've gotten to this stage, I thicken every trace I can to make the etching process as simple as possible. It never hurts to have spare copper on the board and wide power and ground planes are never a bad idea.

There is one place where the doughnuts should be as large as you can possibly make them. That is wherever you have feedthroughs going from one side of the board to the other. The larger the doughnuts, the more tolerance you're going to have in keeping the top and bottom of the board in register later on.

When we continue this article, we'll put the finishing touches on the layout. We'll also show you how to transfer the layout to the copper board. **R-E**

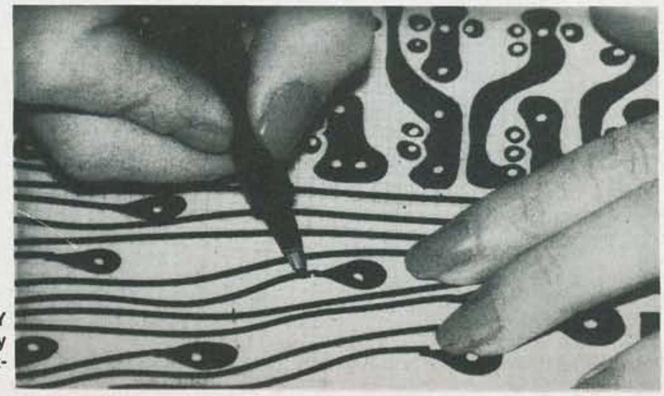

FIG. 5—REPAIR ANY BREAKS in the layout by filling them in with a felt-tipped pen.