

## HOW TO DESIGN PC BOARDS FROM A SCHEMATIC

Simple, inexpensive  
procedure  
takes much of the  
time and effort  
out of  
making pc boards.

BY JAMES BARBARELLO

**M**ANY GOOD electronic project designs never get built because the usual methods of making printed-circuit boards are just too laborious for one-time projects. In this article, we will describe a design-and-fabrication system that can take much of the time and effort out of making pc boards. Using the procedure outlined here, you can, with a few simple and inexpensive tools, lay out and fabricate a pc board in about the same time it would take you to build a circuit using point-to-point wiring.

**Initial Phase.** Once you have completed your circuit design, breadboard the circuit and thoroughly check out its operation. Only when you are satisfied with the circuit's operation can you draw up a full working schematic diagram.

To illustrate the etching-and-drilling

guide layout procedure, we will use the sample schematic diagram shown in Fig. 1. (This circuit is a different configuration of the Snare Accessory featured in the September 1977 "Cabonga" article; see "How It Works" box for description of circuit operation.) Tape a sheet of tracing paper over your schematic diagram or make a photocopy of the diagram. Then start your layout by drawing the IC's as they physically appear with their leads pointing toward you. While you can work directly on graph paper, sizing and spacing your layout for the specific components you will be using, you can also use plain blank paper.

The next step is to draw in the other components near the IC's to which they will be connected. As you draw in each component and complete each interconnection, keep tabs of your progress on

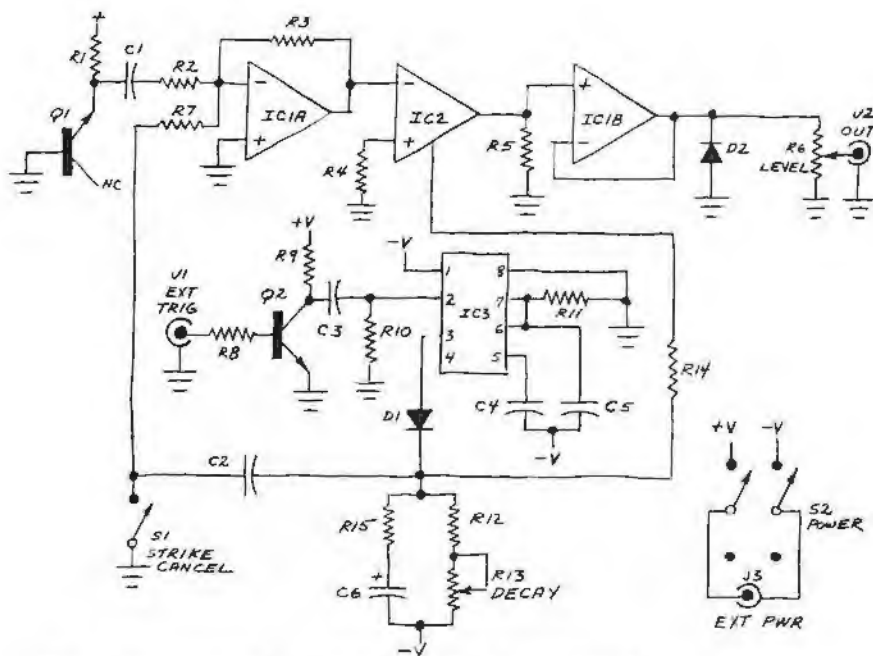


Fig. 1. Example of circuit for which pc board is to be made. See "How It Works" on next page for circuit details.

the schematic's tracing paper or the photocopy. Bear in mind that for every connection you must enter a dot. For example, pins 6 and 7 of IC3 and the top of R11 in Fig. 1 must each have a dot indicated (see Fig. 2).

If connecting lines cannot be made without crossing previously drawn lines, try to rearrange the components and/or reroute the lines to eliminate the crossovers. However, if repeated attempts fail to eliminate crossovers, you must use jumpers. You do this by indicating two dots and a line labeled "J" for each jumper you must use. Do not hesitate to use jumpers where necessary; a medium-sized pc board with 10 to 20 jumpers is not uncommon.

While lines between components cannot be interconnected without using

jumpers, connecting lines can pass between the dots in the areas that will be occupied by resistors, capacitors, diodes, etc., since the component and the connecting line will be on opposite sides of the board. A good example of this is the -V line that passes between the pads for R4, R5, and D1 in Fig. 2.

The connections between the pc board and external components J1, S1, R13, and R6 are shown originating from dots on the pc guide and terminating at the external components via curved lines. Note, too, that all components on the guide are identified, either by symbol number (IC1, R2, D1, etc.) or by component value (10  $\mu$ F, 100K, etc.).

In most cases, you will discover that the physical layout of the etching-and-drilling guide closely resembles the lay-

## HOW IT WORKS

Transistor Q1 serves as a noise source that generates a constant approximation of a white-noise signal. The signal is coupled through C1 to the input of summing amplifier IC1A.

A positive-going trigger pulse applied to J1 is routed to the base of Q2 where it is inverted and coupled through C3 to pin 2 of one-shot timer IC3. The inverted pulse triggers IC3 to produce a 0.25-second positive-going output pulse that is coupled to the input of IC1A and also quickly charges C6 through R15. Capacitor C6 immediately begins to discharge at an exponential rate determined by the values of R12 and R15 and the setting of DECAY control R13. Diode D1 reverse biases as the potential across C6 decays, thus isolating the output of IC3, which would tend to maintain a full charge on C6. In this manner, a voltage envelope with a very fast attack time and an adjustable exponentially decaying release time is generated and used to control the gain of transconductance amplifier IC2.

The gain of IC2 is set by the level of the voltage envelope and the value of R14. The output of IC2 is a combination of the noise signal and the pulse from IC3 (a drum-stick-strike sound) whose voltage envelope is the control voltage described above. Resistor R5 serves as the load for IC2. Integrated circuit IC1B buffers the signal and D2 clips the negative portion of the IC1B output to produce a realistic sound effect. Potentiometer R6 provides for a variable output level.

out of the schematic diagram's components. You can check this by comparing Fig. 1 with Fig. 2.

At this point, it is best to breadboard your circuit once again, carefully following the layout just drawn. Power the circuit up and again check its operation. In some rare cases, you may have to rearrange component placement and/or reroute conductors to eliminate unwanted effects. For example, in very high gain audio circuits, you want to get the inputs and outputs as far apart from each other as possible to prevent feedback. Also, in vhf and uhf circuits, you want lead lengths and component spacing as short as possible to prevent excessive signal losses and reduce the chances of interactions. This done, you can proceed to the final layout phase.

(Continued on page 52)

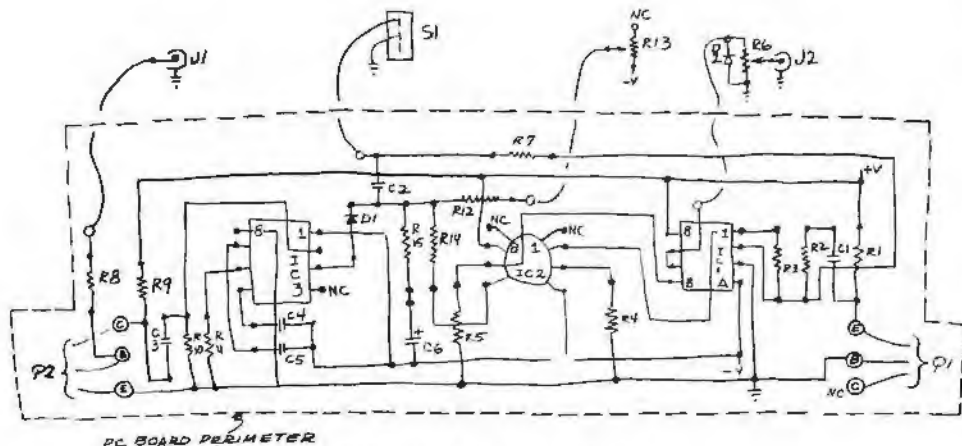


Fig. 2. Redraw circuit as it will be laid out physically.

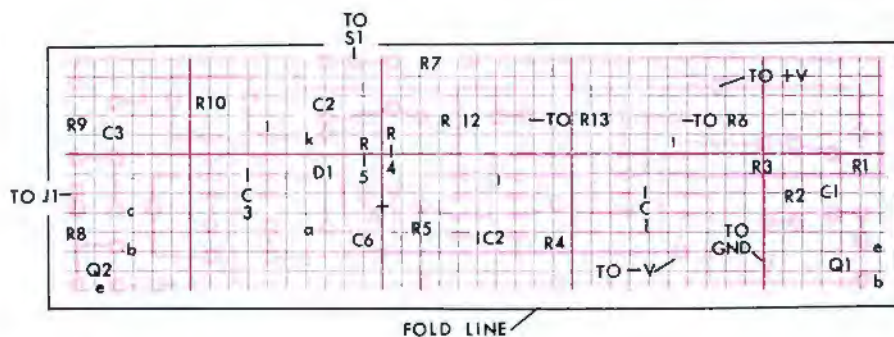


Fig. 3. Final artwork laid out exactly to scale. It helps to use 10 × 10 squares-per-inch graph paper for accurate placement of pads.

**Finalizing the Layout.** The final layout of the etching and drilling guide is essentially a redrawing of the rough layout to the exact scale of the components you plan to use. You can do the layout

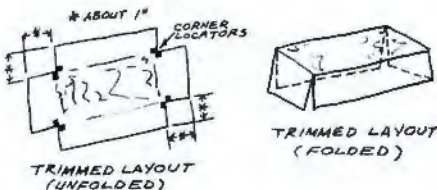


Fig. 4. Trim and fold final artwork as illustrated here.

directly on graph paper and use it as a guide for drilling holes and transferring to the pc blank simple circuit designs with an etch-resist pen. For more complex and critical layouts, it is best to tape a sheet of clear Mylar over the graph paper and use pc artwork aids (donuts, IC pad patterns, tapes, etc.) to lay out the guide. This guide can then be used directly to expose positive photoresist treated pc blanks. For the remainder of this article, we will concentrate on the use of the paper guide. Mylar film guides are used in the traditional manner.

As you work up your layout, plot your progress on either the rough layout or a piece of tracing paper taped over the rough layout.

Typical spacing between component lead pad centers and copper foil conductor widths and spacings between foil conductors are detailed in the Table. For components not listed, take the dimensions directly from the actual components you will be using in your project.

When the final layout is completely drawn up, indicate all component locations by schematic designation and orientation. See Fig. 3 for details.

**Making the PC Board.** Trim and fold your final layout as illustrated in Fig. 4. Then cut the pc blank to the exact size of the folded guide and remove any burrs from the cut edges with a file. Fold the guide over the pc blank with the copper side of the blank directly under the guide. Secure the loose tabs of the guide to the blank side of the pc blank with masking tape (Fig. 5).

Lightly indent the pc blank in the center of every hole location with a sharp-pointed center punch or an awl. Drill a hole at every hole location through both the guide and the board.

When all holes are drilled, carefully remove the guide from the blank and store it away for later reference. Give the pc blank a thorough cleaning with No. 00 steel wool (do not use the type of steel wool that comes saturated with its own soap). Once the blank is cleaned, handle it only by its edges.

The pad-and-conductor pattern on the guide can easily be reproduced on the pc blank with a resist pen because the drilled holes themselves provide a "map" on which to build. In most cases, the resist pattern can be laid down with nothing more than the resist pen, working freehand. If you have a relatively complex pattern that must accommodate many IC's, you can speed the operation with the aid of rub-down pads and fill conductor lines and discrete-compo-

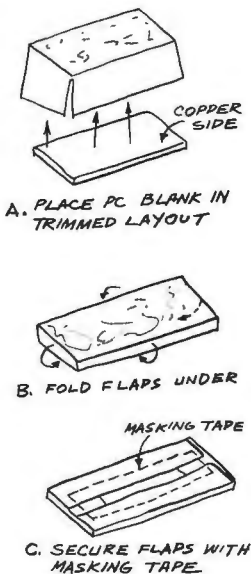
### TYPICAL DIMENSIONS FOR PC ARTWORK

Component	Spacing
¼-watt resistor, signal diode	0.4" (4 boxes) between leads
½-watt resistor, power rectifier diode	0.5" (5 boxes) between leads
Disc capacitors	0.3" (3 boxes) between leads
Radial-lead electrolytic capacitors	0.2" (2 boxes) between leads
Other resistors, capacitors	Measure for spacing
Transistors (small signal)	0.2" (2 boxes) between leads
DIP IC's with up to 18 pins	0.1" (1 box) between pins, 0.3" (3 boxes) between rows of pins
DIP IC's with more than 18 pins	0.1" (1 box) between pins, 0.6" (6 boxes) between rows of pins
Distance between pc copper conductors	0.05" (½ box) minimum, 0.1" (1 box) standard
Spacing between component pads	0.05" (½ box) minimum, 0.1" (1 box) standard

Note: Number of boxes is for 10 box/in. graph paper only.

ment pads freehand with the resist pen. As you work, be certain to fill in the areas around the drilled holes with resist to assure that, during etching, excessive copper will not be removed.

When you are finished reproducing the pattern on the pc blank, carefully compare it with your final-layout guide. The two must be identical. If you have



*Fig. 5. Cover pc blank with trimmed artwork (A), fold flaps under (B), secure with tape (C).*

made an error, you can correct it by erasing with an ordinary pencil eraser and laying down new resist.

Pour pc etchant (ferric chloride or ammonium persulfate) into a plastic or glass tray to a depth of about  $\frac{1}{4}$ " (6.4 mm). Place the prepared pc blank copper-side down into the tray and slowly agitate the solution by rocking the tray gently. Periodically check your progress with the etching action by carefully lifting one end of the pc blank with a toothpick. When etching is complete, all the copper cladding from the uninked areas will be etched away. At this point, you can remove the pc board from the etchant and thoroughly rinse it under running water. (Discard the used etchant. Don't try to save it for re-use.)

Now remove the etch-resist ink from the copper pattern remaining on the board. Use scouring powder and steel wool under running water for this. Then pat the board dry with a paper towel.

The only thing remaining now is to install and solder into place the components that go on the board. Use your final-layout guide as a reference to component locations and orientations. ◇