Making a Printed Circuit Board

Richard L. Lerner

The audio professional is often called upon to construct small circuits. Printed or etched circuit materials can facilitate the job considerably. This article details several of the methods used for small-scale boards.

MANY construction articles and projects show printed circuitry as the wiring method of the device. With a printed circuit, the constructor of occasional circuits can be certain that his copy matches the original in such vital considerations as lead spacing, component orientation, uniform capacitance between leads, and similar interaction effects.

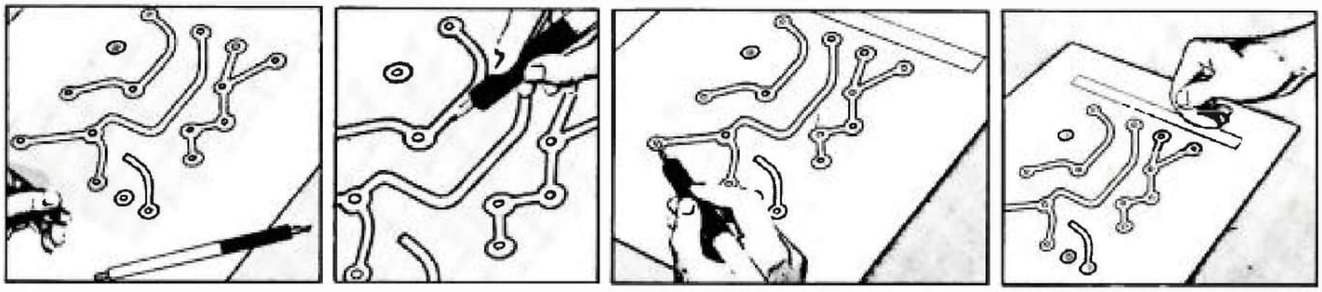
In industry, the single-sided printed circuit is utilized principally because it offers manufacturers a savings in assembly and wiring costs, in addition to reasons of accuracy. Double-sided boards are used for volume reduction needs.

Sequence of Construction

1. *Parts Layout and Circuit Configuration*
2. *Conductor Path Layout*
3. *Conductor Path and Pad Taping on Clean Film*
4. *Photographic Reduction*
5. *Photographic Projection on Photoresist*
6. *Etching and Cleaning*
7. *Plating, Drilling, Cutting, and Eyeletting as required*
8. *Component Insertion*
9. *Soldering*
10. *Test*

This sequence represents one of the roads toward the development of a small-scale printed-circuit. Alternative for creating a copper conductive path on a board are discussed in the text.

Mr. Lerner is a section staff engineer in the environmental test section of Grumman Aircraft Corp. Bethpage, N.Y. He is also an assistant editor of **db**.



(A) (B) (C) (D)

Figure 1. The steps necessary to use Ulano Rubylith® masking films. In (A) a sheet of this transparent deep red material is placed above the original drawing. A sharp knife traces the circuit paths (B) cutting through the red coating. Then lifting the edge with the knife, (C) the red is stripped off (D) leaving a clear plastic coating underneath. This can then be used as either a positive or negative for the etching procedure.

The usual industrial processes will be described, and then the applicable parts will be used for small-scale construction, considering particularly that the average project is not a production effort involving large numbers of boards. The do-it-yourself process consists of laying out the parts and circuit in the desired configuration and connections, making a large-scale pattern, reducing it photographically, imprinting it on the copper clad insulating material in such a way that the undesired copper may be etched away and the conductive paths left. Then fabricate and assemble to produce the desired end product, an electronic module consisting of component parts connected together by the best commercial conductor – high-grade copper.

The industrial designer, starting with a schematic, makes a large scale layout, usually 4:1, of all the components on a grid layout, showing all connecting paths and usually a pattern which will mate with a standard printed circuit connector. A piece of transparent Mylar (DuPont) or similar stable material is fastened over the layout. Tapes and prepunched patterns of 4X shapes such as pads, curves, transistor mountings and just about anything else commonly used, are pasted down in the proper position. These tapes are flat black in color. Corner markings and sometimes cutting lines and identifying markings or lettering are appropriately placed. Great attention is paid to removing any light colored areas or gaps on the patterns obtained, as these might become pinholes in the completed circuitry, or even worse, failure to complete a circuit. The tapes may be joined by overlaying strips, since the next step is not particularly sensitive to depth.

Following a meticulous check, the pattern is taken to a photographic shop, where, depending on the etching process intended, a 1:1 scale clear negative or positive is produced. This is also checked for pinholes. Many economic questions influence the method that will be used from here to completion, but the ultimate result is to obtain a pattern on the copper-clad board that is such that an etching chemical cannot etch away the desired copper areas. Etch-resistant material, commonly called *resist* can be directly silk-screened on the board, using a silk screen made from the photo obtained earlier. Another type *resist* is a light-sensitive emulsion which does not become etch resistant until after light exposure and (sometimes) development processes. Whatever the resist application method, the boards are inspected for undesired resist-free areas, touched up if neccessary, and placed in a bath which removes the undesired copper. Frequently the circuit, after cleaning, is plated with metals such as gold (very thin plating) or solder to improve later solderability and to minimize corrosion. Areas like connectors, where wear is expected, are plated with hard metals such as rhodium. Holes are drilled for the component leads. Frequently small eyelets are used in the holes to improve reliability. Many ingenious machines have been developed for automatic insertion and soldering. The use of these or similar processes is not reserved to industrial giants. Many printed-circuit vendors are equipped to handle and welcome experimental or low-volume orders. Some will work from a hand-drawn schematic and layout.

The process just described is quite common, with variations limited only by the ingenuity and financial resources of the industrial concern. As volume gets higher or packaging density greater, the simple printed-circuit board becomes multi-layer, double-sided, interconnected, modular, miniaturized, flexible (even printed-circuit cables!) and in some cases even has components such as high-frequency inductors and capacitors formed by the copper pattern. There are also techniques, many of them patented, for using the copper lines as microwave waveguides, and as high-efficiency antennas.

How can all this be applied by the occasional fabricator? Let us list the steps in the industrial processes and discuss their applicability or necessity to the low-volume, low-cost, but not necessarily low-precision small project. Remember that most industrial processes are carried out by individuals. In particular, the taping of a printed circuit layout is usually done by one man working on a large, flat surface, placing tapes, pads and sometimes using black ink to form light opaque patterns. The scale used depends on the processes intended; most photo reduction is done at 4:1. There is no reason why you cannot work at 1:1, but closer spacing and neater boards will result if photo reduction is planned and used properly, since minor errors become ¼ the size in 4:1 reduction.

Layout: This is where extra attention yields not only a better looking board, but a board into which components can be assembled and if necessary replaced with ease. The smallest line width that you should consider is 1/16 inch. Line separations of at least that figure are recommended, particularly if a photo reduction process is not contemplated. Effort should be made to have components faced so that values or markings may be read, so that adjustments can be made easily, and to provide uniform orientation. The last is particularly helpful when the circuit contains polarized components such as diodes.

Taping: Taping supplies are available at most drafting supply houses. If not, tapes and prepunched pads may be purchased from several companies including **By-Buk** and **Chart Pak.** There is no reason, for one or a few boards, why you cannot eliminate production steps. The tape may be applied directly to the board, as it is sufficiently etch resistant to be used. Others may prefer to use a very convenient process marketed by **Vector** (and possibly others). Printed pads, lines and other useful patterns are available in sheets. Rubbing the back of the sheet transfers the pattern, which is composed of resist, onto the board. With either tapes or transfer-resist, be careful to inspect all joints to be sure that tapes don't overlap, leaving a gap that permits etchant to creep underneath, and that resist gaps are eliminated. Here, a small amount of liquid etch-resistant material is helpful. In a pinch, enamel paint or nail polish can be used.

There is another process, frequently used by industry, which can be helpful. Assume that the pattern of lines, pads, etc. is available, in correct size. There is a material called *Rubylith®* (Ulano Corp.) available. It is a laminate of red and clear plastics. The red plastic is cut away with a small knife from those areas where it is desired to leave copper. This material, together with photo-resist can be used to make excellent boards. As an added advantage the same piece of Rubylith can be used to make more than one board. This process is illustrated in Figure 1 showing the cutting, peeling, and finished product. The required material is available at many art supply stores in most metropolitan areas, according to the manufacturer.

Photo-reduction: While I haven't tried to do it myself, I understand that many commercial and studio photographers welcome photo-reduction work. Keep the board size small or you will pay quite a fee. The etch-resist process will determine whether a negative or positive image is desired. The advantage of multiple boards from one pattern obviously exists here also. Other advantages are precision, location accuracy, and uniformity. This process usually takes time, so allow for it in planning.

Etch Resistant Application: The simplest and least complicated method of resist application is to paint the desired pattern onto the copper with a fine brush. This technique is frequently used for low-precision one-of-a-kind boards.

Precision may be improved by using tapes or rub-on resist as an etchant mask. If a photoreduction or Rubylith® mask has been made, then the raw board may be evenly coated with photoresist. This material becomes etch resistant only after exposure to intense light. Follow the manufacturer's instructions for application and development, but remember that in this, as in any chemical process, cleanliness, uniform mixing and application, and a respect for caustic materials is extremely important.

Vector, among other companies, offers a variety of etch-resistant patterns which may be transferred to clean copper by rubbing over the backing paper on which the patterns are supplied while carefully holding the board and pattern in alignment. Touch-up resist comes with the patterns. The company also markets, through most of the common radio parts houses, a complete kit containing board material, resist patterns and touch-up resist, and etchants.

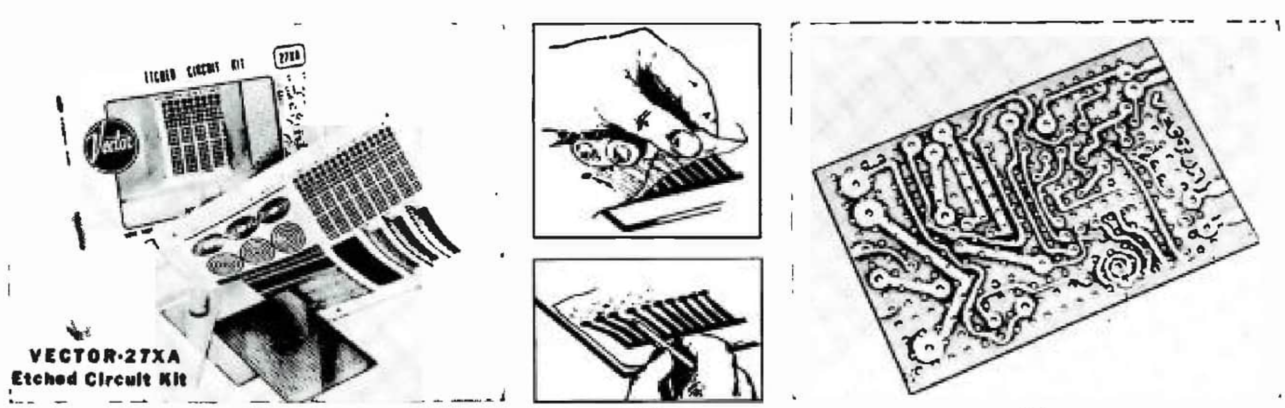
Plating: This step is usually included on commercial boards for its protective qualities, particularly against contact wear and corrosion. The processes involved are sufficiently complex to be out of consideration for small-scale projects. With the exception of some gold-flash dips, most plating requires high current supplies, electrodes, multiple baths and considerable experience. The use of plated circuits is best left to the industrial practitioner.

Drilling, cutting and eyeletting: For the usual project, drilling consists of selecting a drill which makes a hole large enough to pass the thickest component lead without obliterating the pad, and drilling the holes.

During any fabricating process, care should be taken to prevent injury. You should be aware that glass fiber reinforced boards, when cut, have glass chips in the sawdust, which should not be inhaled, or allowed to get into eyes. In addition, some people are allergic to some plastic dusts. Within the above cautions cutting printed-circuit boards may be done with tools such as sabre saws, jig saws or files.

While it does require more space, in terms of larger pads, the use of eyelets is common. An eyelet holds down the copper pad during lead insertion, soldering and repair. Convenient hand tools and supplies of eyelets are available from many of the electronic mail-order supply houses.

Component insertion: This might seem a trivial process,



(A) (B) (C)

Figure 2. Vector supplies a complete kit (A) for the production of small etched-circuit boards. At (B) you can see a detail of the etch-resist transfer sheets and the procedure used to transfer them to the board. In (C) is a finished transfer on a breadboard supplied with the kit.

but care taken here, and in designing a layout to permit proper insertion techniques, can yield a circuit of higher reliability and of easier maintainability, (to say nothing of just looking neater).

Most electronic components can be broken or severely strained by bending their leads too closely to the body. The usual allowance for high reliability boards is 1/8 in. between the component and the start of any lead bend, with a minimum bend radius of several lead diameters. For your occasional construction purposes, adding 1/4 in. to the component length when laying out pad centers is sufficient. Use care not to bend the leads so tightly that they form a sharp corner. Form them parallel to each other and perpendicular to the body. Insert them into the correct holes using care to orient critical components such as diodes, polarized capacitors, and transistors correctly. It is neat and also convenient to orient resistors so that the color codes line up in one direction.

As each pair of leads is inserted, push the component all the way down onto the board, and bend the protruding leads about 45 deg. in the direction of the line connected to the pad. Then clip it off leaving about 1/8 in. There are three reasons for this: the component is held during subsequent operations, the joint when soldered is strong, and servicing will be easier than if the lead had been fully bent.

Soldering: Printed-circuit soldering does not differ from the kind of soldering usually encountered in most electronic kits or assemblies, except that care must be exercised not to overheat the joint. Overheated printed-circuit lines will peel from the board easily. They may even fracture just from the differential rates of expansion between copper and plastic. Most commonly however, the relative neophyte will be so afraid of overheating that he will fail to heat the joint properly, winding up with a cold solder joint. The technique I use is to rest a properly trimmed soldering iron tip of about 25 watts rating, on the component lead until the *lead* is hot enough to melt solder applied to it. Quickly move the iron so that the tip rests against the pad as well as the lead, while applying solder to the lead and the pad. As soon as the solder has flowed smoothly around the desired area, remove the heat. The result should be a smooth, shiny, slightly conical joint. Inspect it for evidence of a cold joint (wrinkled, dull appearance) or of overheating and peeling. Severe overheating might require tacking a piece of wire in place of a conductor. Cold solder joints are easier—simply resolder carefully. It is not necessary to hand solder the entire conductor surface.

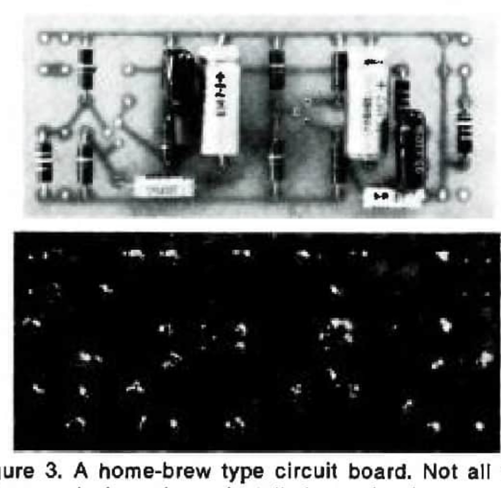


Figure 3. A home-brew type circuit board. Not all the components have been installed at this stage.

Testing: Of course, the final step is testing. Just how much is needed will be determined by the nature and extent of the project. It is always a good idea to check the continuity from one lead on the component side of the board to another connected lead. Indications of over one ohm will indicate likely poor soldering joints or broken lead paths. For one-of-a-kind or first-piece boards more detail may be called for since the board design can control the parameters of effects such as stray capacitance, inductance, and leakage between conductors. Finally, checks of the physical placement and for component damage during assembly is a good idea.

Circuit Board Materials Suppliers

Allied Chemical Corp., Box 70, Morristown, New Jersey 07960: printed-circuit etchants.

By Buk, 4326 West Pico Blvd., Los Angeles, Calif. 90019: electronic markings, pressure sensitive tapes, layout materials.

Chart-Pak, Inc., One River Road, Leeds, Mass. 01053: electronic markings, pressure-sensitive tapes, layout materials.

Cinch-Graphic Div., 200 S. Turnbull Canyon Rd., City of Industry, Calif. 91744: printed-circuit etchants, finished p.c. boards.

Commercial Plastics and Supply Corp., 630 Broadway, N.Y.C. 10012: laminated board materials.

Datak Corp., 63 71st St., Gutenberg, New Jersey 07093: electronic markings, pressure sensitive tapes, layout materials.

Eastman Kodak Co., Department 454, Rochester, N.Y. 14650: photo resist, technical publications and information.

Garlock Electronic Products, 8 Fellowship Rd., Cherry Hill, New Jersey 08034: laminated board materials.

General Electric Company, Inc., Laminated Products Dept., Coshocton, Ohio 43802: laminated board materials.

Kepro Circuit Systems, Inc., 3630 Scarlet Oak Blvd., St. Louis, Mo. 63122: circuit-board kits, photo resist.

Mica Corp., 4031 Elrenda St., Culver City, Calif. 90230: laminated-board materials.

Mica Insulator Div. of 3M, 801 Broadway, Schenectady, N.Y. 12305: laminated-board materials.

Photocircuits Corp., Glen Cove, N.Y. and Anaheim, California: finished p.c. boards.

Russel Industries, Inc., 96 Station Plaza, Lynbrook, N.Y. 11563: electronic markings, pressure-sensitive tapes, layout materials.

Synthane Corp., 12 River Rd., Oaks, Pa. 19456: laminated-board materials.

Ulano Corp., 610 Dean Street, Brooklyn, N.Y. 11238: masking films.

Vector Electronics Co. 1100 Flower St. Glendale, Calif. 91201: photo resist, printed-circuit etchants, circuit-board kits, technical information.

Write to the individual companies for further information as well as local distributors of the product. Other good sources for additional materials are mail-order electronics supply catalogs.