Hugh Wells W6WTU
1411 18th St.
Manhattan Beach CA 90266-4025

# Inside Digital TV/VCR Tuners

*Part 7: Conclusion.*

*The previous section described how the printed circuit boards for the data transmitter and receiver are prepared for processing. This section will complete the process steps, beginning with marking the board.*

It is desirable to have some identification markings on the board indicating voltage values, polarity, IC pin and transistor lead identifiers, etc. Prior to etching, a sharp instrument like a scribe may be used to scratch through the fingernail polish, exposing the copper as shown in **Photo A**. The scratch marks in the form of letters and symbols will be etched into the copper as shown in **Photo B.**

Many of the black ink etch-resist marking pens contain a water-repellent ink that is sometimes used for marking the bare copper prior to etching. Unfortunately, the ink does break down somewhat in the etchant, so that it isn't always satisfactory for making a reliable etch-resist for trace patterns. After etching, the marking pen is more suitable for marking the component placement, orientation, polarity, etc., information on the top side of the board.

**Etching the board**

Ferric chloride is the most common etchant available and perhaps the easiest and safest to use. The major caution with it is that it is dark in color and will severely stain cloth, so care must be taken to prevent spills and splashes.

The board will be ready to etch after the nail polish has dried — usually 30 minutes to an hour is a proper waiting period. There are many methods suitable for etching a board, but the one most suitable to your needs is the one that you should use. A simple and easy method is to pour the etchant to a depth of 1/8–1/4-inch into a flat glass or plastic dish. Float the board copper side down on the surface of the etchant. It will be necessary to lift the board periodically after about 30 minutes to assess the progress of etching


***Photo A.*** *A scribe is being used to scratch through the nail polish for marking/identifying the board.*

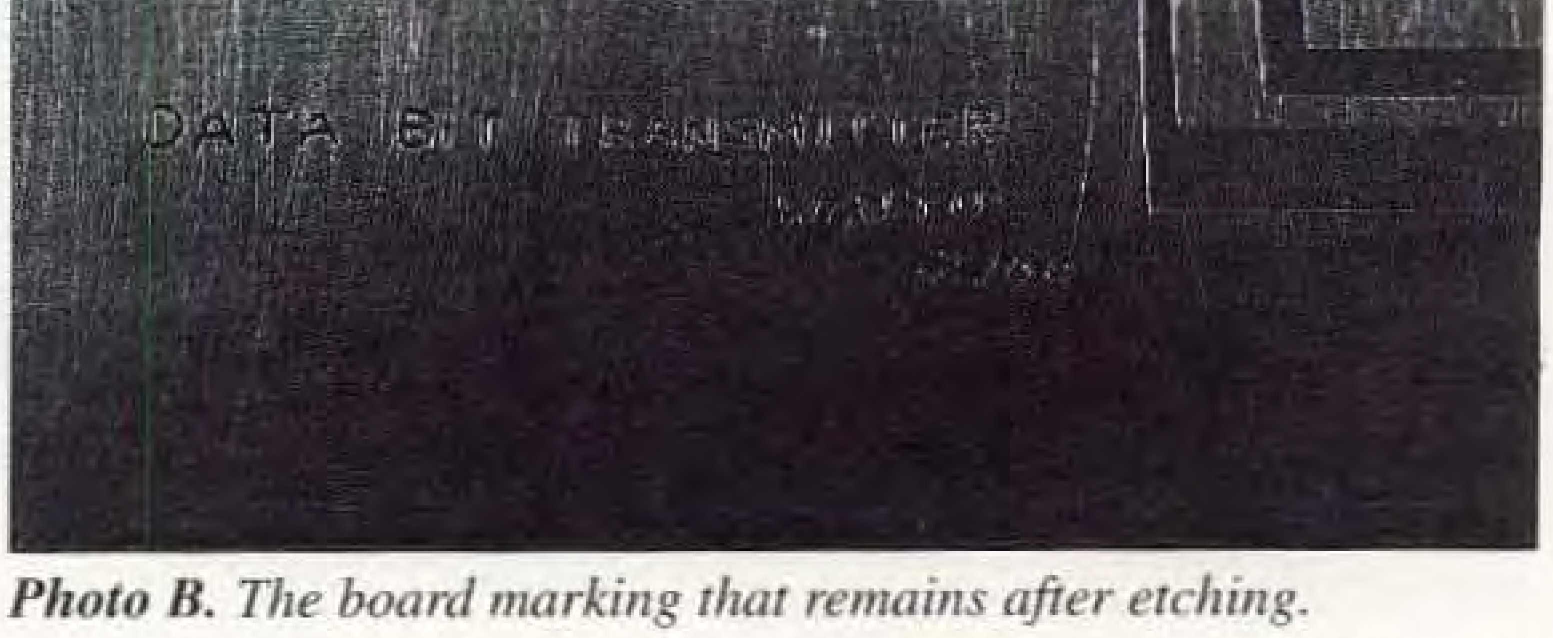

***Photo B.*** *The board marking that remains after etching.*

*Photo C. The circuit board floating on the etchant. A stick is used to lift the board momentarily to purge bubbles.*

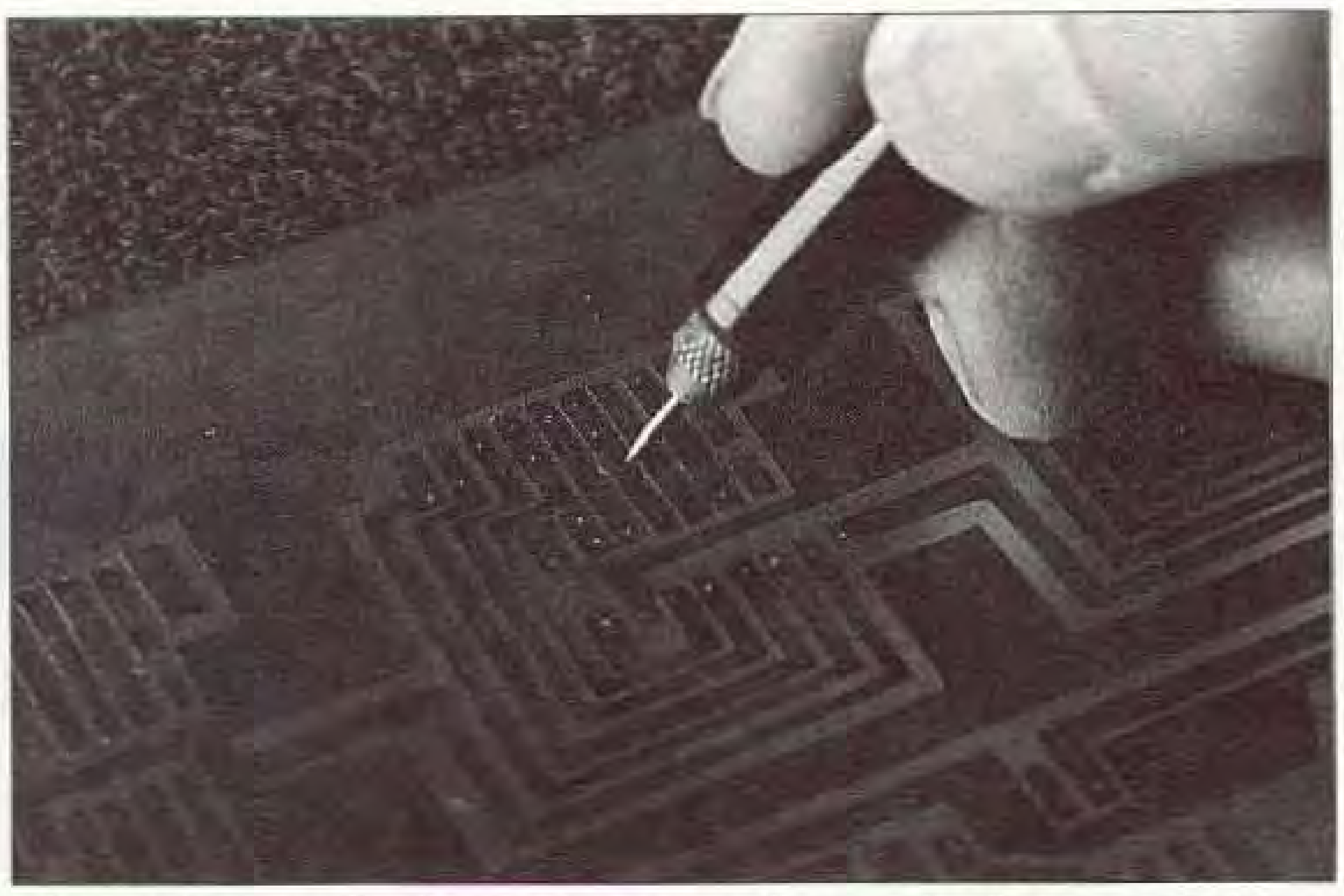

*Photo D. A scribe is used to scratch unetched copper bridges before returning the board to the etch.*

as shown in **Photo C**. A stick or tongue depressor works well for lifting the board. Also, lifting the board, then lowering it slowly, will allow the trapped bubbles to be purged.

When the etching process appears to be complete, remove the board, wash it with tap water, and dry it with a paper towel. The board can now be inspected for completion. Some areas, specifically where bubbles were trapped, will not etch very fast. Scraping those areas with a knife blade or scribe to scratch the copper will allow the copper to be etched a little faster, as shown in **Photo D.** Return the board to the etch and allow the etching process to continue. When wide trace patterns are used, overetching is usually not a problem.

During a cold environment, the etchant works very slowly. Adding a little heat to the process will speed it up considerably. Placing a small light bulb close to the etchant will warm the surrounding air. Also, placing a small cardboard box over the lamp and etchant tray will raise the temperature sufficiently to speed the etching process.

An alternate method for etching boards is to use a zipper-style plastic bag as an etch container. After the board is placed into the bag, about one inch of etchant is poured into the bag. Because some bags have a tendency to leak, it's a good idea to slide the first bag containing the board into another bag for drip protection.

After sealing the bag(s), the board is positioned flat with the copper down. Because the bag is transparent, the etching process can be observed through the bag. Handling the board and bag with care is OK, but excess handling should be avoided to prevent accidental spills caused by bag tears.

**After etching**

Upon completion, the board must be washed with tap water to remove the etchant. If the board is to be a single-sided board, then the nail polish can be removed with lacquer thinner or acetone. A small amount of solvent on a paper tissue works well as a wiper. An etched board with nail polish is shown in **Photo E**.

Following an inspection of the trace pattern for copper bridges and other possible minor defects, the board is ready for drilling (**Photo F**). Hole sizes are a personal choice, but drill sizes from #57 to #62 work well for most applications.

Solder-coating the copper is an optional process, but it does help reduce oxide development on the surface of the copper. The advantages of solder coating are better part solderability and uniform appearance, in addition to

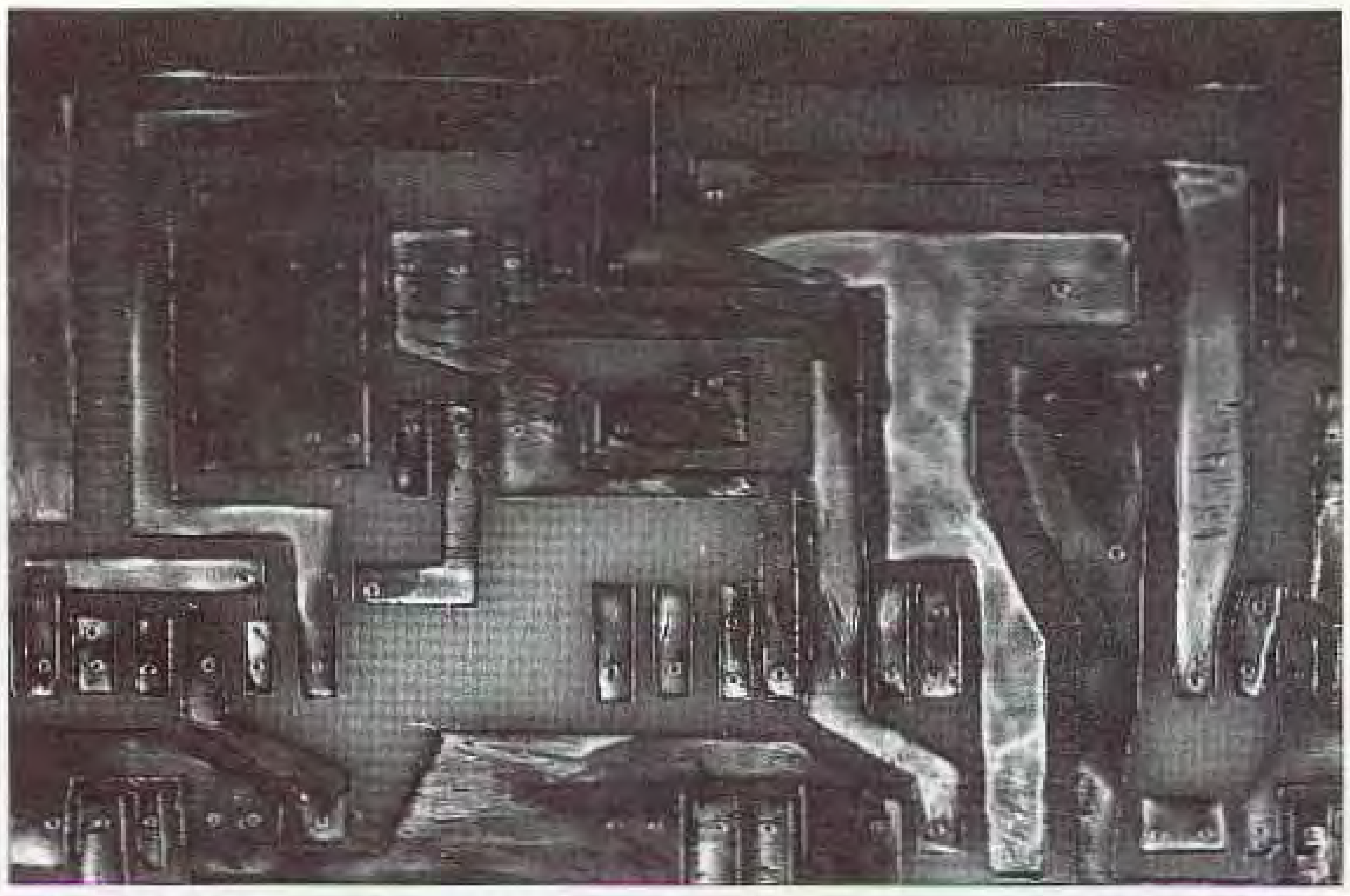

*Photo E. This is the etched board after being washed, but before the nail polish has been removed.*

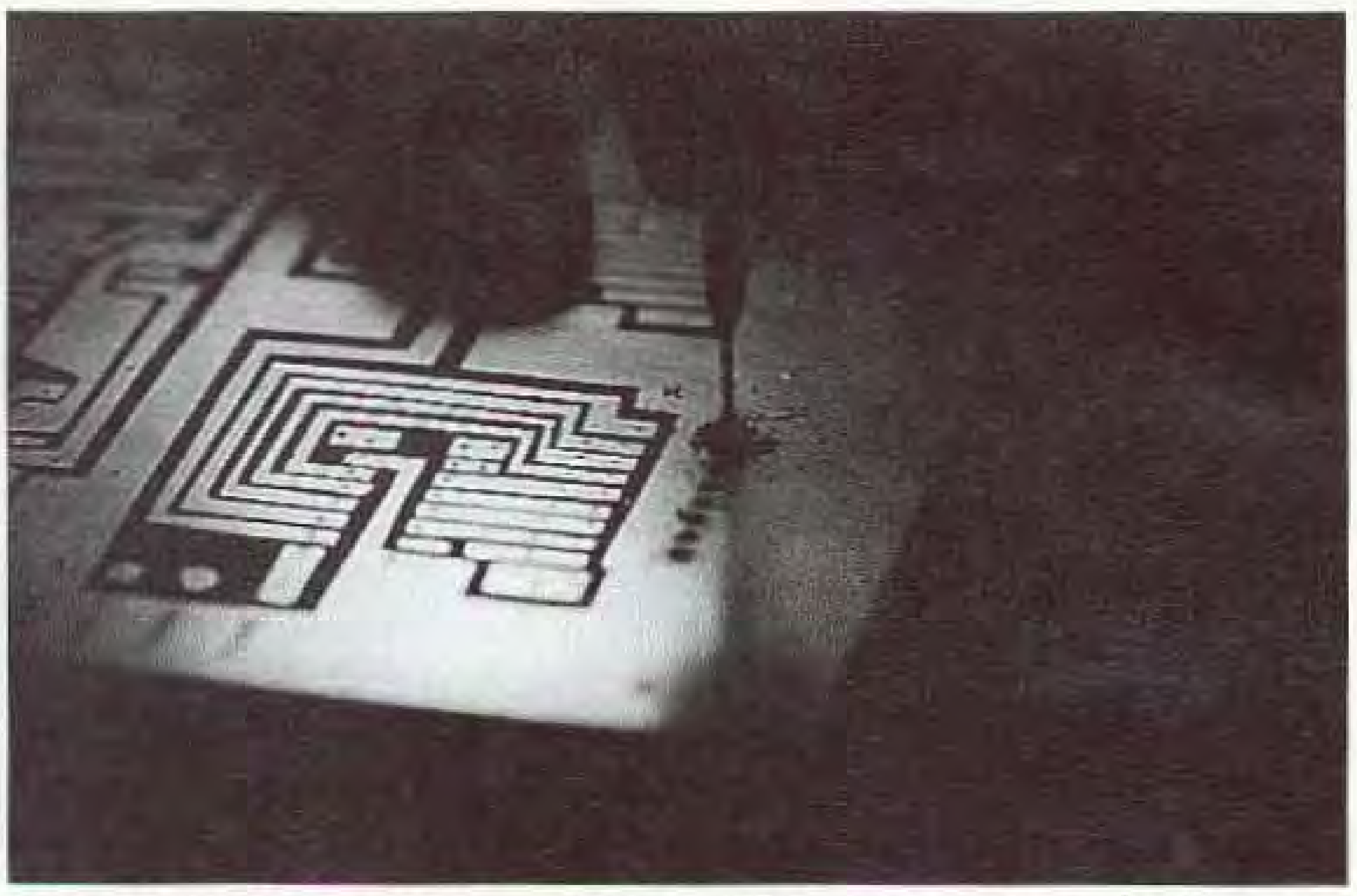

*Photo F. Drill sizes from #57 to #62 work well for drilling the holes.*

the reduction in copper oxide formation. The involved steps begin with cleaning the copper with fine steel wool followed with a solvent rinse. The objective is to remove all traces of nail polish, oxides, and oil.

The procedure for solder coating the board is as follows:

1) Coat the surface of the copper with a very thin coating of solder flux.

2) Place a small drop of solder on the tip of a 25–30 watt soldering iron.

3) Touch the solder to the copper and draw the iron along the copper. A solder trail will be left as the iron moves. Solder may be added as necessary to continue the process.

4) Continue the solder coating process until all of the copper is coated. It will be necessary to move the iron reasonably fast across the copper to reduce the possibility of burning the adhesive below the copper.

5) Clean the coated board with alcohol, lacquer thinner, or acetone to remove the flux.

### Double-sided boards

Making double-sided boards is a little more difficult than the process outlined above. However, the same steps are repeated except for drilling. Drilling is done from the bottom side (most complex trace pattern side) of the board.

During the etching process, the side opposite the pattern being etched must be protected from etching. Coating the "protected" side with nail polish or with a couple of layers of plastic spray is sufficient. After the first side is etched, it is sprayed with plastic or the exposed areas of copper are coated with nail polish to prevent further etching of those areas. Care must be taken so that the "protected" side is not scratched during handling. Coating the "protected" side a few minutes before etching is best.

Preparing the second side of the board follows the same steps as the first, but orientation of the trace pattern is a little tricky. It is best to have the majority of holes drilled before the second side is started, as the holes are

*Continued on page 22*

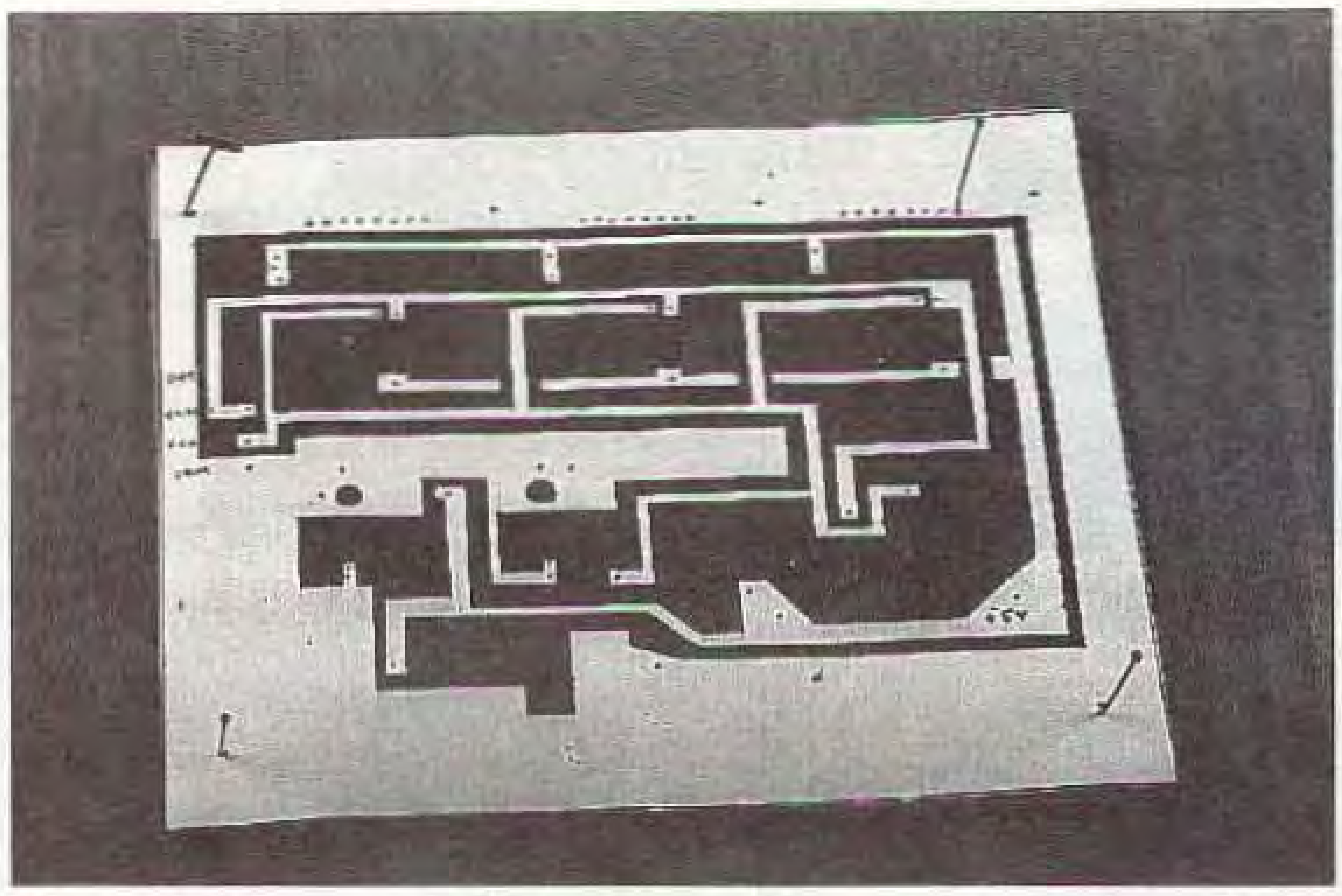

*Photo G. The second side paper mask is positioned prior to being cemented.*

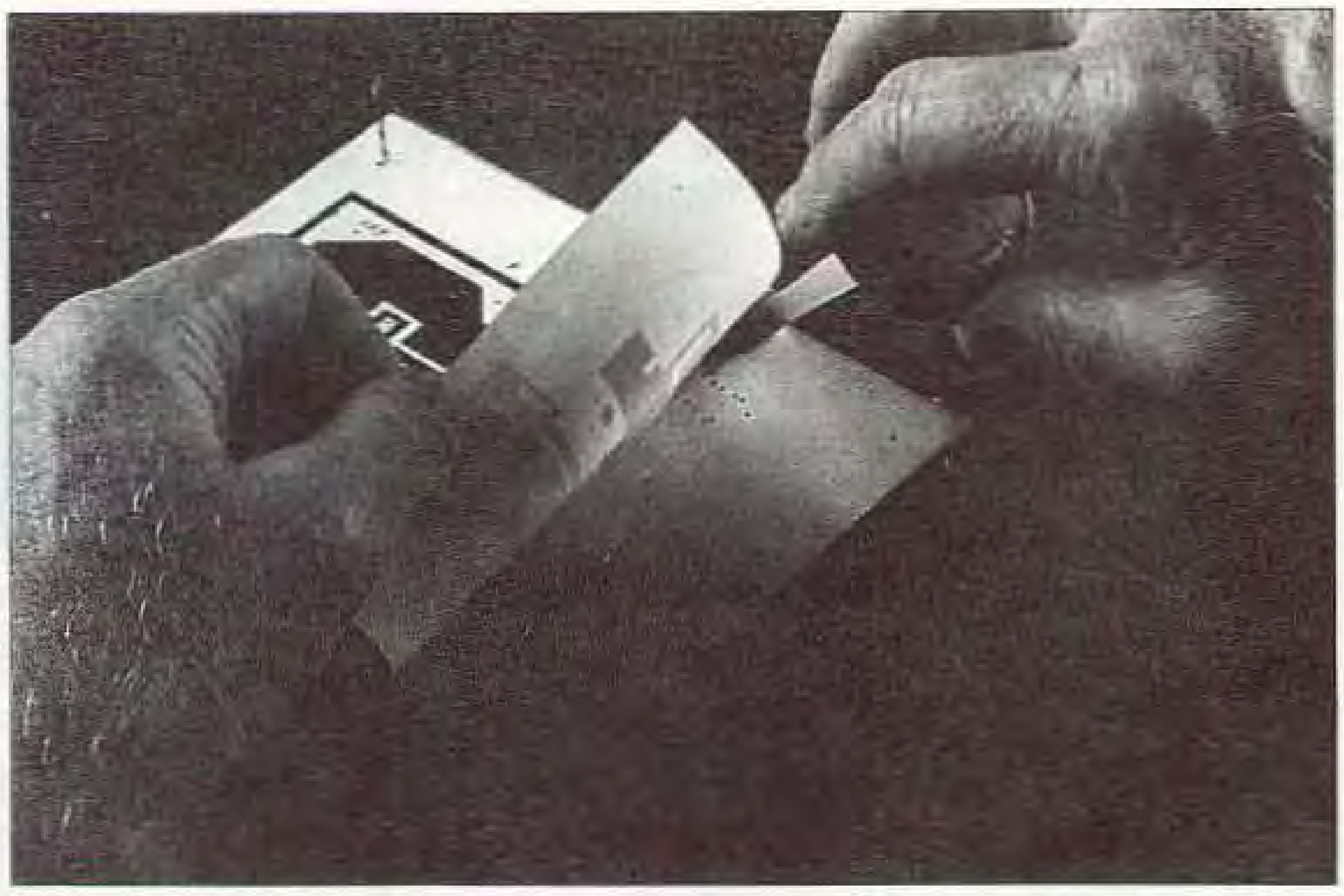

*Photo H. The first half of the paper mask has been aligned and cemented, while the second half is being cemented.*

## Inside Digital TV/VCR Tuners

*continued from page 21*

used as a guide for placing the paper mask.

In preparation for placing the paper mask onto the second side, the following steps are recommended:

1) Place the mask on the board without cement and orient the pattern to match the drilled holes.

2) For an alignment check, hold the board up to a light and observe the light passing through the holes. View the board from the topside pattern.

3) Push two or more straight pins through the corresponding mask "holes" into the board holes and note the proper trace orientation as shown in **Photo G**.

4) When satisfied that the orientation is correct, coat the copper and the paper mask with rubber cement.

5) With the straight pins pushed through the paper mask, use the pins as alignment guides while dropping the paper onto the copper as shown in **Photo H**.

6) Remove the pins and rub out the bubbles and excess cement.

With the mask in place, the trace pattern may be cut and removed, as was done for the first side. During the mask removal process, any misalignment of the trace pattern may be corrected as the process is performed, while using the mask lines as a cutting guide. Deviating from the drawn pattern may be necessary to correctly pick up the drilled holes.

Double-sided boards made commercially usually have plated-through holes. Unfortunately, the plating process required for making the plated holes may be outside of the reach of the average ham, so the data transmitter and receiver boards have been laid out for "Z"-wires as was shown in part 6, Fig. 5. The objective of the "Z"-wire is to connect the circuit trace on one side of the board to the trace on the other side. On the project boards, the solder pads have been extended away from the IC pin or transistor lead far enough for a wire to be passed through the board. Where "Z"-wires are used, component leads need to be soldered only to the bottom side of the board, as the "Z"-wire will transfer the connection between board sides.

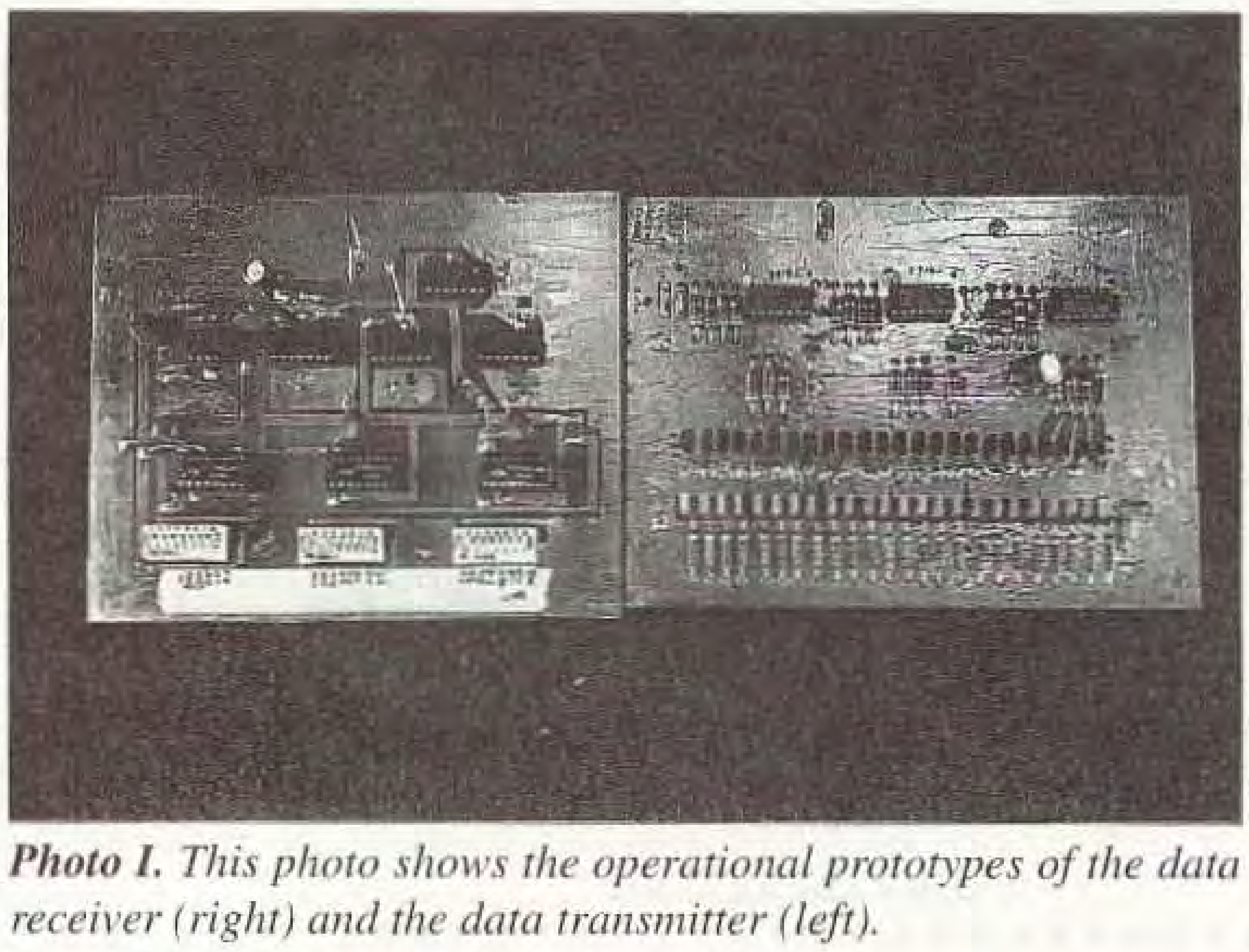

*Photo I. This photo shows the operational prototypes of the data receiver (right) and the data transmitter (left).*

Some bypass capacitors and the filter capacitors will require soldering onto the upper surface traces, because holes have not been provided. Placement of the capacitors is at the user's option. As an example, the 100-500 µF filter capacitor(s) may be placed anywhere on the board where the +5 volt trace runs close by the ground trace.

**Photo I** shows the completed prototype boards for the data transmitter and receiver. Trace pattern errors found in the prototypes have been corrected for the patterns provided herein.

I've been using the fingernail polish resist method for making printed circuit boards for a great many years, and can attest to the reliability of the process. I hope that you will see the merit in the simple process steps involved and will give it a try.

### Alternate boardmaking techniques

Because of the wide variety of printed circuit board processing techniques available, some alternate methods can be found on the Internet. The techniques indicated below utilize the plastic toner used in copy machines and laser printers as an etch-resist.

Here are four good URLs:

- [http://geocities.com/pdmtr]
- [http://www.techniks.com/press-n-peel.html]
- [http://www.qsl.net/ei9gq/pcb.html]
- [http://www.nordicdx.com/dxlab/makepcb2.html]. 73