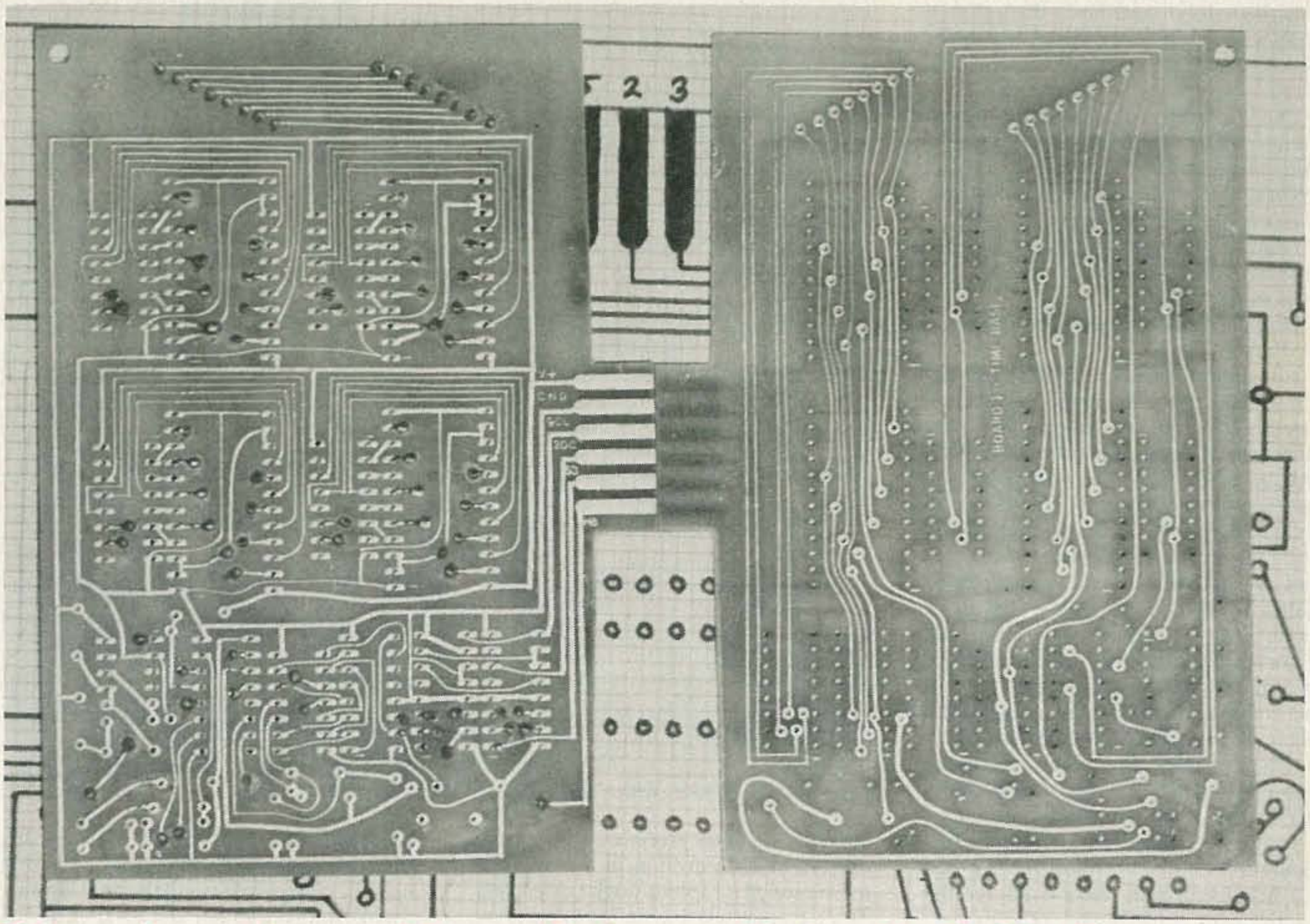

Designing Double-sided PC Boards

Making a double-sided PC board is child's play when designing one. Here are a few hints to ease the design process.

ROBERT GROSSBLATT

Part 2 LAST MONTH WE WERE outlining methods of routing traces on the PC board. Let's finish that discussion and go on to the actual etching process.

We've discovered a few general rules that should be kept in mind when routing traces.

1. You can route a trace through IC pins on the foil side, but doing it on the component side is asking for trouble.
2. Power and ground connections should be made in each section separately. When the whole circuit has been laid out, the individual sections can be joined.
3. Keep all traces except those going between IC pads, at least one tenth of an inch (two divisions on the graph paper) away from the IC pads. That leaves room for solder pads
4. Bus lines that have to be routed to many IC's on the board should be handled as in number 2 above.

If you look at a commercially made board, you'll see that traces are routed through IC pins on both sides of the board. Now, you can lay out your board that way; it's easy to do things like that on paper. But when the time comes to actually make the board, you're going to regret it. If your registration or drilling is even the slightest bit off, you're going to be swamped with shorts and broken traces. And if you use IC sockets, (and you should!), the traces most likely to be causing the problem will be hidden under the wall of the socket.

After you get the IC's drawn in, connect the power, ground, and whatever other pins have to be joined. Once you've done that, start drawing in the other components that are part of this section of the circuit. Don't forget to draw the pads and outlines of the components on the top side of the board. Put in the traces you need to connect everything together, making sure you leave an escape path for the traces that have to go somewhere else on the board.

Getting traces from one side of the board to the other is a major problem with home-made double-sided boards. Commercially made boards use plated-through holes, but we've never found a reliable way to make plated-through holes at home. The best way is to "fake it" by using a component leg as a feedthrough. By drawing solder pads in register on both sides of the board, you can easily get a trace to the other side of the board. The real problem comes with traces that connect only to IC's.

There are a couple of options to handle that type of situation. Which one you choose depends on your circuit, your patience, and your concern with the final appearance of the board.

- Drill slightly larger holes that can fit a small wire as well as the IC leg and solder the wire to the pads on both sides of the board.
- Use wirewrap sockets and leave them sitting a quarter of an inch or so above the component side of the board so you can solder to the pad on the component side.
- Use Molex pins instead of IC sockets.
- Draw in solder pads for small pieces of wire used only for connecting the two sides of the board.
- Don't use sockets and solder directly to the IC legs.

Not all those choices are practical. Not using sockets is ridiculuous and drilling larger holes can be dangerous. The other three options depend on how you want the final board to look, and what plans you have for producing the board. If you're going to have the board commercially made, using wire-wrap sockets or Molex pins is good because the holes can be plated through by the people that make the board. If your board will only be made at home, you can draw in solder pads for through-the-board jumpers. The design of the board is slightly more complicated when you use that method, and assembly is a bit more work. But the advantages are that it can be done at home and the holes can be plated through later on.

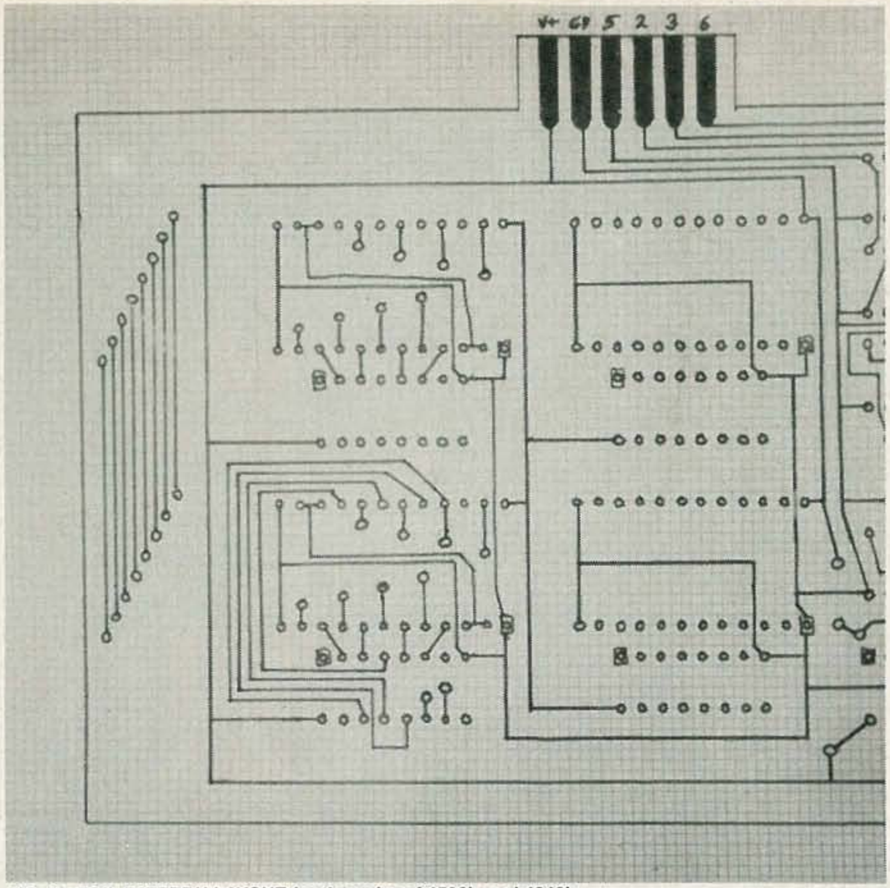

FIG. 6—FOIL PATTERN LAYOUT for the pairs of 4508's and 4040's.

Figure 5 shows the layout with the first section of the circuit drawn in. You can see that extra pads were used for the feedthroughs needed to get some connections to other parts of the circuit. As you're doing the layout for your own board, it's a good practice to always end the I/O of each section in a place with room enough for feedthrough solder pads. That's true even if you decide to use some other method of jumping them to the other side of the board. No matter what kind of PC design you're doing, always keep in mind that lost options are potential problems. In other words, until the final layout is done, don't eliminate any possibilities.

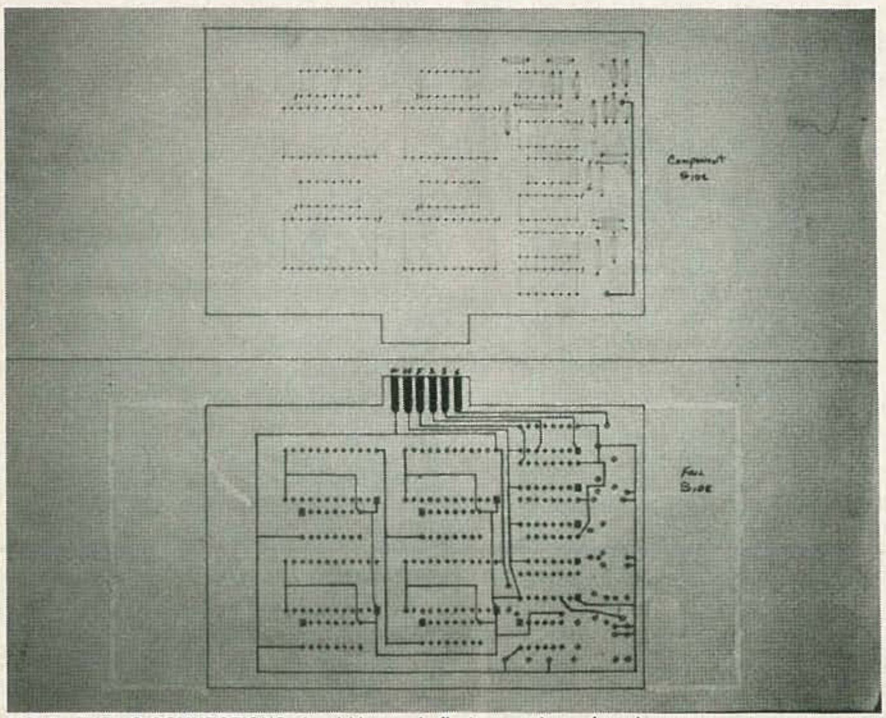

FIG. 5—THE I/O CONNECTIONS should be made first, as we have done here.

As you can see from the block diagram and schematic of our circuit, there is one large section made up of four identical pairs of IC's. (Each pair is made up of a 4508 and a 4040.) Since each of those pairs is connected in the same way, doing the layout for one means we've done the layout for all of them. In Fig. 6, the PC pattern for those parts has been drawn in. Most of the I/O has been brought over to the component side because the foil side got jammed up with the traces connecting the IC's together. Now that we're talking about traces on the component side, we should tell you how to decide which traces should be on which side of the board.

- Put traces on the component side only when there's absolutely no room on the foil side.
- Horizontal runs should be on one side of the board and vertical runs should be on the other.

If you pay careful attention to the first "rule," you may be lucky enough to find that you can add one or two jumpers and reduce the whole layout to a single-sided board. The importance of the second "rule" becomes evident when you get further into the layout. As you start laying out the traces on the foil side that are needed to connect the components together, there

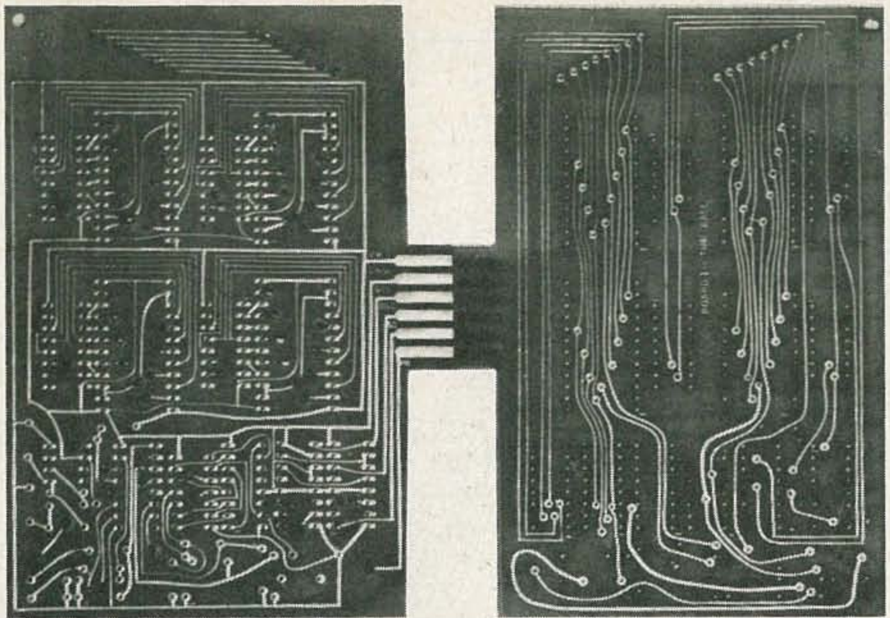

FIG. 7.—THE FINISHED LAYOUT is shown here on an etched board. Note how the traces run between the top and bottom of the board.

will be fewer open paths either vertically or horizontally, depending on your layout. Reserving a board side for each direction makes everything a lot more orderly.

You can see that extra board space was needed just for the I/O traces. That's why you should make the layout for each section as compact as possible. It's impossible to tell at the outset how much extra board space is going to be needed by the layout itself. If you have no restrictions on the size of the board, that's not much of a problem. But if the board has to fit somewhere in particular, you have to be very stingy in allocating board space as your design develops.

The actual procedure for running a continuous trace on both sides of the board is surprisingly easy. Since we 'unfolded' the board, not only are both sides laid out in front of us but both of the layouts are correct left to right. Some other methods of doing double-sided layouts use tracing paper and light boxes and wind up giving you a reversed image of one side of the board. Double-sided layouts are complicated enough without having to work with backwards.

When you want to move a trace to the other side of the board, end the trace with a solder pad and then find the corresponding point on the other side of the board using the graph-paper lines as a guide. Use the dividers to transfer the distance from the baseline, checking it several times to make sure the dividers haven't slipped. Try to keep all the solder pads on the intersections of the graph paper lines; it's easier to keep from making mistakes that way.

As your layout gets near completion, it's not unusual for a trace to make several trips between both sides of the board. Just as traces fill up the foil side, the components themselves will fill up the other side. And since you want to eliminate as many problems as possible, try to keep the traces on the component side of the board as far as you can from the components. That's particularly important with IC's, since the board space next to the holes is going to be permanently covered by the IC sockets.

Figure 7 is the completed board layout. It may seem to be complicated but that's only because it's our layout and not yours. Just as it is with schematics, the circuit is self evident to the person that drew it. By keeping everything in separate sections and building the final layout a section at a time, you can keep all of it straight in your mind. Of course it's always nice to put helpful labels, such as pin and part numbers, on the board. It can't hurt and it could save a lot of head scratching later on.

In Fig. 7, you can see that the power and ground connections have been made and brought out to two of the card-edge fingers. The thing to notice here is that the routing for these traces isn't a straightforward one. They wander over both sides of the board. If we had been connecting each section up as we went along, chances are the layout would have been a lot different looking. Not better, not worse, but only different.

After you've checked the connections against your paperwork and checked the paperwork against the breadboard, the time has come to use drafting aids to blacken the pads, doughnuts, and traces you've drawn in with the blue pencil. Keep things nice and neat and don't worry about the blue lines: standard lithographic film won't see them—all it knows about is black and white. Don't use drafting tape that's less than one sixteenth of an inch wide because it's hard to get a successful etch with a line that's thinner than one thirty second of an inch thick. The traces themselves should be at least 0.2 inch apart for the same reason. If you're working at greater than double size, don't forget to multiply those dimensions accordingly.

Once things are all blacked in, put registration marks on the corners and the artwork is ready to take to your local lithographer. He can reduce your layout to the right size and make the necessary negatives or positives for you on lithographic film.

Now we get to the easiest part of the entire task, etching the printed-circuit board itself. Although that is a completely different topic from the one we've been dealing with in this article, we've assumed that you are familiar with the technique and have not covered it here. If that is not the case, you can learn about the etching process in "Etch Your Own PC Boards, a two part article that appeared in the January and February 1983 issues of **Radio-Electronics.**

Details, details!

One thing that hasn't been mentioned so far is that the secret ingredient to something as complex as a double-sided PC layout is an overwhelming attention to detail.

As your layout starts to unfold on paper, it gets easier and easier to make a mistake. An incorrectly transferred measurement, a wrong trace, IC's drawn in backward, etc. can be disastrous. *Be careful, and keep good notes!* By the time you finish

ONCE THE LAYOUT IS FINISHED, it's time to etch the board using chemicals and aids like the ones shown here.

the layout for the last section on the board, it's easy to forget what goes where in the first one.

Laying out a double-sided PC layout is much easier than you think it is. But like a lot of other things, the only way to learn how to do it is to do it. Start with a simple one, preferably a circuit for which you've already done a single-sided layout. The double-sided layout will make the board smaller and more compact. (The layout we've been using as an example stuffed 14 IC's and a bunch of other components onto a 3×5-inch board.) And though compactness might not be your goal, remember that you can always make a board bigger, but it's sometimes hard to to make it smaller. **R-E**