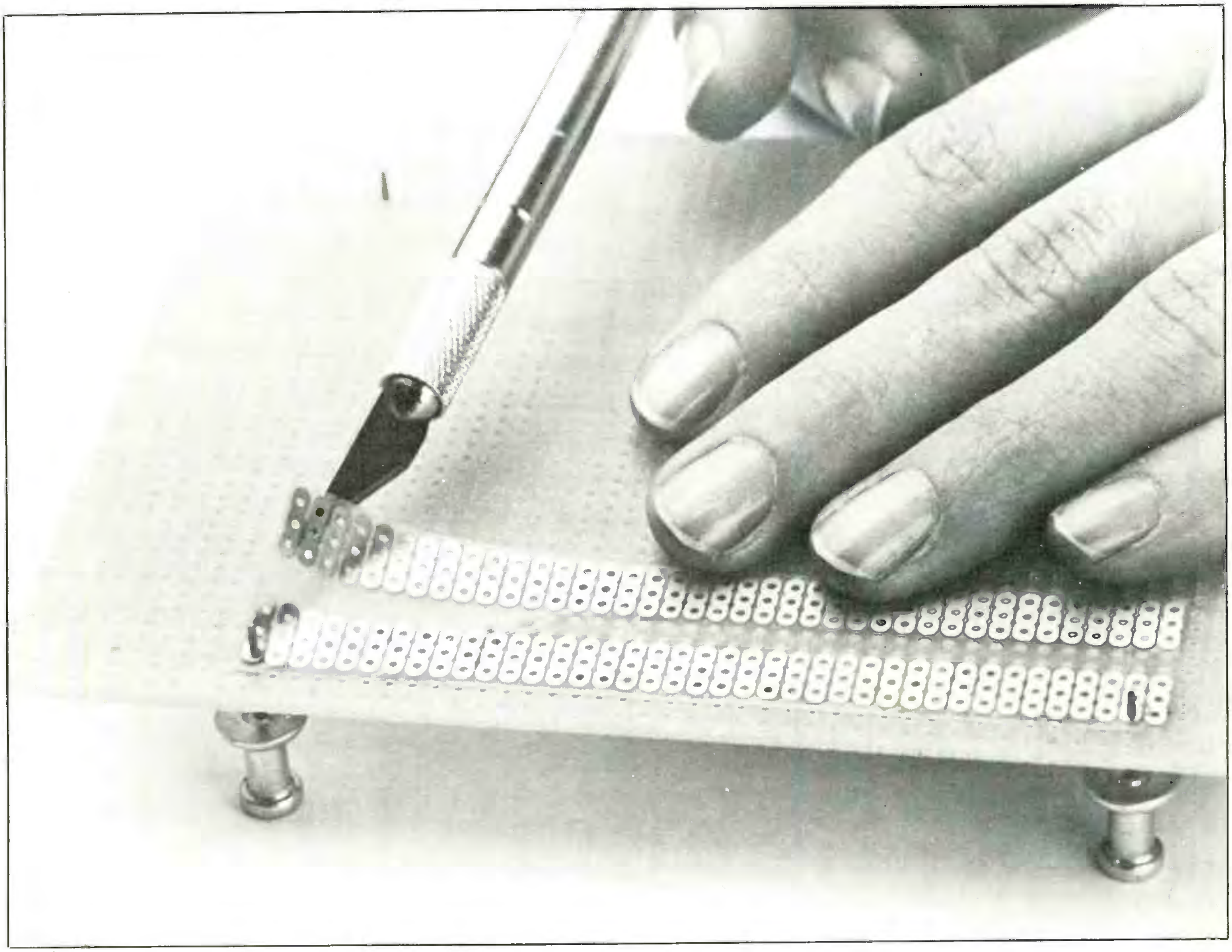


The "Easy Circuit" Way To Make Circuit Boards

Adhesive-backed materials eliminate photography and chemicals to speed circuit-board fabrication

By Harold Wright

There are a variety of ways to produce printed circuits singly. Typically, photochemical techniques are used on copper-laminated boards. This method enables a builder to accurately replicate printed-circuit foil patterns such as those printed along with construction projects in *Moden Electronics*. To many people this is an odious task, requiring photographic methods, dealing with messy chemicals, drilling component-lead pad holes, and careful monitoring time.

An interesting alternate method, the subject of this article, avoids all this, substituting instead the laying down of pressure-sensitive, copper-clad tape on pre-drilled boards. This E-Z Circuit™ system by Bishop Graphics. Inc., Westlake Village, CA, is examined here.

The Materials

Pressure-sensitive materials used with E-Z Circuit come in the form of copper patterns that are laminated to a thin layer of epoxy-fiberglass substrate. The underside of the substrate is coated with a special adhesive that is protected by a release liner. The adhesive is formulated to provide both good shelf life (as long as the release liner hasn't been disturbed) and good adhesion when the liner is removed and the pattern is placed on an appropriate circuit-board substrate. Patterns can be interconnected with copper tapes of various widths. The tapes have no substrate; instead, the adhesive is applied directly to one side and is protected by a peel-away release liner.

Adhesion of the patterns and tapes increases with time up to about 48 hours after the liner is removed and the materials are placed on the circuit board material. After 48 hours of curing, adhesion is approximately twice as good as it was after one hour. A pattern or tape can be removed and repositioned up to 24 hours after it has been applied to a circuit board. However, patterns removed after about 24 hours are generally no longer reusable, since their adhesion will have diminished to near zero.

Pressure-sensitive patterns and tapes will bond to most epoxy-glass circuit-board materials. However, they will not bond to Teflon or untreated polyolefin boards. So keep this in mind when you're planning to translate a circuit's conductor pattern to a working board design.

Many types of pressure-sensitive patterns are available (see Fig. 1), including pads for ICs and transistors, donuts for terminating a copper-conductor run, and various tapes ranging in width from 0.015" to 0.250". You'll also find circuit board edge-connector patterns for plug-in boards, multiple parallel tapes on a single substrate, power-distribution strips, and many others. If you need a special-purpose pattern that isn't available you can make your own by cutting them from a sheet of cut-and-peel copper, which also has the special adhesive on one side.

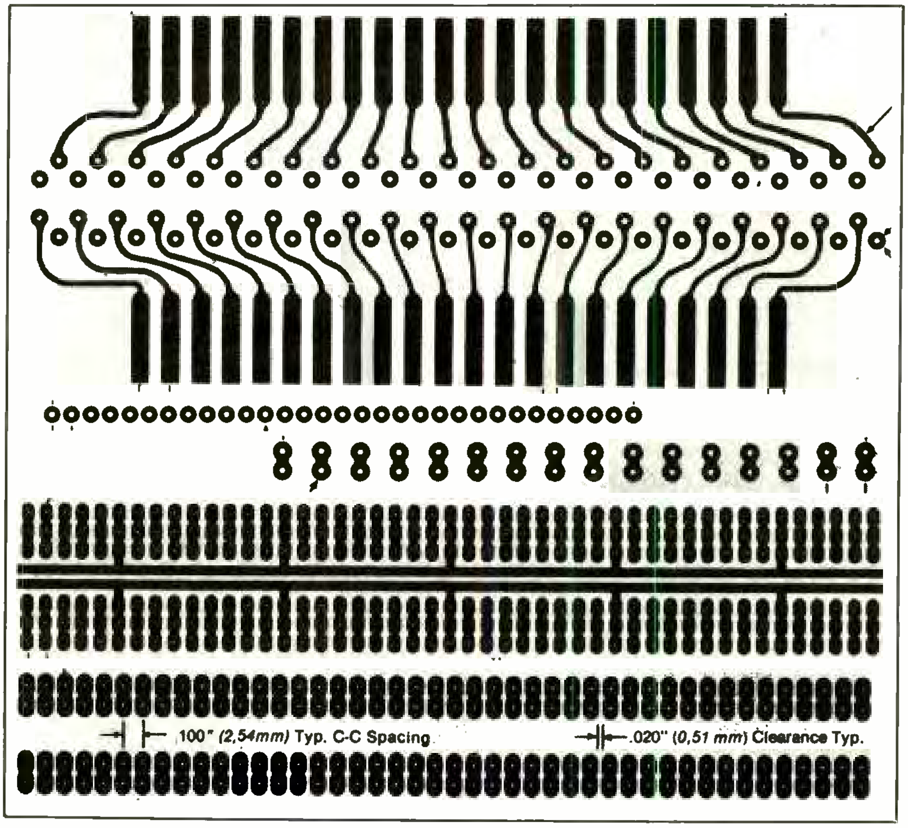


Fig. 1. Many types of pressure-sensitive patterns are available, as demonstrated by the small sampling shown here.

Perhaps the most versatile pattern in use today is the DIP strip popularly used for dual in-line package (hence the DIP acronym) integrated circuits. Such strips contain two continuous rows of pads spaced on 0.1" centers and with 0.3" spacing between the rows. Each pad can have two, three or four holes.

DIP strips can be cut to the lengths needed for use with 8-, 14-, 16-, 18-, and 20-pin DIP ICs, as well as for DIP-type bridge rectifier assemblies and optocouplers that require only four pads. They are equally useful for mounting DIP switches and DIP-type resistor arrays. The double- and triple-hole strips can be very useful where two or three components must meet. Also, these strips can be used as termination points where a row of resistors and/or capacitors must mate with wires leaving the board.

For 24- to 40-pin ICs, there are strips available with the 0.6" row spacing required for these devices. However, if you don't anticipate heavy use for these strips and prefer to save the expense of stocking them, the narrower DIP strips can be cut down the center and manually spaced when you do need them.

Even if you're committed to point-to-point wiring, the pads alone will

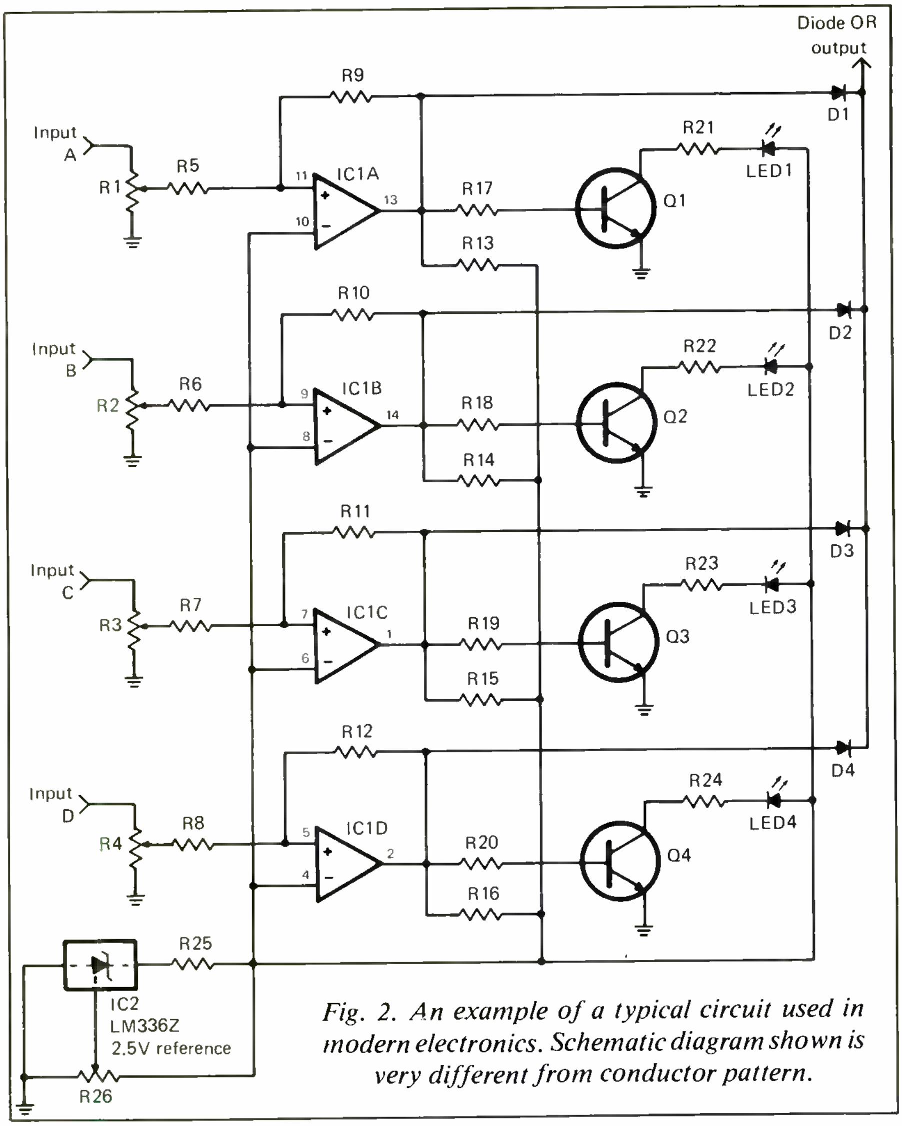


Fig. 2. An example of a typical circuit used in modern electronics. Schematic diagram shown is very different from conductor pattern.

greatly simplify the task of obtaining good, reliable connections to IC and socket pins. While it costs more to use self-adhering patterns than to go strictly the point-to-point wiring route, it costs less than going the make-it-yourself printed-circuit board route. As compared to wire-wrapping circuits, it's easier to trace when troubleshooting and much simpler to make making last-minute changes in the circuit.

Planning The Layout

The procedure for planning the copper tape-and-pattern layout of a given circuit is much the same as that for a printed-circuit layout. If you're laying out a project of your own design, it's a good idea to first assemble the circuit on a solderless socket and check out its operation to be certain it performs as you want it to.

The next step is to redraw the schematic diagram with the ICs and transistors shown as they appear on the conductor side of the board, assuming this is available. Keep in mind that pinout diagrams are invariably top views for ICs and usually bottom views for transistors. Keep in mind when the IC is on the board and is examined from the *conductor* side, its image will be reversed.

Since many circuits use multiple-device ICs nowadays, we'll use one such as our design example. In the schematic diagram, the triangles that represent the single sections of a quad op amp (Fig. 2) are placed to give the simplest, easiest-to-follow circuit. When the circuit is converted to this new format as shown, it may not look simple at all from the circuit board component layout point of view. The schematic layout in Fig. 2 has been planned for maximum simplicity, while Fig. 3 shows a conductor layout for the circuit.

When working out the first crude layout of a circuit's conductor pattern, one of the limitations of the tape system may be encountered—insufficient space to run a trace between two pads. A common practice in photochemically produced pc boards, where the pads are made very narrow, you won't normally be able to do this with pressure-sensitive materials. However, if there's an unused pin on an IC that doesn't have to be grounded or tied to B+ (as in the case with all CMOS and some non-CMOS devices), the pad for that pin can be trimmed away to leave a space wide enough for two narrow tape runs, as shown in Fig. 4. If you should do this, make sure to cut away the corresponding pin from the socket or IC so that it doesn't protrude through the hole and contact the tape.

Tape runs down the center between the rows of pads spaced 0.3″ apart are generally limited to three or four, using the narrower tapes. Patterns with 0.6″ spacing between rows of pads have more space and, thus, accommodate a greater number of end runs. Keep these points in mind as the layout develops.

Note in Fig. 3 that many tape runs don't terminate in donut pads. This occurs when several tape runs end side-by-side and are spaced 0.1″ apart. Since only very small donut pads could be used in such a case, soldering is much simpler if you puncture the end runs with a darning needle through the underlying holes. Note also in Fig. 3 that considerable use has been made of two-hole pads snipped from a DIP pattern.

Although Fig. 3 is a view from the conductor side, the components are shown in the positions they would normally occupy on the other side of the board. This has been done here to aid in relating Fig. 2 to Fig. 3. Normal practice when showing component placement gives a component-side view of the board assembly. If such a diagram is needed, it can be derived from the conductor-side pattern simply by laying the latter wrong side up on a piece of glass, shining a light through, and tracing the pattern onto a sheet of paper. Then all you need do is add the components to the tracing.

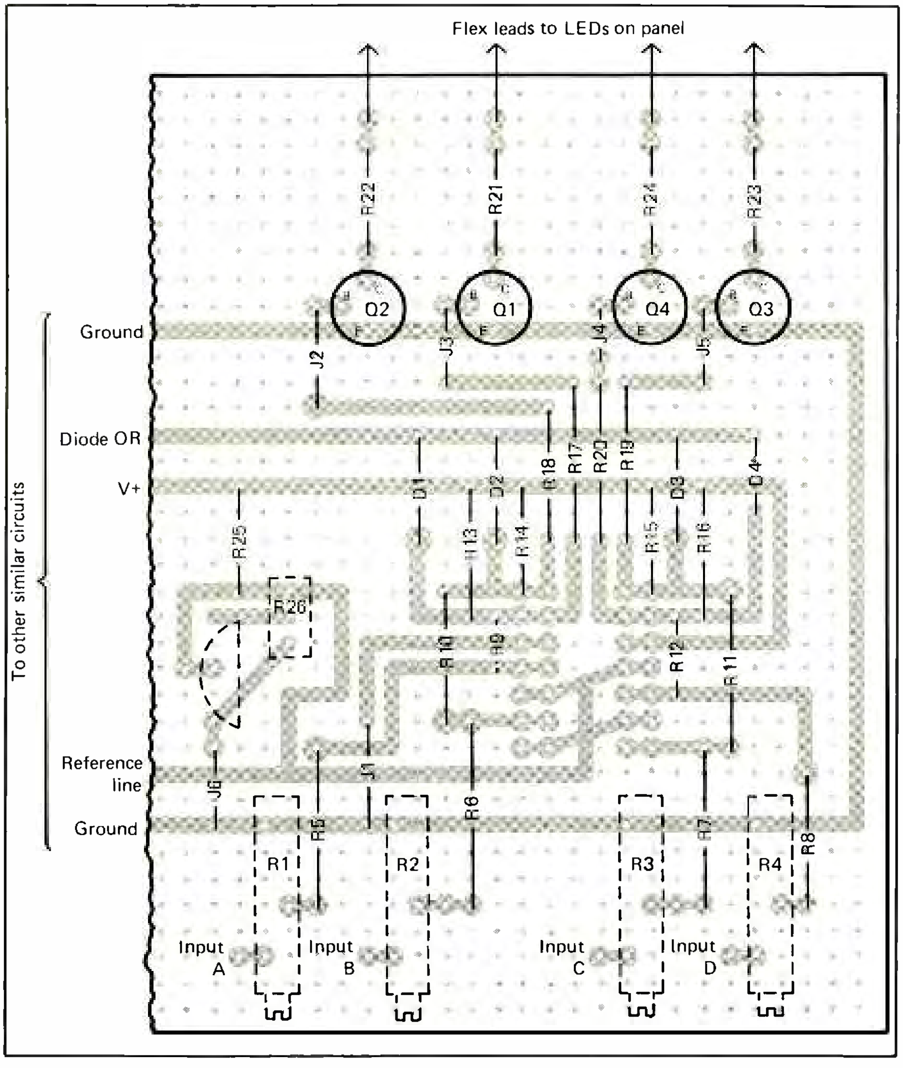
When you use pressure-sensitive materials, you can frequently avoid the need for jumper wires. Pressure-sensitive tape is available in various widths. When snipped to size and applied over existing conductors, you can bridge the conductors with copper tape to obtain insulated and mechanically stable crossovers (Fig. 5) without having to resort to traditional wire-in jumpers. This technique will usually suffice, except in high-frequency circuits and especially with wider tapes. (Where the tapes cross, a tiny capacitor is formed and at high frequencies could cause unwanted coupling between traces.) If in doubt, use a standard wire jumper on the *component* side of the board. Beneficially, if a wide positive supply tape crosses a wide ground bus tape, a small r-f bypass effect is obtained. Figure 5 shows an insulated crossover.

The final drawing of the scale layout for the patterns and tapes will be easier to draw if the work is done on grid paper, preferably with a 0.1" spacing. It will be easier still if you work two times actual size. A large-scale equivalent of standard perforated board can be made by placing ink dots every 0.2" horizontally and vertically on the grid paper. There are also film substrates with grids available from the copper pattern manufacturers and their distributors.

Working on a dotted grid will make it easy to determine if there will be room on the final board for the various components and how much space will be needed between each. For example, a standard ¼-watt resistor will require five holes spaced 0.1" apart (one hole for each lead and three holes for the resistor body). If the same resistor is mounted on-end, you may need no more than 0.2" of space. For other components, simply measure their bodies, taking into account lead requirements, and size your layout accordingly.

Projects using this system are easiest to wire if you use perforated board with 0.1" spacing between holes. Attempts to place a set of patterns and tapes on a nonperforated board blank can be disastrous, since you must drill holes through the holes in the patterns. Unless you use exactly the correct size bit and get the bit exactly on-center, the patterns and tapes will almost invariably "climb" the bit, perhaps even taking with them adjacent tapes and patterns.

Fig. 3. This is the conductor pattern for the partial circuit shown in Fig. 2. Note that IC1 plugs into the double row of pads in the center of the drawing, between resistors R9 and R12. This is not a practical circuit for you to build.



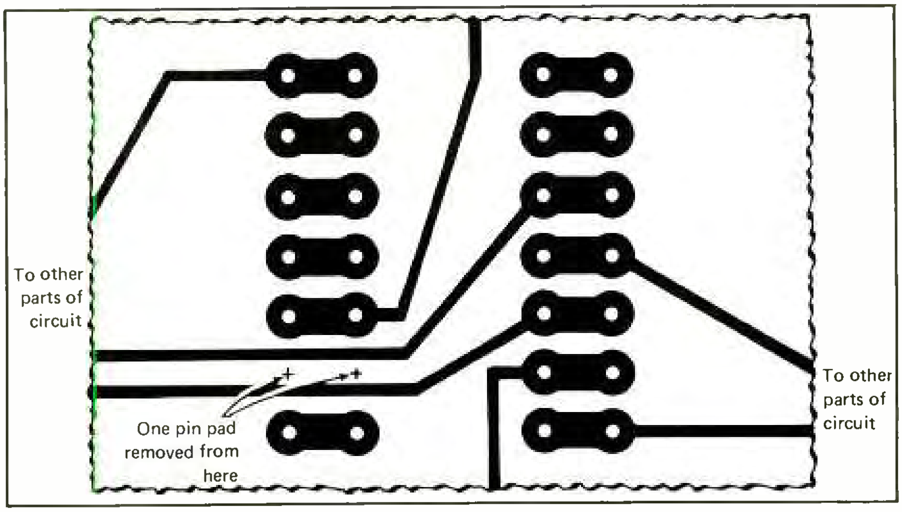


Fig. 4. How to route one or more conductors between IC pins. Use this technique only if unused IC pins do not have to be tied to B+ or ground.

When your layout is satisfactorily drawn on grid paper, simply transfer it to the perf board blank by counting holes horizontally and vertically. You will find that your tape layout is equivalent to a printed-circuit pattern. It can, in fact, be used for photochemical pc work at a later time should the need arise. Also printed-circuit layouts provided in electronics magazines can be duplicated using the tape-and-pattern system. If runs are shown between IC pins, however, some modification of the etching-and-drilling guide may be necessary. This could be as simple as using a couple of jumpers. Be aware, however, that where a construction project has a supplier for the pc board, it's simpler, less time-consuming and usually less expensive to purchase the ready-made board, especially if it's very complex.

Another method of board construction uses cut-and-peel copper sheets. Applying a copper sheet to a board blank allows you to draw the conductor pattern directly on the copper and to use an X-acto™ knife to cut away unwanted copper. This method, however, should be confined to simpler circuits, such as a power supply, and be used in conjunction with standard prefabricated patterns.

Tape runs must follow straight lines and various angles—but no curves! Don't attempt to bend a tape into an arc, no matter how narrow the tape or shallow the arc. This isn't really a handicap, because most routing required can be produced with a series of wide angles at the expense of a few extra solder joints. While 90° patterns are available, they do add an extra joint and increase cost.

Getting It Onto The Board

If you prepared a 1:1 or 2:1 drawing of the required pattern, assembly will go very quickly. The drawing should have a frame around it to define the size and shape of the final board (and that takes into account the needs of mounting hardware). Get the DIP patterns in place first. Count the number of holes from a reference corner in both directions in your diagram to determine where to place the first pattern, aligned with the same hole on the board.

To aid in registering patterns, insert a bulletin-board push-pin into the reference hole from the component side of the board. If you don't have a board-holding jig or vise, push

(Continued on page 80)

Fig. 5. Insulated crossover, shown to right of IC pad, obviates need for a physical wire jumper in many cases.

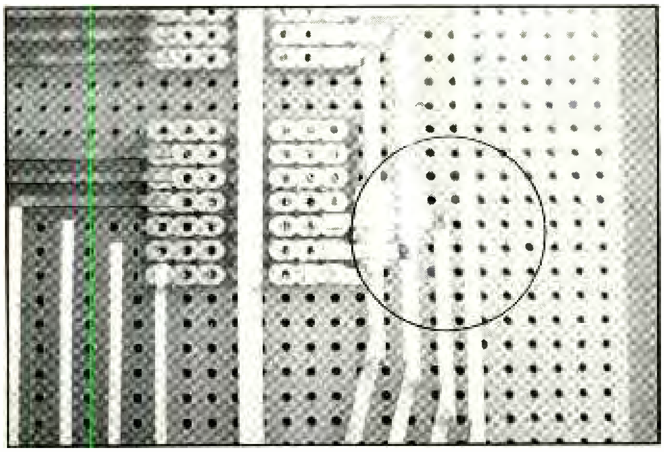
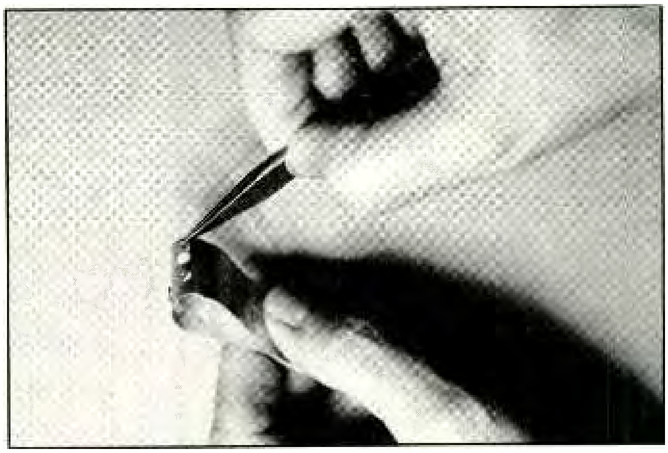


Fig. 6. Shown here is one way to get the release liner started. Use sharp-pointed tweezers for handling patterns.



three or more pins through holes near the corners of the board to keep it level on your work surface.

If a 14-pin pattern is to be positioned, cut off 14 sections in one piece from the DIP strip and peel off the release liner. Figure 6 shows one method for getting the release liner started. The liner can also be started from a corner by using a fingernail to pry it up. Figure 7 shows how to use the push-pin to achieve accurate alignment between patterns and board holes. Alignment is sometimes more easily accomplished with a pair of large darning needles.

Use a fine pointed tweezer to handle the pattern and avoid finger contact with the adhesive side. Slide the pattern's outermost hole over the pin (or spear the hole on one darning needle) and line it up so that the row of holes is in exact registration with the row on the board. A darning needle can be used at the other end of the pattern to further aid in registering the pattern. If the pad is a bit out of line, it can be levered into position with the darning needle.

When the pattern in properly aligned, press it firmly onto the board's surface. Hold the pad down alongside the needle with a fingernail and remove the needle and push-pin. Firmly press the pattern down over its entire surface. However, try to avoid touching the raw copper traces on the pattern and the copper tapes when you use them with your fingers. Place a piece of paper or plastic film over them before you use your fingers to press the patterns and tapes into place.

With all IC and transistor patterns in place, the circuit's conductor pattern can be completed by following your drawing to interconnect the various points with copper tape. Where there's room, terminate the tape ends with donut pads.

A word of caution: When a roll of copper tape is first used, it has a tendency to unravel. If it does this, the tape is almost impossible to roll up again. If you succeed in getting it rolled up again, it will invariably be kinked and wrinkled in places. When this occurs, the release liner is likely to work loose, allowing the adhesive to dry out and rendering those portions of the tape useless. To prevent any of this from happening, it's a good idea to gently press newly opened tape packs onto the adhesive side of a piece of wide masking tape to keep it wound.

Figure 8 shows a short length of tape being positioned on a circuit board. Figure 9 shows a short length of tape positioned to connect with a DIP pad being trimmed to the correct length with an X-acto knife.

There are two methods for making connections between two tapes. The manufacturer recommends butting together the two tapes and then flow-soldering the ends at the joint. If the ends of the tapes don't butt exactly and the cuts aren't precise, you'll have difficulty getting the solder to "take" across the joint. Even a hairline separation between the two butt ends will defeat soldering. Solder will build up on both sides of the joint and refuse to flow together. This is the same action that's so much of an asset when soldering the closely spaced pads on an IC DIP pattern.

The second method requires less precision when cutting the tape. Here, the tapes are overlapped and burnished flat. Figure 10 illustrates good and bad examples for overlap joints. (With both methods of joining tapes and tapes to other patterns, it's essential that the ends be absolutely flat. There must be no curl where they meet or overlap. If any curl is overlooked, the joint, when soldered, might appear to be perfect but will usually be an open circuit.)

It is essential that every soldered junction be checked with a low-range ohmmeter *as soon as it is made!* If you wait until you've finished solder-

Fig. 7. How to use registration pins to obtain correct alignment between perf board and EZ Circuit pattern holes.

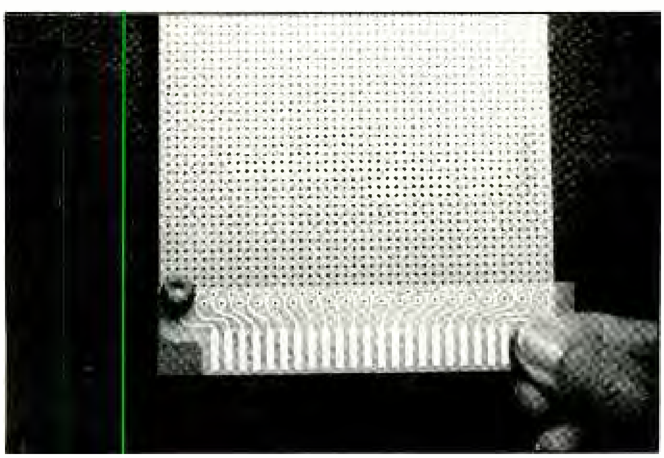
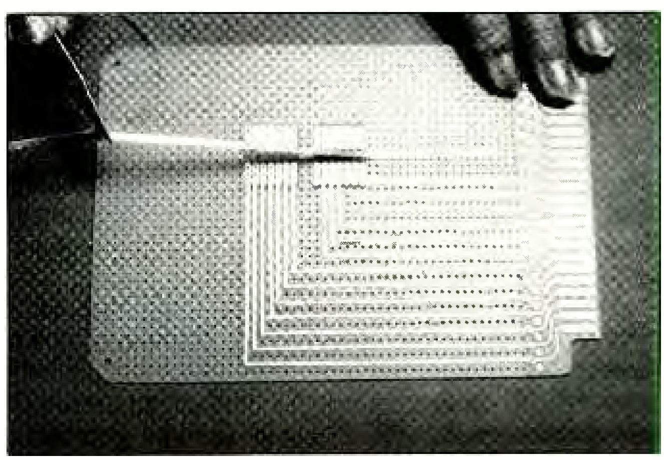


Fig. 8. Correct method of laying out tape. Use tweezers to avoid unnecessary handling of tape copper and adhesive.



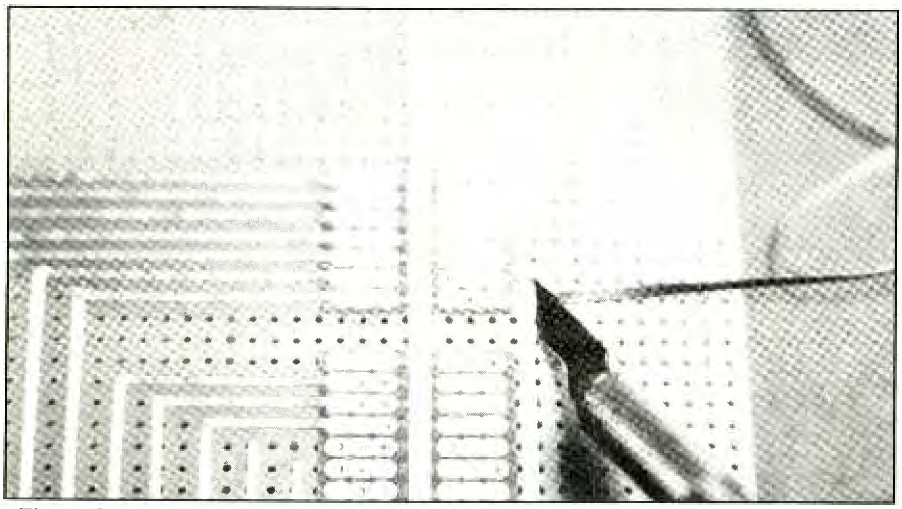


Fig. 9. Shown here is a short length of adhesive-backed copper tape being positioned to connect to a DIP pad. The terminating end of the tape is best trimmed cleanly and squarely with an X-acto knife or other very sharp cutting tool.

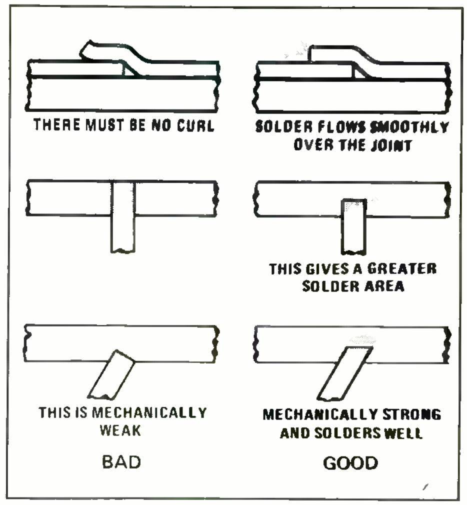


Fig. 10. Drawings illustrate good and bad tape terminations. Shaded areas show where solder is to be flowed.

ing all joints, you're likely to miss one or two, each of which is a potential problem.

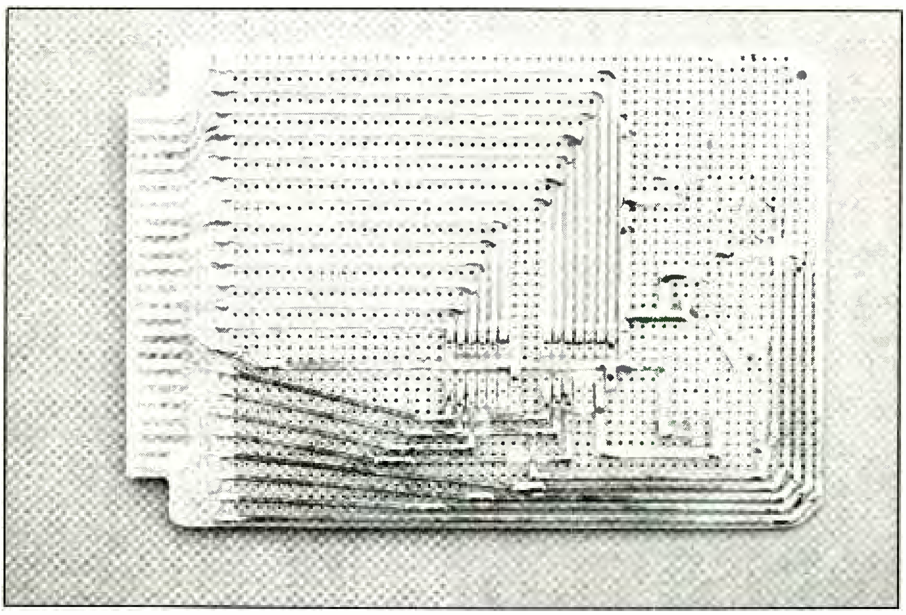
Occasionally, a tape joint will lift when soldering heat is applied. If one does, it may be possible to save the joint by pressing down on the tape with the point of a darning needle while reapplying heat. Solder won't stick to the needle, which can be removed as the solder cools. Very narrow and short pieces of tape must be soldered with care or they may be picked up by the soldering iron. If this happens, the old solder must be heated and wiped off the surface with a cotton cloth. Before you make another try at it, this surface must be free of solder bumps and ridges.

When overlap joints are used, solder bridges the joint more readily if the lower tape is heated and solder is flowed onto it first. Solder can then be flowed over the edge and onto the upper tape to complete the joint. Tape runs that connect to DIP and other patterns should overlap about halfway across pad holes. If a hole is to accommodate a component lead as well as the tape, push a large darning needle through the hole from the conductor side. This forms the tape to the hole contour and permits the component lead to enter the hole without pushing the tape off the pad.

Where tapes are connected to DIP pads at the inner rows of holes, the tape should overlap the hole completely and then be punctured with the needle. This ensures a good soldered joint between IC or socket pin and copper tape.

If you're soldering a component lead to one of the spare holes of a double- or triple-hole DIP pattern, make sure the joint between the IC pin and copper tape is also secure to the copper tape pattern. It's possible to have a good soldered connection between IC pin and tape while having and open circuit between them and

This is what a completed Easy Circuit board looks like after all connections have been soldered. Notice how turns are made by angling the copper tapes. This photo shows examples of 90° and angled turns and an insulated crossover.



the pattern pad. Solder must be flowed onto the pattern as well as the tape. However, reserve soldering of connections to DIP pin holes until *after* the socket or IC pin is in place. Otherwise, the solder may prevent entry of the pins.

With tapes and patterns all in place, carefully inspect the circuit to make certain that it agrees with both the conductor pattern drawing *and* the schematic diagram. If inspection discloses an error in the schematic or the pattern drawing, it's much easier to correct before any soldering has been done. You can now solder all connection points, including the pins of ICs or their sockets. Then carefully inspect every solder point, preferably with a jeweler's loupe. Look particularly for poor soldering and possible solder bridges, the latter most likely around IC solder pads. Having done this and corrected any suspicious connection, you can proceed to install and solder into place the remaining components.

When you're done, once again carefully inspect your work. Look for bad solder joints, possible solder bridging between closely spaced conductors and pads, and particularly for *unsoldered* points.

You might be wondering if you can work up a double-sided circuit board using pressure-sensitive materials. The answer is yest—to a limited degree. The main difficulty isn't so much that you must perfectly register the patterns on both sides of the board (this isn't too difficult in any case, considering that you'll be working with perforated board and prepared patterns), but the fact that you can't make plated-through holes. If you're planning to build a project that contains ICs, you won't be able to use sockets, because the sockets will prevent you from gaining access to the patterns on the component side of the board. If you forego sockets and solder the ICs directly onto the board, you run the risk of damaging the ICs with excessive heat. The answer, of course, is to use Molex Soldercon pins, which substitute for sockets and obviate the possibility of causing heat damage.

Where tape runs in double-sided work require interconnection between traces on both sides of the board, you can puncture both and use pretinned hookup wire to form bridges. Insert the wire, solder it to the tapes on both sides of the board, and trim away excess length with flush cutters.

Conclusion

You'll find that you need very few and simple tools to produce printed-circuit boards using the system described. These consist of a fine-point tweezer, an X-acto knife or Gillette "Widget" safety knife, a half-dozen or so bulletin-board push-pins, a couple of large darning needles, and a low-wattage soldering iron and appropriate accessories should do it.

Bear in mind, too, that E-Z Circuit material comes in many sizes and shapes for a variety of applications. For example, edge-connector boards are available for Apple II computer applications, complemented by insertion-type connector patterns. Other bus formats are available, too. Pressure-sensitive insulating tapes can be used to prevent short-circuits should you have any crossover points on your copper circuitry; copper power and ground distribution strips, which are nicely thick and wide, are among the many other adhesive strips and patterns one can use to simplify work.

The attributes provided by this method of making prototype experimental printed-circuit boards and single boards for construction projects should be appealing to many electronics experimenters and professionals who disdain the bother of photochemical work. **ME**