

# FEATURE ARTICLE

Paul Verhage

## A Class Act

### Making Your Own PCBs

School's in session as Paul shows us what it takes to design and etch your own printed circuit boards. If you're looking for a way to avoid the expense of using a custom board house, the "fab lab" approach may be just the solution for you.



The best way to build an electronics project is to solder the electronic components to a printed circuit board (PCB). After being soldered, the completed circuit will be stable and the component connections less likely to break. Before soldering, you must either purchase or make your own PCB. A mass-produced PCB, or board, is inexpensive, but limited to the circuits that other people have designed and etched. If you want a unique circuit, you'll have to design and etch your own.

Board houses will help you with this endeavor, but they usually require a minimum order of \$100. For a single PCB, \$100 is not reasonable. It's more expensive if you make a design error.

To get around this cost, many people use a photocopier to make transferable toner resist masks. With this method, you copy the PCB pattern onto a transparency using either a laser printer or photocopier. A hot iron transfers the toner from the transparency to a clean copper clad board.

Afterward, you can etch the board with ferric chloride. The layer of toner prevents the copper from being etched away. My results with toner masks

were not as satisfying as having a professionally manufactured PCB. Today though, I make my own high-quality PCBs at an affordable cost to facilitate my hobby of near space exploration.

With the lab I developed, I can go from concept to completed project during one weekend. And the best part is that a single-sided, 4" x 6", etched and drilled PCB costs me less than \$4. The process is clean and simple. With a little care and practice, I can make double-sided boards, too.

The cost of building my PCB fabrication lab was about \$300 excluding the PC and laser printer. I have no other costs except for purchasing inexpensive, sensitized copper clad boards, laser printer transparency sheets, and two inexpensive chemicals (developer and etchant). Having a PCB fab lab is like having a photographic darkroom. In this article, I'll tell you how to make and use a similar lab.

#### THE PARTS

You'll purchase most of the parts ready to use. Only the UV table and etcher will be built or modified from locally available parts. The items you'll need are listed in Table 1. Ares Lite is a Windows 95-based PCB CAD program. You'll use it to lay out the parts, traces, and ground planes (highly recommended) of your PCB.

Ares Lite prints UV-ready PCB masks on transparency sheets with your laser printer. The homemade UV table exposes sensitized copper clad boards to your PCB mask (see Photo 1). Constructing it will be an afternoon project. Because you don't want to shake a tray of etchant for 30 min., modify the cheapest foot massager you can find to do it.

To drill holes into the etched PCB, use a Dremel hand tool and drill press (unless you find a less expensive drill press). Carbide drill bits drill lead holes into the PCB. Don't use steel drill bits; the carbide bits cut a cleaner hole and stay sharp longer.

You need two inexpensive plastic photographic trays to develop and etch the exposed copper clad board. To keep the etching process clean, use freezer bags when etching the copper clad. Use GLAD-LOCK zipper bags.

Ares Lite PCB design software	\$65
UV table (homemade)	\$70
Dremel hand tool	\$55
Dremel drill press	\$50
No. 67 carbide drill bit	\$4
Two photographic developing trays	\$10
PCB etcher (a modified foot massager)	\$25
Bug light and clamp-on reflector	\$15
Two measuring cups and spoons	\$6
One package, five sensitized 4" x 6" single-sided, copper clad (Kepro, DF1-465M)	\$16
2 lbs of developer (Kepro, DFD-12G)	\$14
4 lbs of etchant (Kepro, SP-1G)	\$21
One package of one-quart freezer bags (Glad Bags)	\$4
One box of 50 laser printer transparency sheets (Staples, Apollo CG7060)	\$10

**Table 1**—Here is everything you need. The consumable products, which determine your cost for making PCBs, are included.

Only open the bag of sensitized copper clad in the subdued light of a yellow bug light. Because I don't want to leave a bug light in the socket in my bathroom (my current PCB fab lab), I use a portable light fixture. To measure the proper amount of water and chemicals, stop by the kitchen section of your local department store and get a teaspoon and tablespoon. Also get one 1/2 cup and 1/3 cup measuring cups or a beaker.

Now that I've introduced you to the required parts, let's start building the UV table. Later, you'll make a foot massager into the etcher. While building the UV table and etcher, purchase your chemicals and sensitized boards from Kepro. The most time-consuming part of making a lab is learning how to use Ares Lite. However, it's not a difficult program.

## THE UV TABLE

Building the UV table is straightforward and will be accomplished using hand tools. According to Kepro, UV light sources that work with its boards include UV fluorescent tubes, photoflood bulbs (I assume these are the same as tanning bulbs), or the sun. I wasn't pleased with tanning bulbs and I want to keep the light source consistent, so I don't use sunlight. Instead, I use two fluorescent UV lamps. These lamps are white and do not have black coating like black lights. (You know you have the correct lamps if they burn your retinas when you stare at them!)

Next, pick up two 18" fluorescent lamp kits (available at stores like Wal-Mart). These lamp kits are used as extension lighting and are not permanently wired into household wiring. I

use the Under Cabinet Light Fixture from Lights of America. While at the store, pick up Stic Mount Cord Clamps, the kind used for tacking down extension cords or outdoor Christmas lights. Get two metal hinges (2.5" wide), a bag of drywall screws (1 5/8" long), a packet of four nail-in feet, a 3" mending brace, white paint, and a paintbrush. Pick up the cheapest six-outlet power strip you can find. Finally, buy a yard of 1/4" foam rubber and a plastic pull handle. These miscellaneous parts should cost less than \$30.

At a glass store, purchase a 1/8" thick sheet of glass, cut to 19.5" by 9.75". I assume all window glass is sufficiently transparent to UV.

To make the UV table, pick up a piece of 1" x 4" pine, 8' long. Have the board cut into four pieces; two should be 20.5" long and the other two should be 10" long. And, purchase a sheet of 3/4" plywood and have two pieces cut out of it that measure 20.5" x 11". You'll probably have to purchase a 4' x 4' sheet of plywood.

The last pieces of wood you need to buy are for mounting the glass into the table. At a hobby store, find basswood strips that measure 1/2" x 1/8". These come in 24" long strips, so purchase three of them. Also buy a 3' x 1' sheet of 0.010" thick polystyrene plastic.

## BUILDING THE UV TABLE

Now it's time to build the UV table. Use wood glue and drywall screws to attach the 1" x 4" pine boards to one piece of the 3/4" plywood to form a box (without a lid). The box measures 20.5" x 11" and is 4.25" tall. The plywood is the bottom of the UV table and the pine boards are its sides. On

the 3/4" wide edges of the frame (at the top), glue the basswood strips. Position them on the outside edge of the frame to create a recess for holding the glass. This recess also keeps the sharp edges of the glass hidden from fingers that may brush against them.

Using an Exacto saw, cut a small notch into one side of the box that's large enough to pass the fluorescent lamp cords through. You may be tempted to drill a hole in the side of the frame, however, a hole large enough to pass the plugs of the lamp cords through would be about 3/4" in diameter. A hole that large would leak UV and may be a hazard to eyes. Give the box and the remaining piece of plywood a thick coat of white paint. I assume the white paint will reflect UV the best. Finish by nailing feet onto the bottom of the box.

Now it's time to attach the lid. Align the lid on top of the box. While securely holding it in place, screw in the hinges on the long side of the box. Be careful and make sure the box will open after you screw on the hinges.

Open the box and mount the fluorescent lamp fixtures inside the table, centered on the bottom. There's a drilling guide on the lamp package for determining where to drill the holes. Use the Stic Mount Cord Clamps to mount the lamp cords securely to the inside bottom of the table so they can't pop up and cast shadows on your PCB.

Mount the power strip on the outside of the table. You want the power strip on the side with the notch for the power cords. Use a few of the drywall screws to hold the power strip to the side of the UV table. Be sure to mount the power strip so it's flush with one edge of the UV table's side.

To keep the power strip from sliding off its mounting screws, make a stop with the mending brace. Screw the brace to a corner of the table on the side that the power strip wants to slide off. Bolt the brace flush against the power strip.

With the mending brace flush against the power strip, the power strip can't slide off its mounting bolts. If you ever need to remove the power strip, you'll need to remove the mending brace first.

It's time to get back to the electrical work. Plug the lamp cords into the power strip. Using the power strip, you'll have a single switch to turn on/off both UV lamps. At this point, turn on the lamps and the power strip. When you know they work properly, switch off only the power strip (after you glue in the glass, you won't be able to reach the lamps). Next, remove the lamp covers and fluorescent tubes. Replace the original fluorescent tubes with the UV lamps, but don't replace the lamp covers.

Mount the pull handle to the center of the lid using the screws that came with the handle. Be sure to screw the handle tightly to the lid. Before proceeding, thoroughly clean the glass, getting rid of any dirt or fingerprints. Place the glass into the recess at the top of the box and use a little silicone glue in the corners to hold it down.

While the silicone glue is setting, cut a piece of 1/4" foam rubber the same size as the lid. Glue the 1/4" foam rubber to the inside of the lid using silicone glue. Finish the inside of the lid by gluing the 0.010" thick piece of styrene plastic over the face of the thin foam rubber. The plastic sheet and foam rubber create even pressure on the copper clad board and resist mask.

After the glass is in place, use silicone glue and attach a steel ruler or square to the top of the glass. The ruler will align the copper clad with the transparency. A metal square can be used to align doubled-sided boards.

Your UV table is now complete. The total cost should be around \$70.

## MAKING THE ETCHER

I started etching boards by agitating trays by hand. This made me realize that I had better things to do with my time than to shake trays for 30 min. So, I purchased a foot massager on sale that could agitate boards for me. At first, I tried placing the etchant and board into a sealed plastic freezer bag and dropping it into the foot massager filled with warm water. This resulted in unevenly etched boards.

Currently, I use a thin plywood platform built on top of the foot massager to shake a photographic tray filled with warm water, a freezer bag

containing etchant, and an exposed copper clad board (see Photo 2). Eventually I want to find a thin box that I can set inside the massager when filled with warm water. Inside the box would be etchant and the board(s) standing vertically on edge. The water surrounding the thin box would keep the etchant warm, speeding up the etching process.

Let's get back to my current design. The etcher's wooden platform (table) is made from 1/8" airplane plywood purchased at the hobby shop. I glued a basswood strip to the bottom of one side of the plywood to keep it level when it is placed on top of the foot massager. I recommend using small metal screws to attach the wooden table to the top rim of the massager, although I use a set of dowels.

Glue down 1/4" basswood strips to form a rectangular barrier that holds the photographic tray. This allows you to set the etching tray on the table without permanently attaching the tray to the table. This barrier is high enough to stop the tray from bouncing all over the place. The two nice features of this etcher are that it's cheap and etches PCBs unsupervised.

## SUPPLIES FOR ETCHING A PCB

To develop and etch exposed copper clad boards, pick up two inexpensive plastic photographic trays (one for developing and the other for etching), a plastic Tupperware juice container (for storing the dry etchant powder), and a set of measuring cups and spoons. Use a bug light to see what you are doing while the sensitized boards are outside their lightproof bag. (A photographic supply store may have a portable light fixture that you can use instead of the bug light.)

Also, buy a cellulose sponge to finish your boards. Cut the sponge into 1" cubes. I don't recommend sponges containing germicide, because I don't know how the resist will chemically respond to the germicide.

Purchase your PCB supplies from Kepro. To save money with experimental designs, you can use a 1/16"-thick, single-sided, 1-ounce, copper clad, flame-retardant, polyester board. They come in packets of five for

\$16.50. Each board has a layer of sensitized resist and is covered with a clear plastic sheet. This sheet is transparent to UV and protects the resist from being scratched while handling. Kepro also sells epoxy composite and glass epoxy-based boards. Polyester boards work well for my projects, but I would use a glass epoxy board for a PCB that I planned to sell.

Now that you have purchased all your equipment, constructed the lab, and learned to use Ares Lite, you're ready to start making your first printed circuit board. There is an 11-step process for fabricating a PCB. First, design the board in software, double-checking your design. Second, print an inverted-color copy of the artwork onto a transparency sheet with a laser printer. Then, trim and tape the transparency to the glass in the UV table.

Fourth, place the board copper side down on the transparency and expose it to UV for 60 s. For the next step, let the board's resist harden for 15 min. After that, develop the board for 60 s. Seventh, place the developed board into a freezer bag filled with etchant and seal the bag.

The next step is to agitate the board for about 30 min. Ninth, clean the board and let it dry. Next, drill holes in the board. And finally, the eleventh step is trimming the PCB.

Don't let these 11 steps scare you away from making a PCB; they're not difficult. Now, let's look at these steps in greater detail.

## ARES LITE

I use Ares Lite as my PCB CAD program. This inexpensive Windows 95 program is easy to use. It has related software that does schematic captures and circuit emulations just like PSpice. But, I have been able to design my board without schematic capture, autorouting, or testing.

Ares Lite comes with a large number of component pinouts. You can develop any that are missing from the database. A background grid exists in the workspace allowing you to snap down components at 50-mils intervals. The program lays down copper traces with widths from 5 to 1000 mils. After

laying down the copper traces, you can create ground planes to fill in the open spaces on the board.

In most circuit board designs, ground planes are recommended. I use ground planes to reduce the amount of copper that needs to be etched from the board. Ares Lite allows you to design double-sided PCBs (the top copper is displayed in red and the bottom copper is blue). The software allows you to designate 14 additional layers, four mechanical layers, and a component side.

### CREATING THE RESIST MASK

Start designing your PCB by loading the components from the database. At anytime during board development, you can load additional components. Change the orientation of loaded components by rotating or flipping them. After properly oriented, drag the component to the workspace and place it anywhere. At anytime while laying out the board, you can duplicate or drag components to a new location. You can even rotate components in the



Photo 1—Check out the UV table with mask. You can see the UV lamps beneath the glass.

workspace without deleting them.

Ares Lite lets you lay down your board's copper traces whenever you wish. To lay down traces, first select the trace tool, then the side of the board you want the trace to be on (top or bottom copper), and then the width of the trace. On the workspace, make a series of clicks to define the path of the trace. And then to end the trace, right click the mouse.

While laying out components or the traces, you can change pad sizes if necessary. After components and traces are in place, create ground planes by filling open spaces with cop-

per. The software allows you to create polygon-shaped ground planes. Before finishing, add hardware mounting holes to the board.

You should finish by doing something that most techies don't do, document your work! There is a text feature in Ares Lite that enables you to draw rectangles, polygons, circles, and write text. This is especially handy when you go back to that board you were designing last week.

When finished designing your product, print the resist mask and any notes regarding component placement. To print the resist mask, select Output, then select Print. Select only the copper layer you want to print (in most cases, that's the bottom layer). Be sure to select the option to invert the colors. Usually I print a paper copy review. As long as you don't scratch the final transparency copy, you can use it to expose unlimited copper clads.

Next, print a copy of the component layer and your notes on paper. Don't select a copper layer when print-

Year 2000  
in Review

<http://www.chipcenter.com>

## What do the leading silicon vendors know about BIOS?

They know the right BIOS is key to the success of embedded designs—and configurability is key to the right BIOS.

That's why AMD, Intel, and STMicro ship General Software's Embedded BIOS pre-installed on their embedded platform evaluation boards.

With over 400 configuration options, Embedded BIOS offers the advanced configurability you need to run your custom target environment without editing the core BIOS source code.

Contact us today for detailed information and a free sample BIOS binary for your standard reference design.

### Embedded BIOS™

#### ADAPTATION KIT:

Full source code automatically configured with over 400 parameters using BIOStart™ expert system

#### CORE BIOS FEATURES:

ROM/RAM/Flash disks, Setup system, console re-direction, manufacturing mode, WinCE loader, configurable PCI, integrated debugger, modular callouts to chipset, board, and CPU-level modules

#### CHIPSETS:

ALI—Aladdin V, Finali  
AMD—186, SC300, SC400, SC520  
INTEL—386EX, 430HX/TX, 440BX, 810, 840  
NSC—Geode GxM, GxLv  
ServerWorks—ServerSet III HE/LE  
STMicroelectronics—STPC family

#### IDEAL FOR:

Windows 95/98/CE/NT  
Embedded, Linux, DOS, and all x86-based operating systems



[www.gensw.com](http://www.gensw.com) • [sales@gensw.com](mailto:sales@gensw.com) • 800-850-5755 • 425-454-5755

© 2000 General Software, Inc. All rights reserved.



ing notes. Also remember to deselect the box for reversing colors.

Carefully cut the transparency so it fits inside the UV table. Make sure the toner side is up by looking from a glancing angle. If the toner side is up, the face of the transparency will be dull over the black areas. If it's shiny, you have it upside down.

Use masking tape to hold the transparency on the glass. Use as little masking tape as possible, because it will act as a mask increasing the amount of copper that gets etched. Don't use transparent tape because it is too difficult to remove from the glass and mask. I usually trim and tape the resist mask up against the steel ruler. When taping down the transparency mask, avoid touching the glass, because you don't want to leave fingerprints where you may be exposing the board. Occasionally give the glass a light brushing to remove dust.

## EXPOSING THE BOARD

The lights must be off and the yellow bug light turned on. When you take out a board from the copper clad package, tape the bag closed to protect the other boards from light. Turn the board copper side down and place it over the mask. Do not play with the plastic covering on the copper side of the board because you risk leaving wrinkles in the plastic protector that can show up on the copper traces.



**Photo 2**—The developed PCB is in a freezer bag filled with etchant. The bag is in a tray of warm water. The foot massager will be turned on next.

Place the board so that no traces on the mask are left uncovered. You don't want to shoot only half of the board. You can tape the copper clad down, but I haven't found this necessary. After positioning the board over the mask, close the lid of the UV table. The lid protects your eyes from the UV lamp, so it's important to close the lid every time you expose a board. Don't shake the copper clad around when closing the lid. I like to place a little pressure on the lid to make sure the copper clad is flat against the mask and glass.

Next, turn on the UV lamps by using the power strip switch. After the lamps start, wait 60 s for the proper exposure and then shut off the lamps. You've now exposed your soon-to-be-completed PCB.

After exposing the board to UV, let the resist harden for 15 min. inside the UV table (lamps off). If you need to shoot more boards, put the exposed

board into a lightproof bag and shoot the next board on the UV table.

## DEVELOPING EXPOSED BOARDS

Two of the goals while making a PCB are to keep the cost of consumable products low and keep the process clean. The method I use will let you shoot single boards without mess or waste.

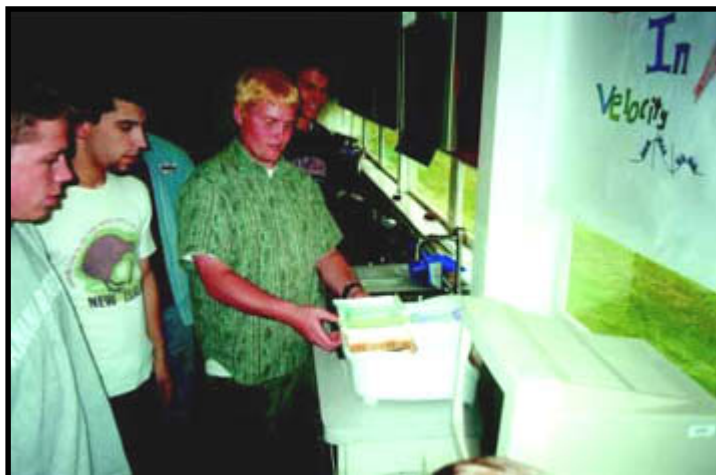
While the exposed resist hardens, get the developer and etchant ready. The Kepro developer is just washing soda. It comes as a dry

powder in a 2-lbs plastic bottle. When mixed, it will develop 120<sup>2</sup> of board (720 4" × 6" single-sided boards). To mix only enough developer for one board, pour 1 tsp (5 ml) of dry developer into a photographic tray. Do not add water yet.

To mix just the amount of etchant needed for a 4" × 6" single-sided board, pour 1 tsp plus 1 tbsp (20 ml total) of etchant into a Glad freezer bag. I use sodium persulfate (SP) as my etchant. I haven't had to use the accelerator that comes with the SP etchant. Don't use the developer or etchant to complete a second board, use fresh chemicals for each board.

After the resist hardens, pour 2/3 cups (150 ml) of hot water into the developer tray and dissolve the washing soda. Still with the lights off and the yellow bug light on, remove the exposed board from the UV table. Look at the exposed surface of the board and you'll see the faint traces of the latent image on the resist. After developing, the image will have much more contrast.

Peel off the plastic protective cover from the copper side and drop the board into the photographic tray filled with developer (keep the board face up). Spend 1 min. agitating the tray. The resist layer will turn purple underneath the bug light (go K-State!). After 1 min. of shaking, lift the board on its edge while it's still over the tray of developer. Gently wipe the copper surface with a small sponge dipped in developer. The copper to be etched



**Photo 3**—At Nampa High School, Idaho, my electronics students design a PCB for a 555 IC LED flasher. Here, Adam Rainwater starts the etcher with fellow students (from left) Josh Roach, Bennett Yankey, and Steve Hyer.

will come clean, leaving bright copper that contrasts with the purple dyed resist. Notice the resist has a slick feeling to it whereas the copper doesn't. Once, the resist completely dissolved off the board. I imagine this happened because I left the board in the developer too long.

It will be obvious when the copper traces are clear of resist, but you may lose a board learning this. The critical part of making a PCB is the development process; every other step has a little slop built into it. After developing the board, rinse it in warm water to stop the developing. Do not throw out the tray of developer yet, because you'll use it to remove the remaining resist after etching the board.

If you find any defects in the resist, let the board dry before repairing it. Use a hair dryer or Chem Wipes to speed up the drying, but don't use tissue paper because they will leave lint on the damp board. The boards usually come out of the developer fine. However, if you need to repair a notched trace, use a permanent fine-tip



**Photo 4**—From left to right, here's an etched PCB, PCB with developed resist before etching, and a mask on a transparency.

marker. Lightly tap the pen onto the trace and leave a spot of permanent ink. You should do this several times, but let the ink dry between markings.

### ETCHING THE BOARD

Etch the board soon after developing it. In fact, if you're happy with the traces, you can start etching as soon as you rinse off the developer. Being a cautious fellow, I still leave the lights off while beginning this process. The etchant is a clear liquid that turns an attractive blue when etching copper.

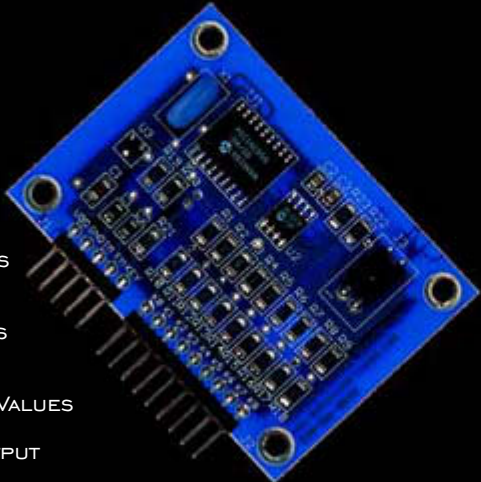
This etchant is not nearly as foul nor messy as ferric chloride. Kepro sells the etchant in a powder, which will store longer than when in solution. Two and a half pounds (the smallest amount Kepro sells) of dry etchant powder makes 1 gal of etchant. One gallon is enough for dozens of boards. I transfer the dry etchant into a Tupperware container and store it in a location safe from my house rabbits. You should do the same.

Pour 1/3 to 1/2 cup (100 ml) of hot water into the freezer bag containing the etchant. Knead the powder in the bag to dissolve it. After dissolving the etchant, place the developed board into the freezer bag and seal it. Double-check the seal. Then, set the freezer bag aside and pour hot water into the etching tray.


Place the freezer bag in the tray with the copper side of the board facing down. Wrap most of the excess bag around the board, leaving the excess bag on top. After placing the tray on top of the etcher, turn on the foot massager (see Photo 3). I like to set the

# MEMKey

A VERSATILE KEYPAD INTERFACE



- READS UP TO 20 KEYS (4X5 MATRIX)
- PROGRAMMABLE ROWS & COLUMNS
- PROGRAMMABLE KEY VALUES
- PC/AT OR SERIAL OUTPUT
- AUTOMATIC OR POLLED KEY REPORTING
- PROGRAMMABLE DEBOUNCE & TYPEMATIC TIMES
- 64 BYTES OF USER ACCESSIBLE EEPROM



**Solutions<sup>3</sup>**

SOLUTIONS CUBED  
(530) 891-8045 PHONE  
WWW.SOLUTIONS-CUBED.COM

foot massager on a towel so it can't move around and to collect water drops. It takes less than 30 min. to etch a 4" × 6" board with SP. About halfway through the etching, change the water in the tray, replacing it with hot water.

You can etch two boards simultaneously. Place each board in its own bag filled with etchant and drop one on top of the other in the tray. Midway through etching, when you change the water, switch the position of the bags. I haven't tried a third board, but it would probably work with a deeper tray. After you start etching, you can turn on the room lights.

After etching the board, remove it from the freezer bag and thoroughly rinse it. Place the etched board back into the developer tray (the developer will be cold) and let it sit for a few minutes. You'll start to see the purple resist curdle and peel off the board. One minute later, you can use a sponge to finish wiping off the resist. You now have a clean and etched PCB (see Photo 4). Rinse the board and prop it up on its side to dry.

## AFTER ETCHING

Drill your PCB with a carbide drill bit chucked into a Dremel mounted to a small Dremel drill press. You may want to tape a piece of paper over the base of the Dremel drill stand so that the PCB surface stays clean. I typically use a no. 67 drill bit, but it's too small for 0.025" male headers. I experimented with miniature steel drill bits purchased from the local hardware store and discovered that they leave a pile of melted dust. I also discovered that carbide drill bits in surplus magazines do well. The bits I purchase from Kepro are only \$4 a piece, so I usually use those.

The problem with carbide drill bits is that they are brittle. Although they remain sharp for a long time, they cannot tolerate sideways force; a jiggle while drilling is enough to snap them. With care, you will be able to drill hundreds (possibly thousands) of holes with carbides, but always keep extra bits on hand.

The next step is to cut and trim your PCB. Seldom will your design be exactly 4" × 6", so you'll have to cut off

the excess board. Use a razor cutter mounted to a handle to trim the PCB. Using a straight edge, score the PCB by cutting it several times with the razor. After scoring the board, carefully snap the board along the score. Use sandpaper to smooth and round the edge. I lay sandpaper on a flat surface and run the board edge across it.

Some people tin their boards, but it's unnecessary. The Kepro boards are clean and I've easily soldered components to them. If you're going to store the board before soldering components, you may want to tin the board first, because the copper will oxidize over time.

Tinning compounds are available in a dry powder. To use them, mix the tinning powder with water and soak the board. Some of the tin in the solution will replace some of the copper on the board. After the tinning process, remove the board from the solution and let it dry.

## THE FINAL PRODUCT

After you finish drilling and trimming (and possibly tinning) your PCB, it's time to solder the components. Now, use your printout of the component layer and notes. Chances are you'll never remember the placement or values of components, so your notes should be complete. I cannot stress enough the importance of complete documentation.

It also pays to get a good solder station and tools. Your boards will be excellent quality and deserve clean and trim solder joints. Although soldering is outside the scope of this article, I'd like to make a few recommendations. Always wipe the solder tip on a damp sponge and immediately tin it. It's a good idea to get yourself a solder sucker to clean up excess solder or to remove solder from pads. I use a set of sharp cutters to clean up my solder joints and exposed wires. Be careful, I have managed to cut a copper trace off a board with these cutters.

And, get a lead bender and use it regularly. It will bend resistor leads to the correct length the first time, saving strain on the leads. I use masking tape to hold components in place when soldering on the other side of the

board. Solder two diagonal pins of an IC sockets first. Then, press on the socket as you reheat the two soldered pins. This allows you to seat the socket completely before soldering the remaining pins.

I hope you'll make your own PCB fab lab. It's a great hobby that saves money while allowing you to create a quality PCB. It's difficult to overemphasize the quality of these PCBs. It's almost automatic to end up with a board as good as those you purchase from a board house. The only thing lacking is plated-through holes. I'll bet Kepro has those for sale. 📧

*Author's Note: I thank Steve Kelly of Kansas State University who inspired this project. Steve introduced me to Kepro and how to design and make top-notch PCBs. Also, I'm interested in making a training video based on this article. Please contact me if you're in the Boise area and can help make the video.*

*Paul Verhage is a science and electronics teacher at Nampa High School in Nampa, Idaho. His interests include astronomy, meteorology, digital electronics, and outer space. To facilitate his hobbies, Paul designs his own PCBs. Recent PCB designs include a Lego robot controller, a digital display for a hat, and a controller for nearcraft (a high altitude, model satellite). He can be contacted at pverhage@sd131.k12.id.us*

## SOURCES

### PCB boards, chemicals, and supplies

Kepro Circuit Systems, Inc.  
(314) 426-2600  
(800) 325-3878  
www.kepro.com

### Ares Lite

Labcenter of Electronics  
+44 (0) 1756 753440  
Fax: +44 (0) 1756 752857  
www.labcenter.co.uk

### Handtool and drill press

Dremel  
(800) 437-3635  
(262) 554-1390  
www.dremel.com