Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

Making Your Own Printed Circuit Boards

To make a PC board, resist material is applied to a copper-clad bare PC board, which is then immersed into an acid etching bath to remove selected areas of copper. In a finished board, the conductive copper is formed into a pattern of conductors or "traces" that form the actual wiring of the circuit. The following sections describe how to make your own PC boards. A special section by Chuck Adams, K7QO, discusses making your own PC boards from artwork created on a laser printer.

PC BOARD MATERIALS AND SUPPLIES

PC BOARD STOCK

PC board stock consists of a sheet made from insulating material, usually glass epoxy or phenolic, coated with conductive copper. Copper-clad stock is manufactured with phenolic, FR-4 fiberglass and Teflon base materials in thicknesses up to 1/8 inch. The copper thickness varies. It is usually plated from 1 to 2 oz per square foot of bare stock.

RESISTS

Resist is a material that is applied to a PC board to prevent the acid etchant from eating away the copper on those areas of the board that are to be used as conductors. There are several different types of resist materials, both commercial and home brew. When resist is applied to those areas of the board that are to remain as copper traces, it "resists" the acid action of the etchant.

The PC board stock must be clean before any resist is applied. This is discussed later in the chapter. After you have applied resist, by whatever means, protect the board by handling it only at its edges. Do not let it get scraped. Etch the board as soon as possible, to minimize the likelihood of oxidation, moisture or oils contaminating the resist or bare board.

Resist Pens

Several electronics suppliers sell resist pens. Use a resist pen to draw PC-board artwork directly onto a bare board. Commercially available resist pens work well. Several types of permanent markers also function as resist, especially the "Sharpie" brand. They come in fine-point and regular sizes; keep two of each on hand.

Tape

To make a single PC board, Scotch, adhesive or masking tape, securely applied, makes a good resist. (Don't use drafting tape; its glue may be too weak to hold in the etching bath.) Apply the tape to the entire board, transfer the circuit pattern by means of carbon paper, then cut out

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

and remove the sections of tape where the copper is to be etched away. An X-Acto hobby knife is excellent for this purpose.

Paint

Some paints are good resists. Exterior enamel works well. Nail polish is also good, although it tends to dry quickly so you must work fast. Paint the pattern onto the copper surface of the board to be etched. Use an artist's brush to duplicate the PC board pattern onto bare PC-board stock. Tape a piece of carbon paper to the PC-board stock. Tape the PC-board pattern to the carbon paper. Trace over the original layout with a ballpoint pen. The carbon paper transfers the outline of the pattern onto the bare board. Fill in the outline with the resist paint. After paint has been applied, allow it to dry thoroughly before etching.

Rub-On Transfer

Several companies produce rub-on transfer material that can also be used as resist. Patterns are made with various width traces and for most components, including ICs. As the name implies, the pad or trace is positioned on the bare board and rubbed to adhere to the board.

ETCHANT

Etchant is an acid solution that is designed to remove the unwanted copper areas on PC-board stock, leaving other areas to function as conductors. Almost any strong acid bath can serve as an etchant, but some acids are too strong to be safe for general use. Two different etchants are commonly used to fabricate prototype PC boards: ammonium persulphate and ferric chloride. The section "Making PC Boards With Printed Artwork" describes a hydrogen peroxide-muriatic acid etchant. Ferric chloride is the most commonly used etchant and is usually sold ready-mixed. It is made from one part ferric chloride crystals and two parts water, by volume. No catalyst is required and it does not lose strength when stored.

Etchant solutions become exhausted as they are used. Keep a supply on hand. Dispose of the used solution safely; follow the instructions of your local environmental protection authority.

Most etchants work better if they are hot. A board that takes 45 minutes to etch at room temperature will take only a few minutes if the etchant is hot. Use a heat lamp to warm the etchant to the desired temperature. A darkroom thermometer is handy for monitoring the temperature of the bath.

Be careful! Do not heat your etchant above the recommended temperature, typically 160°F. If it gets too hot, it will probably damage the resist. Hot or boiling etchant is also a safety hazard.

Insert the board to be etched into the solution and agitate it continuously to keep fresh chemicals near the board surface. This speeds up the etching process. Normally, the circuit board should be placed in the bath with the copper side facing up.

After the etching process is completed, remove the board from the tray and wash it thoroughly with water. Use a household scrubbing pad, such as Scotchbrite**,** to rub off the resist. Using fine steel wool will also clean the board, but may leave fine steel particles behind.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

WARNING: Use a glass or other non-reactive container to hold etching chemicals. Most etchants will react with a metal container. Etchant is caustic and can burn eyes or skin easily. Use rubber gloves and wear old clothing, or a lab smock, when working with any chemicals. If you get some on your skin, wash it with soap and cold water. Wear safety goggles (the kind that fit snugly on your face) when working with any dangerous chemicals. Read the safety labels and follow them carefully. If you get etchant in your eyes, wash immediately with large amounts of cool water and seek immediate medical help. Even a small chemical burn on your eye can develop into a serious problem.

PLANNING A LAYOUT BY HAND

A later section of this chapter explains how to turn a schematic into a working circuit. It is not as simple as laying out the PC board just like the circuit is drawn on the schematic. Read that section before you design a PC board.

Traditionally, amateurs have laid out artwork for the PC board by hand. While this is still quite viable, it can become tedious and error-prone for more complex circuits. There are many PC board layout software packages available for free or at very low cost. For example, the author of the section on making PC boards from printed artwork, Chuck Adams, K7QO, uses *Eagle-5.3.0* layout software running under *Linux*. The free version can be used to lay out board sizes up to 3.25 by 4.0 inches with up to two layers. It has schematic capture, auto-router, and additional output formats to allow you to send files to a commercial PC board manufacturer if you want to do a club project requiring many boards to be made. There are a number of such packages listed in the PCB section of the **Computer-Aided Circuit Design** chapter.

ROUGH LAYOUT

Start by drawing a rough scale pictorial diagram of the layout. Draw the interconnecting leads to represent the traces that are needed on the board. Rearrange the layout as necessary to find an arrangement that completes all of the circuit traces with a minimum number of jumper-wire connections. In some cases, however, it is not possible to complete a design without at least a few jumpers.

LAYOUT

After you have completed a rough layout, redraw the physical layout on a grid. Graph paper works well for this. Use graph paper that has 10 lines per inch to draw artwork at 1:1 and estimate the distance halfway between lines for 0.05-inch spacing. Drafting templates are helpful in the layout stage. Local drafting-supply stores should be able to supply them. The templates usually come in either full-scale or twice normal size.

To lay out a double-sided board, ensure that the lines on both sides of the paper line up (hold the paper up to the light). You can then use each side of the paper for each side of the board. Remember that the side opposite the components (the “circuit side” or “foil side”) will have pin

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

arrangements that are reversed from the component top view! When using graph paper for a PC-board layout, include bolt holes, notches for wires and other mechanical considerations. Fit the circuit into and around them, maintaining clearance between parts.

Most through-hole IC pins are on 0.1-inch centers. Most modern components have leads on 0.1-inch centers. The rows of dual-inline-package (DIP) IC pins are spaced 0.3 or 0.4 inch. Measure the spacing for other components. Transfer the dimensions to the graph paper. It is useful to draw a schematic symbol of the component onto the layout.

The layout of a PCB for surface-mount devices is similar to that for through-hole components except that holes and pads are largely omitted. Traces end where parts are installed. SMT components have much smaller pin spacing than through-hole parts, so more care is required in the layout. If you lay out a board with computer software, the differences in laying out a board are largely transparent. One error that is frequently made by newcomers to SMT board layout is to make the board too compact and not leave room between components for probing.

Draw the traces and pads the way they will look. Using dots and lines is confusing. It's okay to connect more than one lead per pad, or run a lead through a pad, although using more than two creates a complicated layout. In that case, there may be problems with solder bridges that form short circuits. Traces can run under some components; it is possible to put two or three traces between 0.4-inch centers for a 1/4-W resistor, for example.

Leave power-supply and other dc paths for last. These can usually run just about anywhere, and jumper wires are fine for these noncritical paths. (This is not the case on high-speed digital boards and if EMI from the board is to be minimized.)

Do not use traces less than 0.010-inch (10 mil) wide. If 1-oz PC board stock is used, a 10-mil trace can safely carry up to 500 mA. To carry higher current, increase the width of the traces in proportion. (A trace should be 0.200 inch wide to carry 10 A, for example.) Allow 0.1 inch between traces for each kilovolt in the circuit.

When doing a double-sided board, use pads on both sides of the board to connect traces through the board. Home-brew PC boards do not use plated-through holes (a manufacturing technique that has copper and tin plating inside all of the holes to form electrical connections). Use a through hole and solder the associated component to both sides of the board. Make other through-hole connections with a small piece of bus wire providing the connection through the board; solder it on both sides. This serves the same purpose as the plated-through holes found in commercially manufactured boards.

After you have planned the physical design of the board, decide the best way to complete the design. For one or two simple boards, draw the design directly onto the board, using a resist pen, paint or rub-on resist materials. To transfer the design to the PC board, draw light, accurate pencil lines at 0.1- or 0.05-inch centers on the PC board. Draw both horizontal and vertical lines, forming a grid. You only need lines on one side. For single-sided boards, use this grid to transfer the layout directly onto the board surface. To make drilling easier, use a center punch to punch the centers of holes accurately. Do this before applying the resist so the grid is visible.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

When drawing a pad with plenty of room around it, use a pad about 0.05 to 0.1 inch in diameter. For ICs, or other close quarters, make the pad as small as 0.03 inch or so. A "ring" that is too narrow invites soldering problems; the copper may delaminate from the heat. Pads need not be round. It's okay to shave one or more edges if necessary, to allow a trace to pass nearby.

Draw the traces next. A drafting triangle can help. It should be spaced about 0.1 inch above the table, to avoid smudging the artwork. Use a 9-inch or larger triangle, with a rubber grommet taped to each corner (to hold it off the table). Select a sturdy triangle that doesn't bend easily.

Align the triangle with the grid lines by eye and make straight, even traces similar to the layout drawing. The triangle can help with angled lines, too. Practice on a few pieces of scrap board.

Make sure that the resist adheres well to the PC board. Most problems can be seen by eye; there can be weak areas or bare spots. If necessary, touch up problems with additional resist. If the board is not clean the resist will not adhere properly. If necessary, remove the resist, clean the board and start from the beginning.

Discard troublesome pens. Resist pens dry out quickly. Keep a few on hand, switch back and forth and put the cap back on each for a bit to give the pen a chance to recover.

Once all of the artwork on the board is drawn, check it against the original artwork. It is easy to leave out a trace. It is not easy to put copper back after a board is etched. In a pinch, replace the missing trace with a small wire.

Applied resist takes about an hour to dry at room temperature. Fifteen minutes in a 200°F oven is also adequate.

Special techniques are used to make double-sided PC boards. See the section on double-sided boards for a description.

MAKING A PC BOARD

Several techniques can be used to make PC boards. They usually start with a PC-board "pattern" or artwork. All of the techniques have one thing in common: this pattern needs to be transferred to the copper surface of the PC board. Unwanted copper is then removed by chemical or mechanical means. Most variations in PC-board manufacturing technique involve differences in resist or etchant materials or techniques.

No matter what technique you use, you should determine the required size of the PC board, and then cut the board to size. Trimming off excess PC-board material can be difficult after the components are installed.

The bare (unetched) PC-board stock should be clean and dry before any resist is applied. (This is not necessary if you are using stock that has been treated with pre-sensitized photoresist.) Wear rubber gloves when working with the stock to avoid getting fingerprints on the copper surface. Clean the board with soap and water, and then scrub the board with #000 steel wool. Rinse the board thoroughly then dry it with a clean, lint-free cloth. Keep the board clean and free of fingerprints or foreign substances throughout the entire manufacturing process.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

NO-ETCH PC BOARDS

The most straightforward way to make very simple PC boards is to mechanically remove the unwanted copper. Use a grinding tool, such as the Moto-Tool manufactured by the Dremel Company (available at most hardware or hobby stores). Another technique is to score the copper with a strong, sharp knife, then remove unwanted copper by heating it with a soldering iron and lifting it off with a knife while it is still hot. This technique requires some practice and is not very accurate. It often fails with thin traces, so use it only for simple designs.

PHOTOGRAPHIC PROCESS

Many magazine articles feature printed-circuit layouts. Some of these patterns are difficult to duplicate accurately by hand. A photographic process is the most efficient way to transfer a layout from a magazine page to a circuit board.

The resist ink, tape or dry-transfer processes can be time consuming and tedious for very complex circuit boards. As an alternative, consider the photo process. Not only does the accuracy improve, you need not trace the circuit pattern yourself!

A copper board coated with a light-sensitive chemical is at the heart of the photographic process. In a sense, this board becomes your photographic film.

Make a contact print of the desired pattern by transferring the printed-circuit artwork to special copy film. This film is attached to the copper side of the board and both are exposed to intense light. The areas of the board that are exposed to the light — those areas not shielded by the black portions of the artwork — undergo a chemical change. This creates a transparent image of the artwork on the copper surface.

Figure 1 — PC-board materials are available from several sources. This kit includes layout materials, etchant, and PC board stock.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

Develop the PC board, using techniques and chemicals specified by the manufacturer. After the board is developed, etch it to remove the copper from all areas of the board that were exposed to the light. The result is a PC board that looks like it was made in a factory.

Materials and supplies for all types of PC-board manufacturing are available through a variety of electronic distributors. If you're looking for printed-circuit board kits (see **Figure 1)**, chemicals, tools and other materials, review the articles on PC board construction in the ARRL Technical Information Service at **www.arrl.org/tis** in the Radio Technology area under Circuit Construction.

IRON-ON RESIST

An artwork positive is made using a standard photocopier or laser printer. A clothes iron transfers the printed resist pattern to the bare PC board. The technique for transferring the pattern to the blank PC board is described in the section "Making PC Boards With Printed Artwork".

The key to making high quality boards with the photocopy techniques is to be good at retouching the transferred resist. Fortunately, the problems are usually easy to retouch, if you have a bit of patience. A resist pen does a good job of reinforcing any spotty areas in large areas of copper.

DOUBLE-SIDED PC BOARDS

All of the examples used to describe the above techniques were single-sided PC boards, with traces on one side of the board and either a bare board or a ground plane on the other side. PC boards can also have patterns etched onto both sides, or even have multiple layers. Most home-construction projects use single-sided boards, although some kit builders supply double-sided boards. Multilayer boards are rare in ham construction. One method for making double-sided boards is described in the final section of this article, "Double-Sided PC Boards — by Hand!"

TIN PLATING

Most commercial PC boards are tin-plated, to make them easier to solder. Commercial tin-plating techniques require electroplating equipment not readily available to the home constructor. Immersion tin plating solutions can deposit a thin layer of tin onto a copper PC board. Using them is easy; put some of the solution into a plastic container and immerse the board in the solution for a few minutes. The chemical action of the tin-plating solution replaces some of the copper on the board with tin. The result looks nearly as good as a commercially made board. Agitate the board or solution from time to time. When the tinning is complete, take the board out of the solution and rinse it for five minutes under running water. If you don't remove all of the residue, solder may not adhere well to the surface. Immersion tin plating solution is available from electronics vendors.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

DRILLING A PC BOARD

After you make a PC board using one of the above techniques, you need to drill holes in the board for the components. Use a drill press, or at least improvise one. Boards can be drilled entirely "free hand" with a hand-held drill but the potential for error is great. A drill press or a small Moto-Tool in an accessory drill press makes the job a lot easier. A single-sided board should be drilled after it is etched; the easiest way to do a double-sided board is to do it before the resist is applied.

To drill in straight lines, build a small movable guide for the drill press so you can slide one edge of the board against it and line up all of the holes on one grid line at a time. See **Figure 2.** This is similar to the "rip fence" set up by most woodworkers to make accurate and repeatable cuts with a table saw.

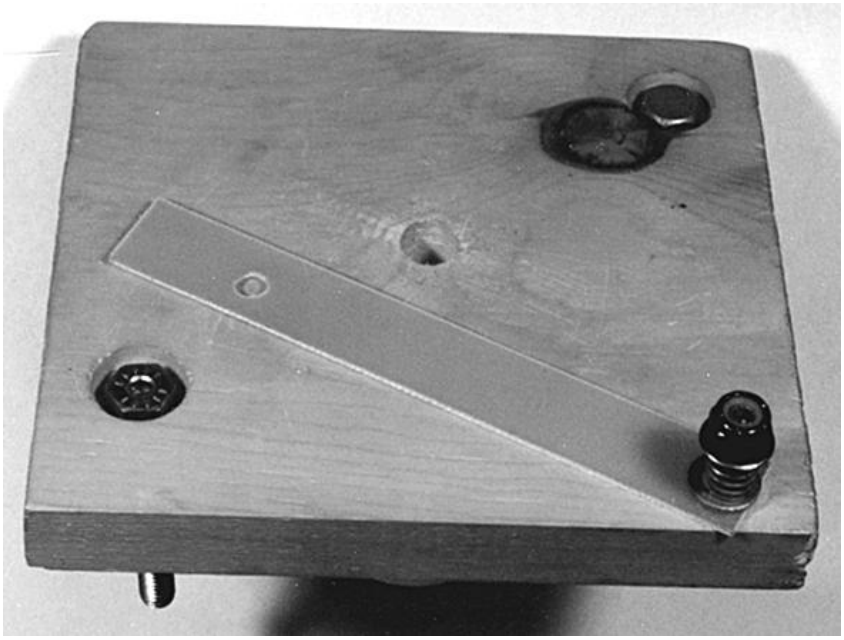

Figure 2 — This home-built drill fence makes it easy to drill PC-board holes in straight rows.

The drill-bit sizes available in hardware stores are too big for PC boards. You can use high-speed steel bits, but glass epoxy stock tends to dull these after a few hundred holes. (When your drill bit becomes worn, it makes a little "hill" around each drilled hole, as the worn bit pushes and pulls the copper rather than drilling it.) A PC-board drill bit, available from many electronic suppliers, will last for thousands of holes! If you are doing a lot of boards, it is clearly worth the investment. Small drill bits are usually ordered by number. Here are some useful numbers and their sizes:

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

Number	*Diameter (in)*
68	0.0310
65	0.0350
62	0.0380
60	0.0400

Use high RPM and light pressure to make good holes. Safety goggles and good lighting are a must! Count the holes on both the board and your layout drawing to ensure that none are missed. Use a larger-size drill bit, lightly spun between your fingers, to remove any burrs. Don't use too much pressure; remove only the burr.

Making PC Boards with Printed Artwork

Chuck, K7QO, is a long time homebrewer. He operates what he builds and builds what he operates. Since he was first licensed 50 years ago, he has built rigs using nearly every conceivable construction technique available. He has recently developed this technique for fabricating printed circuit boards at home with easily available materials and a laser printer.

MATERIALS

- 1/4-cup and 1/2-cup plastic measuring cups (do NOT use these cups to prepare food)
- An ordinary clothes iron with a plain metallic ironing surface rather than one with a nonstick coating. The nonstick coating will wear off quickly with use in making printed circuit boards.
- Pyrex dish with 3-cup or 750-ml capacity with plastic/rubber snap-on cover.
- Rubber gloves, heavy duty and acid proof.
- Safety goggles
- Surgical gloves, disposable, for handling boards and chemicals.
- Small plastic tongs for lifting board from harsh chemicals.
- Disposable plastic food forks for moving board around in harsh chemicals.
- A larger glass container with screw-on cover for storage of used etchant.
- Bottle of 2% hydrogen peroxide.
- Clear enamel spray.
- Plastic scrubbing sponge such as ScotchBrite used on kitchen items. (Using steel wool leaves small iron filings behind.)
- Bottle of Tarn-X for cleaning copper.
- Large bag of large cotton balls. This for using both the Tarn-X to remove corrosion from the PC board material and for using acetone to remove the toner after the board is etched.
- Acetone, which can usually be found in the paint department of hardware stores or home improvement stores. Acetone is a hazardous chemical and should be used in a ventilated area and

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

not breathed. Handle it with rubber gloves as it can cause skin irritation. Keep it tightly capped when not use.

- Muriatic acid. This can be found in the swimming pool section of home improvement stores or in the outdoor section as it is used for cleaning bricks, concrete, and pools.
- Glossy photo-printing paper. Commercial papers that work include Domtar Microprint (32 lb, 112 brightness, 8.5 × 11 in, 20 g/m2) or Staples color laser photo supreme (SKU # 651611). Or you can use a commercial resist paper as discussed earlier in the section.

PC BOARD MATERIAL

One ounce or heavier copper-clad board on FR-4 substrate is best for homebrew PCBs. This is available surplus from a number of channels. The 0.060-inch thickness is best for all-around use.

While Chuck uses a band saw to cut the material into the smaller sizes, a shear, a paper cutter or a hacksaw also work well. The FR-4 substrate material is quite abrasive and will dull blades quickly. Use all safety precautions in using any cutting tool. Use a sanding block immediately after cutting to remove sharp edges along both top and bottom surfaces and the substrate.

INEXPENSIVE AND EFFECTIVE ETCHANT

The secret to etching PC boards is the etchant. Chuck uses a mixture of hydrogen peroxide and muriatic acid. Contrary to what you might think, the acid is not doing the etching. It is the hydrogen peroxide. The oxygen in the peroxide oxidizes the copper and then dissolves into the acidic solution. While ferric chloride is often recommended for etching printed circuit boards, this mixture works as well or better and it is less expensive to make.

The proper ratio of the etching solution is 2:1 for 2 parts hydrogen peroxide to 1 part muriatic acid. To make the solution, while wearing the heavy rubber gloves and safety goggles, first pour 1/2 cup of hydrogen peroxide into the Pyrex container. The order is important. The peroxide goes into the Pyrex container first, then slowly add in 1/4 cup of muriatic acid. This mixture will etch about two double-sided 3.25 × 3.0-inch boards.

Only make a single batch of the hydrogen peroxide/muriatic acid mixture. The mixture will not hold its etching strength for very long, so be frugal and don't make it unless you are going to use it. The bottle of hydrogen peroxide should always be tightly capped and stored in a cool dark place as it readily oxidizes and loses its strength.

TRANSFERRING THE LAYOUT TO THE BOARD

Design your layout. (Don't forget to reserve a small blank area to put your name, call sign and date!) Print the layout on the glossy paper. The printed layout should be a mirror of the final board. The software will do this automatically.

Clean the board with the plastic abrasive sponge. It should be shiny. Put some blank paper under the board to protect the underlying surface.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

Carefully place the glossy image with the printed circuit board image on the board where you want it, face down with the toner in contact with the board surface. With the iron on its hottest setting and fully warmed up, carefully iron the image onto the board. Do not use the steam setting. Use heavy pressure; 20 lbs or more is adequate.

Use the pointy end of the iron to apply pressure everywhere on the paper. Don't scrimp here. Try to think about where all the traces are underneath the paper and aim for them all and especially the smallest ones. Patience is key. Be thorough and detailed. This makes a difference in how well your final board will turn out. Don't touch the paper when you are done as it will be hot. Take a break while the paper cools.

After it has cooled, put very hot water in a container and let the board and paper soak for about 15 to 30 minutes. Then come back and carefully peel the paper away from the board. Don't force it. It is easiest to do it from all four edges carefully. Don't touch the toner area. Handle the board by the edges.

Don't try to get all the paper the first time. Just avoid pulling the toner traces off the board. You should not see any toner on the paper being pulled from the board if you are doing this right.

With the heat used to apply the toner to the board, the board will darken and become oxidized. Don't worry about it—that part of the board will be etched away.

After you do the first paper removal, put the board back into some fresh *hot* water and let it soak some 15 minutes more. Then, using only the fingers and your skin, rub the rest of the paper off as much as possible. Don't use your fingernails or anything to help. Doing this under running lukewarm water from the faucet helps. It is not important that you get every bit of paper off the toner. It will look gray or white in some sections because the paper fibers are absorbed or stuck within the toner itself. Some people use an old toothbrush for this.

ETCHING THE BOARD

Take the board and place it in the Pyrex dish with the peroxide/muriatic acid mix. You'll know you are doing this correctly and the mix is right when the board starts to darken almost immediately. This is the copper oxidizing. The clear liquid will start to turn a light green with dissolved copper in short order. Stay away from the fumes and stand upwind from any draft and do this in a well ventilated area.

Use the two plastic forks to gently push the board back and forth in the mixture, being extra careful not to touch the toner. If it comes lose, then that area of the copper will be etched away.

Move the board around until all the copper is etched away from the board except the areas covered by the toner. Don't go off and leave and forget about the etching process as it will ruin the board. Take the board out using the tongs and wash it under running water to stop the etching process.

FINISHING THE BOARD

To remove the toner after the etching process, use acetone and cotton balls, while wearing surgical gloves. A well-ventilated area is a must for this. Do it outdoors if possible. This

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

procedure takes some practice, as sometimes the toner smears and sticks to the board a little if not enough acetone is used and kept on the surface. Don't overdo it, but practice on a few boards to get the hang of it.

After you get the toner off and rinse the board, use cotton balls and the Tarn-X to clean the copper surface.

Now the surface is exposed to the nasty atmosphere of the Earth. Protect it by very lightly coating the surface with the clear enamel in a clean, dust-free, well ventilated area also. This coating protects the surfaces and it also acts as a solder mask.

Chuck uses a 0.7 millimeter silicon carbide drill bit in a Dremel Moto-Tool mounted in a drill press to drill holes for through hole parts. A drill press is necessary as the drill bit is very small and easily damaged if you do this by hand. The 0.7 mm drill will work for most of the holes. One can go to a larger sized bit for parts that have larger leads.

You are now ready to stuff the board with your parts. Congratulations — you have just designed, fabricated and built a printed circuit board!

Double-Sided PC Boards — by Hand!

Forget those nightmares about expensive photoresists that didn't work; forget that business of fifty bucks a board! You don't need computer-aided design to make a double-sided PC board; just improve on the basics, and keep it simple. Anyone can make low-cost double-sided boards with traces down to 0.020 inch, with perfect front-to-back hole registration.

To make a double-sided board, drill the holes before applying the resist artwork; that is the only way to assure good front-to-back registration. The artwork on both sides can then be properly positioned to the holes. PC-board drilling was discussed earlier in the text.

After you have drilled the board, clean its surface thoroughly. After that, wear clean rubber or cotton gloves to keep it clean. One fingerprint can really mess up the application of resist or the etchant.

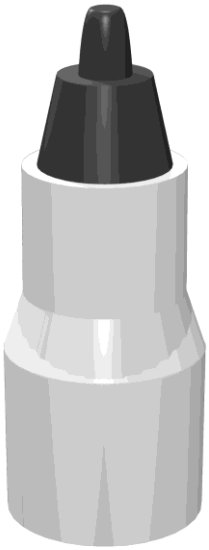

Figure 3 — Make a permanent marker into a specialized PC-board drawing tool. Simply press the marker point into a drilled hole to form a modified point as shown. More pressure produces a wider shoulder that makes larger pads on the PC board.

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

Tape the board to your work surface, making sure it can't move around. Transfer the artwork from your layout grid to the PC board, drawing by hand with a resist pen.

Allot enough time to finish at least one side of the artwork in one sitting. Start with the pads. To make a handy pad-drawing tool, press the tip of a regular-size Sharpie indelible ink felt-tip marker into one of the drilled PC-board holes. This "smooshes" the tip into the shape of the hole, leaving a flat shoulder to draw the pad. See **Figure 3**. The diameter of the pad is determined by how hard the pen is pressed; pressing too hard forms a pad that is way too large for most applications. Practice on scrap board first. Use this modified pen to fill in all the holes and draw the pads at the same time. Use an unmodified resist pen to draw all of the traces and to touch up any voids or weak areas in the pads. For the rest of the drawing, the procedure described for single-sided boards applies to double-sided boards, too.

After the resist is applied to the first side, carefully draw the second side. Inspect the board thoroughly; you may have scratched or smudged the first side while you were drawing the second.

Etching a double-sided board is not much different than etching a single-sided board, except that you must ensure that the etchant is able to reach both sides of the board. If you dunk the board in and out of the etchant solution, both sides are exposed to the etchant. If you use a tray, put some spacers on the bottom and rest the board on the spacers. (The spacers must be put on the board edges, not where you want to actually etch.) This ensures that etchant gets to both sides. If you use this method, turn the board over once or twice during the process. — *Dave Reynolds, KE7QF*

OR PHOTO-ETCHED

You can also make double-sided boards at home without drawing the layout by hand. This procedure can't produce results to match the finest professionally made double-sided boards, but it can make boards that are good enough for many moderately complex projects.

Start with the same sort of artwork used for single-sided boards, but leave a margin for taping at one edge. It is critical that the patterns for the two sides are accurately sized. The chief limiting factor in this technique is the requirement that matching pads on the two sides are positioned correctly. Not only must the two sides match each other, but they must also be the correct size for the parts in the project. Slight reproduction errors can accumulate to major problems in the length of a 40-pin DIP IC. One good tool to achieve this requirement is a photocopy machine that can make reductions and enlargements in 1% steps. Perform a few experiments to arrive at settings that yield accurately sized patterns.

Choose two holes at opposite corners of the etching patterns. Tape one of the two patterns to one side of the PC board. Choose some small wire and a drill bit that closely matches the wire diameter. For example, #20 AWG enameled wire is a close match for a #62 or a #65 drill, depending on the thickness of the wire's enamel coating. Drill through the pattern and the board at the two chosen holes. Drill the chosen holes through the second pattern. Place two pieces of the wire through the PC board and slide the second pattern down these wire "pins" to locate the pattern on the board. Tape the second pattern in position and remove the pins. From this point

Copyright © 2013 American Radio Relay League, Inc. All rights reserved.

on, expose and process each side of the board as if it were a single-sided board, but take care when exposing each side to keep the reverse side protected from light.