HARDWARE HACKER

Optical reprint sources
Hacker pc breakthough
Clock and doorbell chip
Faking double-sided boards
Printed circuit resource list

PC-board breakthrough

DON LANCASTER

OUR STUPENDOUSLY MAJOR NEW breakthrough for this month is a brand new way of doing hacker printed-circuit boards that I'll call the *direct toner method*. Believe it or not, all you need is an iron and your favorite word processor.

This new process is ridiculously faster, simpler, and cheaper than any of the old ways. Since it's so new, we sure could use your personal help in further testing and debugging.

But first, let's review some of the older ways of making printed-circuit boards. We might start off by going over some...

Circuit-board fundamentals

Printed circuit boards first became popular in the early 1950's because of their overwhelming advantages over point-to-point wiring. The PC wiring pattern was always the same, virtually eliminating wiring errors. Stray inductance and capacitance were much lower and far more uniform. And the manufacturing could be totally automated. Production times became much shorter, and labor costs dropped sharply. So did size and weight.

A printed-circuit board often will consist of an insulating *substrate* that has one or more layers of conducting patterns placed on or in it. Figure 1 shows some popular forms of printed-circuit boards.

You will find three main substrate materials in use today. They include phenolic, FR-4 (or G-10) glass epoxy, and CEM-1 composite epoxies. While phenolic is the cheapest, it does chip and shatter easily, and should be heated before punching or drilling. It is often used for single-sided layouts in toys, appliances, and any other high-volume applications. For us hackers, phenolic is nearly useless.

Glass epoxy is pretty near the same stuff that a fiberglass boat is made of. It has great electrical and mechanical properties, and is nearly ideal for any double-sided and multilayer boards. Hacker disadvantages are that glass epoxy costs more and dulls drills at an amazing rate. Carbide drills are just about mandatory for all but the shortest of production runs.

The CEM-1 material has only a pair of fiberglass layers impregnated into an epoxy body. Because it's cheaper and easier to drill than glass epoxy, it's a good choice for hacker use. It also drills and punches well. Glass-epoxy boards are well suited for all but the most precise and exacting needs. They even come in a wide variety of colors.

The simplest variation is a *single-sided* board. The substrate is most often 1/16th of an inch thick, and has a single layer of copper foil laminated to one surface only. Two popular thicknesses of copper are used. *One-ounce* copper is around 0.00135 inches thick; *two-ounce* copper is double that, or around 0.00270 inches thick.

Thus, one-ounce copper is a tad over one mil thick, and two-ounce copper is somewhat over two mils thick. Two-ounce copper is normally reserved for higher-current uses or where extreme reliability is needed.

On a traditional single-sided circuit board, most of the components get mounted on the bare side of the board, giving us a *component side* and a *foil side* to work with. That allows a dip, a reflow, or wave soldering of all the parts at once. The components tend to pull the foil toward the substrate, rather than trying to peel the foil from the board.

Single-sided boards limit both your minimum size and how much you can connect where, unless you go to an unacceptable number of interconnecting jumpers. Because of that, most modern boards are *double-sided*, and have foil on both surfaces. While the most common means of routing connections between the two board sides is with *plated-through holes*, hacker alternatives are eyelets, wire tabs, the component leads by themselves, or individual socket pins. *Mill-Max* is a leading source of low-cost socket pins, and *Stimpson* is a good eyelet source.

A double-sided plate-through

NEED HELP?

Phone or write your **Hardware Hacker** questions directly to:
Don Lancaster
Synergetics
Box 809
Thatcher, AZ 85552
(602) 428-4073

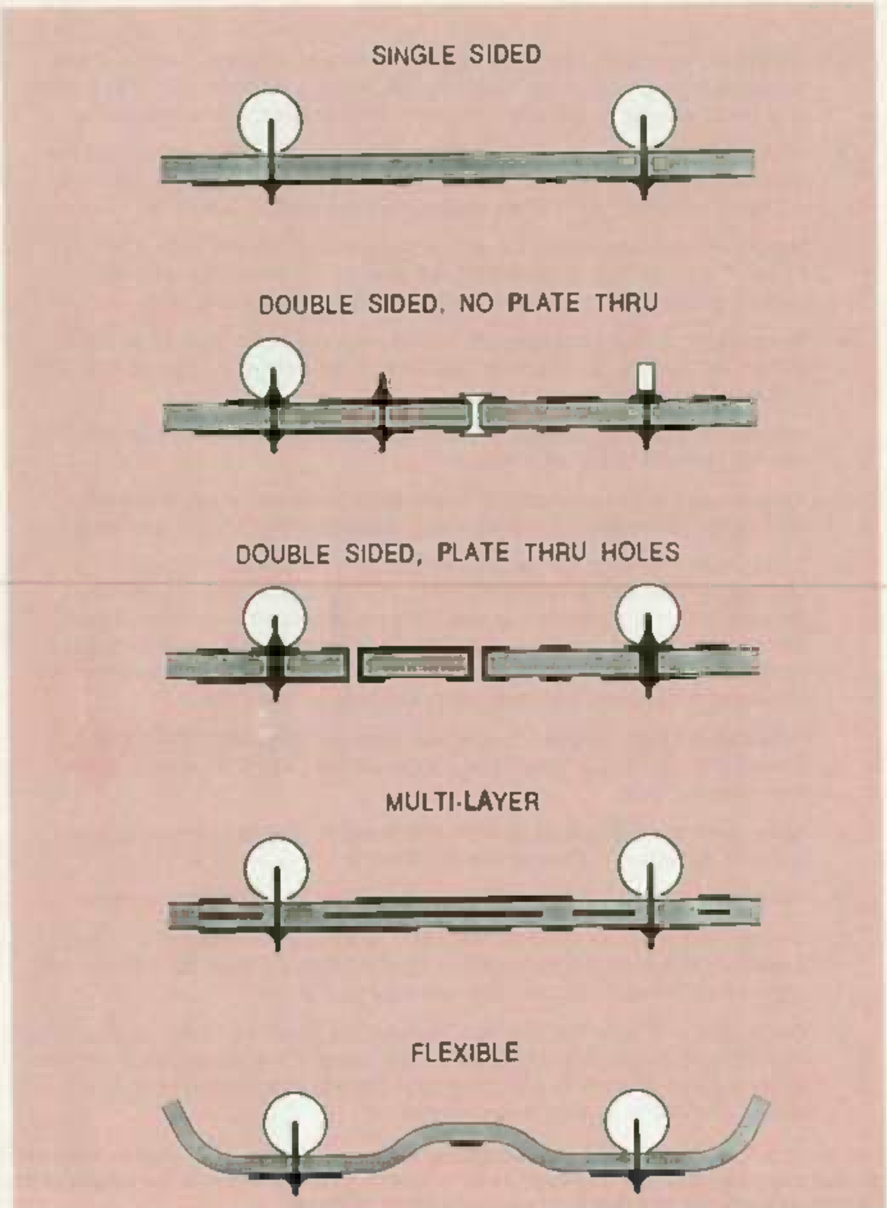


FIG. 1—SEVERAL POPULAR TYPES of printed-circuit boards. Note that a double-sided plate-through board can be hacker-faked by using component leads, eyelets, wire tabs, or low-cost individual pin sockets.

setup is beyond what most hackers would care to attempt. The tanks and such alone can set you back the better part of $10,000.00. Nasty chemicals are involved that are hard to get in small quantities. Worse yet, it takes a long time and involves several dozen steps, all of which have to function perfectly to ever get any product out at the far end. You could farm out a lot of prototype boards at $30 to $60 before you could ever justify the investment.

Fortunately, the latest of the *surface-mount* technology components tend to greatly minimize both the number of holes and the need for plate-through. So jumpers, eyelets, or individual socket pins are not really all that bad an alternative for your prototype boards.

The next step beyond double-layer boards are *multi-layer* boards, where circuitry is placed *inside* the substrate, as well as on both surfaces. Typically, there will be four layers. Your horizontal runs will dominate on the top surface, followed by a lower power-supply plane, a ground plane that is lower still, and the vertical runs that dominate the bottom surface.

As you might guess, all four-layer boards are quite expensive and are extremely hard to modify, but they do offer superior shielding and extreme component densities. Multi-layer PC boards as dense as 24 layers have been built. Quite often, the multi-layer PC board will be the most expensive part of an electronic system.

Flexible boards are also becoming popular. They are often thinner and use a *Kapton* substrate. Uses include mounting connectors, and for highly dense or unusual packaging. *Rogers Corp* is a leading source of flexible PC-board supplies.

Creating a printed-circuit

There's a number of good ways to create a final printed-circuit board. In general, those methods that put new conductors on an insulating substrate are *additive*; those that remove unwanted conductors from unneeded areas are *subtractive*. Very often, both additive and subtractive techniques will be used in combination.

Four of the traditional board-production techniques include *direct, mechanical, silk screen*, and *photographic*.

In the direct method, an etch-resistant pattern is applied by hand to the printed-circuit stock. Most any paint, lacquer, instant transfer, or ink will work, as will the tape and dots intended for initial layout work. So does a fingerprint or spilled root beer. *Bishop Graphics* is a leading supplier of PC tape and dots, and *Datak* is one source for instant-transfer products. There are also some rubber-stamping layout aides being offered.

Actually, the direct method is more hassle than it is worth, and ends up just about totally useless. Some problems here are pattern alignment, preventing fingerprints, tape lifting, a lack of uniformity, and too many defects.

The mechanical methods physically remove unwanted copper, usually by routing, special drills, or by milling. They're another concept that looks much better on paper than in the real world. Several specialized systems are usable for the mechanical PC layouts. Invariably, they are both laughingly and obscenely overpriced.

The silk-screen PC method is quite simple and is widely used commercially, especially for single-sided boards of fairly low tolerances. The process is exactly the same as silk screening a T-shirt or a greeting card. Oversize artwork is created, usually at a 2:1 or sometimes a 4:1 scale. A litho negative gets shot from the artwork, which in turn creates a photo master for the screen. High-resolution screens are used, often in a 20XX density.

To print a board, etch-resistant ink is placed on the screen, and a squeegee is used to force the ink through the open portions of the screen. The board is then etched to remove all copper that is *not* covered by the inked image.

The advantages of the silk-screen method are that it is cheap, fast, and relatively low tech. One disadvantage is that the $30 setup charge per screen gets out of hand when you want only a single prototype board. A second is the inability to do very fine lines or precisely aligned work.

Ulano is one major source of silk-screen films. The screens themselves are available from such

1. Always do your layouts double sized (2X) on a blue gridded mylar sheet, available from any drafting supply house. Always work on a light box. Use only "real" printed circuit tape and dots. *Bishop Graphics* is one source.
2. Watch which side you tape from. Pin one of an integrated circuit is at the *lower left* when viewed from the top as shown in the data book. Pin one will be at the *lower right* when etched from the bottom board foil.
3. Never cut your tape with an X-acto knife! Instead, lay the knife down flat and pull the free end of the tape back against the blade. Always firmly mash the tape in place after routing. A teaspoon is ideal for this.
4. Never do your own photography! A litho negative costs only $3 at a jiffy printer, ad agency, or lithographers. This is the *only way* to get the proper precision and density.
4. Never coat your own boards! Always use commercially precoated dry film boards, such as those from *Kepro*.
5. Always use dry film photoresist, rather than spray-on or liquid coated KPR types. Otherwise, pinholes, dust, and uniformity will eat you alive.
6. If you must coat your own board, incredible cleanliness is essential. Thoroughyly scour the board with ultra fine steel wool and a chlorine activated (*Comet*) cleanser at least three times, spending not less than two minutes per square inch of board. Dry promptly and avoid all finger prints. If possible, follow up with a chemical copper cleaner. Then etch for a few seconds, rinse four times, dry, and use immediately.
7. Note that properly cleaned copper will allow an unbroken film of water to flow over it, and that it will *not* be copper colored at all. Instead, it will be a uniform hot pink.
8. Never print through the negative base! Always have the photo emulsion in direct contact with the dry film photoresist.
9. Never use a ferric chloride etchant! Always use ammonium persulfate.
10. Always etch at an elevated temperature, around 120 degrees Farenheit. A warming plate from a yard sale is ideal for this. Agitage the etchant with a gentle sloshing or bubbles from an aquarium pump.
11. Never etch with your foil side up!. Support the board vertically, or else *foil side down* at least 1/2 inch above the bottom of the etchant tray. A mirror *under* a glass etchant tray lets you view etching progress. Use only plastic or glass in contact with your etchant.

FIG. 2—SOME REALLY DUMB MISTAKES are often made by hackers who do their own printed-circuit boards the "old way." Here is how to avoid the worst of the pitfalls of the traditional methods. But this is all ancient history, because...

PRINTED CIRCUIT RESOURCES

Advance Process Supply
400 North Noble Street
Chicago, IL 60622
(312) 829-1400

Bishop Graphics
5210 Lewis Road
Agoura Hills, CA 91376
(818) 991-2600

Black Lightning
RR 1-87 Depot Road
Hartland, VT 05048
(802) 359-2790

Circuits Manufacturing
500 Howard Street
San Francisco, CA 94105
(415) 397-1881

Datak
3117 Paterson Plank Road
North Bergen, NJ 07047
(201) 863-7667

DuPont Riston
3945 Freedom Circle F4
Santa Clara, CA 95054
(408) 562-9300

Electronic Manufacturing
17730 W Peterson Road
Libertyville, IL 60048
(312) 362-8711

Electronic Packaging
1350 E Touhy Avenue
Des Plains, IL 60018
(312) 635-8800

Kepro Circuit Systems
630 Axminister Drive
Fenton, MO 63026
(314) 343-1630

Kodak
343 State Street
Rochester, NY 14650
(716) 724-4000

sources as *Dick Blick*, *Southern Sign Supply*, and *Advance Screen*. Two trade journals that serve the field are *Screen Printing* and *Signcraft* magazines.

At one time, printed-circuit boards were etched using a ferric chloride solution. Today, ferric chloride is a *very poor* choice of etchant. A much better etchant choice is ammonium persulfate. It is much cleaner, faster, and easier to use. Being a light blue solution, ammonium persulfate also lets you view the board as it is being etched. Etching best takes place at an elevated temperature, typically 120 degrees Fahrenheit. You could easily hit that temperature with a modified aquarium heater, a warming plate from a yard sale, or any of the strip heaters found on the surplus market.

Everything that comes in contact with the etchant must be glass or plastic. PVC is often usable. Ideally, your etchant should be sprayed onto the vertically held boards. Other ways to keep the etchant moving would be a simple manual sloshing or injecting air from an aquarium pump.

One really dumb mistake that most hackers make when etching their first PC boards is to place their board *face up* in the etchant solution. All that does is redeposit sediments and any crud removed from the board back on itself, leading to all sorts of nasty problems.

Instead, always support your boards vertically in the etchant, or else use surface tension to float

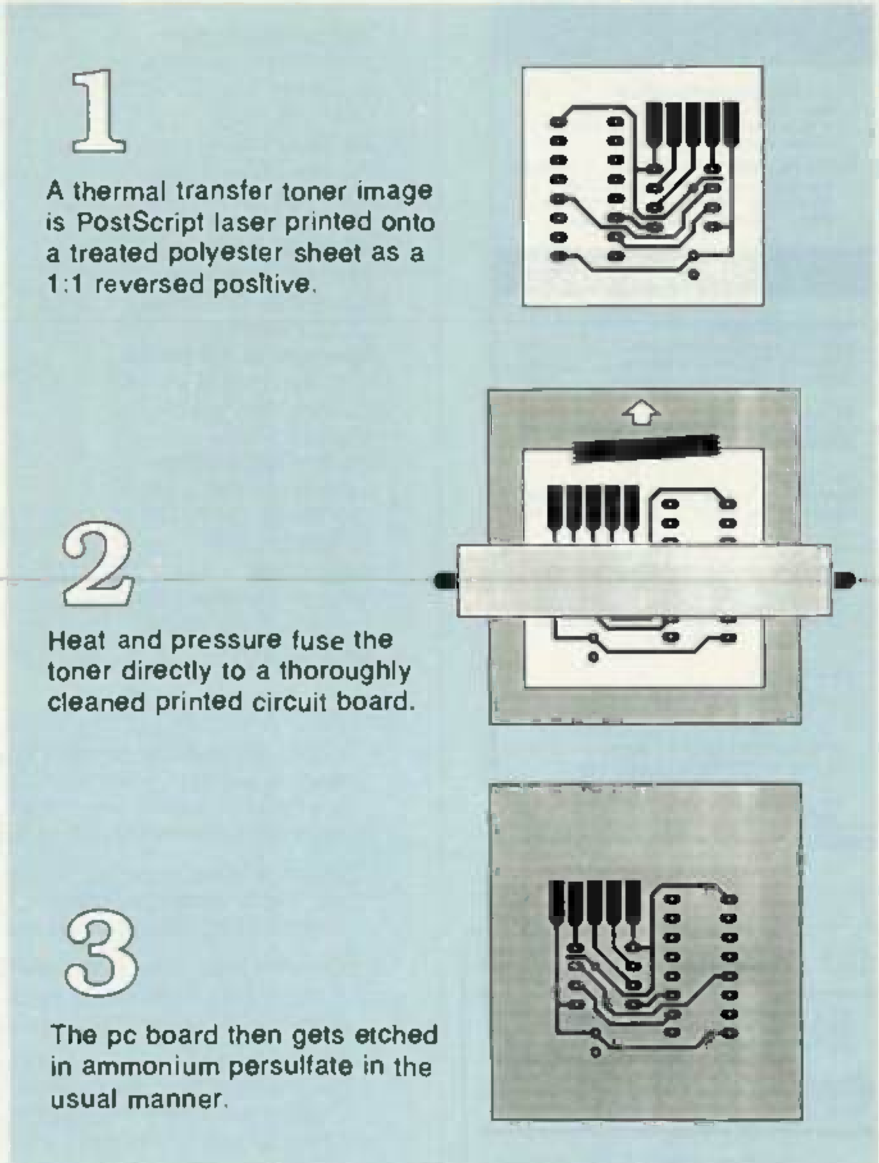


FIG. 3—OUR BRAND NEW DIRECT TONER TRANSFER method can dramatically simplify and speed up making all of your hacker printed-circuits at a cost only of pennies per board. Here are the three key steps in this breakthrough process.

Lazer Products
12741 E. Caley Ave. #130
Englewood, CO 80155
(303) 792-5277
Meadowlake
25 Blanchard Drive
Northport, NY 11768
(516) 757-3385
Miller-Stephenson
Box 950
Danbury, CT 06813
(203) 743-4447
Mill-Max
190 B Pine Hollow Road
Oyster Bay, NY 11771
(516) 922-6000
Rogers Corp
100 S. Roosevelt Avenue
Chandler, AZ 85226
(602) 961-1382
Screen Printing
407 Gilbert Avenue
Cincinatti, OH 45202
(513) 421-2050
Sign Craft
1938 Hill Avenue
Fort Myers, FL 33906
(813) 939-4644
Southern Sign Supply
127 Roesler Road
Glen Burnie, MD 21061
(301) 768-8600
Stimpson
900 Sylvan Avenue
Bayport, NY 11705
(516) 472-2000
Surface Mount Technology
17730 W. Peterson Road
Libertyville, IL 60048
(312) 362-8711
Synergetics
Box 809
Thatcher, AZ 85552
(602) 428-4073
Thiokol Dynachem
2631 Michelle Drive
Tustin, CA 92681
(714) 730-4200
Thompson & Thompson
23072 Lake Center #100
El Toro, CA 92630
(714) 855-3838
Ulano Corp
255 Butler Street
Brooklyn, NY 11217
(718) 622-5200

NAMES AND NUMBERS

Adobe PostScript
1585 Charleston Road
Mountain View, CA 94039
(415) 961-4400

All Electronics
PO Box 567
Van Nuys, CA 91408
(800) 826-5432

EG&G Reticon
345 Potrero Avenue
Sunnyvale, CA 94086
(408) 738-4266

Freedom of the Press
900 Tech Park Drive #8
Billerica, MA 01821
(800) 873-4367

Go Script/LaserGo
9235 Trade Place, Ste A
San Diego, CA 92126
(619) 530-2400

Kroy Kolor
14555 N. Hayden Road
Scottsdale, AZ 85260
(800) 521-4997

JKL Components
13343 Paxton Street
Pacoima, CA 91331
(800) 421-7244

Robert A Main & Sons
555 Goffle Rd, Box 159
Wyckoff, NJ 07481
(201) 447-3700

Rohm Corporation
8 Whatney
Irvine, CA 92718
(714) 855-0819

SPIE
PO Box 10
Bellingham, WA 98227
(206) 676-3290

Synergetics
Box 809
Thatcher, AZ 85552
(602) 428-4073

Winzeler Inc
7355 W. Wilson Avenue
Chicago, IL 60656
(312) 867-7971

1. Create a PostScript printed circuit artwork image on disk, using your favorite word processor and the sample routines of figure five and six, the full code found in my *PostScript Show and Tell*, or some suitable third party printed circuit layout package.

2. Run a positive, reversed 1:1 proof on a PostScript speaking laser printer, such as an *Apple LaserWriter IINT*. Many copy shops offer this service. Low cost PostScript printers are available through *Don Thompson*.

3. Optional step: Take a polyester based, laser printable overhead transparency material and lightly coat one side with *Miller-Stephenson* type MS-136 heated mold release agent. Mark the coated side.

4. PostScript laser print a 1:1 positive reversed (black = foil; right = left) onto the coated side of the polyester sheet, using a special thermal transfer (T-shirt) toner from *Black Lightning, Don Thompson, or Lazer Products*. Other graphic toners might also work; try them and see.

6. Sharpen and smooth the leading edge of an oversize piece of 1/16th inch printed circuit material with a file and steel wool, so that it can be sent through a fake *Kroy Kolor* machine without hurting the rollers. Use a 3/8 to 1/2 inch leading slope.

7. Thoroughly clean this oversize printed circuit board, scouring it three times with fine steel wool and *Comet* cleanser, followed by a chemical cleaner, followed by a brief etch. The board must be a uniform hot pink in color and must allow an unbroken stream of water to flow smoothly over it. Be sure to avoid any and all fingerprints.

8. Tape the leading edge of the polyester sheet *toner side down* to the copper side of the printed circuit board, using a suitable high temperature tape. Make sure the polyester sheet lies flat.

9. Run the board, image side up, through a preheated fake *Kroy Kolor* machine adjusted to a medium temperature. One source of these machines is *Lazer Products*. See the November 88 *Radio Electronics* or my *hardware Hacker II* reprints for details on building your own machine.

10. Optional step: Chill the board suddenly in a freezer before lifting the polyester sheet. Allow to warm to room temperature, then bake for fifteen seconds at 300 degrees F in a kitchen oven.

11. Etch in the usual manner in ammonium persulfate etchant.

FIG. 4—THE STEP-BY-STEP "BASELINE" process for the new toner transfer PC method. An ordinary iron can substituted for the Kroy Kolor machine, but the results may not be as good. Let us know your experiences here.

```
% Copyright © 1988 by Don Lancaster & Synergetics, Box 809, Thatcher AZ, 85552.
% (602) 428-4073. All rights reserved. Personal, non-commercial use permitted so long
% as this header remains both present and intact. Show & Tell disk costs $39.50.

/quadpixel {transform 4 div round 4 mul itransform} def

/setgrid {save /rubbersnap exch def /size exch def quadpixel exch quadpixel exch translate
size dup scale } def

/drawlines {72 300 div lw mul size div setlinewidth /hposs 0 def #hlines gs div 1 add cvi
{hposs 0 moveto 0 #vlines rlineto stroke /hposs hposs gs add def} repeat /vposs 0 def
#vlines gs div 1 add cvi {0 vposs moveto #hlines 0 rlineto stroke /vposs vposs gs add
def} repeat} def

/showgrid{gsave /#vlines exch def /#hlines exch def 106 45 {pop pop 0} setscreen 0.9
setgray /gs 1 def /lw 1 def drawlines grestore} def

/1X {30 72 mul 300 div setgrid} def

/trace20 {6 30 div setlinewidth} def /trace50 {16 30 div setlinewidth} def /trace80 {24
30 div setlinewidth} def

/am {newpath moveto} def
/tdraw {rlineto currentpoint stroke moveto} def

/u {0 exch tdraw} def          /r+ {dup tdraw} def
/r {0 tdraw} def               /r- {dup neg tdraw} def
/d {0 exch neg tdraw} def      /l- {neg dup tdraw} def
/l {neg 0 tdraw} def           /l+ {dup neg exch tdraw} def

/black {0 setgray} def         /white {1 setgray} def

/xrpt{gsave aload pop /trips exch def /dist exch def /rproc exch def trips { gsave rproc
grestore dist 0 translate } repeat grestore} def

/yrpt{gsave aload pop /trips exch def /dist exch def /rproc exch def trips { gsave
rproc grestore 0 dist translate } repeat grestore} def

/hole {gsave 150 div /dia exch def newpath dia 2 div 0 360 arc white fill grestore} def

/icpad1v {save /psnap exch def trace50 2 copy gsave exch 0.2 sub exch am 0.4 r grestore
20 hole psnap restore} def

/edgeconu {gsave translate 0.4 0 moveto 0 -2 0.4 0 180 arcn 0 2 rlineto closepath fill
grestore} def

/feedpad {save /fpsnap exch def newpath 2 copy black 0.25 0 360 arc fill 18 hole clear
fpsnap restore } def

/circpad2 {save /fpsnap exch def newpath 2 copy black 0.30 0 360 arc fill 22 hole clear
fpsnap restore } def

/dip8v{gsave translate [{0 0 icpad1v} 1 4] yrpt [{3 0 icpad1v} 1 4] yrpt grestore} def

/dip16v{gsave translate [{0 0 icpad1v} 1 8] yrpt [{3 0 icpad1v} 1 8] yrpt grestore} def

1 setlinejoin 1 setlinecap
```

FIG. 5—SOME SAMPLE POSTSCRIPT PC-LAYOUT routines that were excerpted from my PostScript Show & Tell disk. Just shove this listing into your favorite word processor as a prolog to your actual layout.

the board *upside down* on the surface of the etchant. Another ploy is to add nylon spacers to your PC board so the foil faces *down* in your etching solution. Or else throw some nylon hex nuts in the etchant and sit the board upside down on the nuts. Once again, *never etch a PC board face up!*

A second stupid mistake that lots of hackers make is failing to clean the boards properly. It is not possible to clean a circuit board at home without spending at least *two minutes per square inch of board.* Begin by using *Comet* or another chlorine-activated cleanser with a fine steel-wool pad or *Scotchbrite* pad. Rinse thoroughly and wipe on an inner fresh turn of a new roll of paper towels. Repeat that at least three times, avoiding any and all fingerprints.

Note that fairly clean copper will allow an unbroken stream of water to flow over it without any running or beading. Your key secret is that a genuinely and totally clean copper will not be copper-colored at all. Instead, it will be certainly a uniform hot pink. Commercial copper cleaners, such as CU3 from *Kepro*, are a great help, but are somewhat expensive.

The ultimate final cleanliness step is to place the copper in ammonium persulfate and etch it for ten to fifteen seconds or so. Then thoroughly rinse three times and air dry immediately. If you get a uniform hot pink result, then your copper is clean enough for immediate use.

The photographic methods get rather complicated, but they can be used for arbitrarily fine lines and for all of the precision you will ever need. In fact, the same techniques are used to manufacture integrated circuits to a fraction of a micron accuracy. Note that there are 20 microns in a mil.

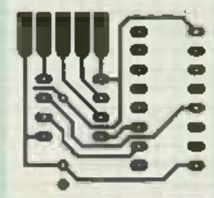
With the photo processes, a light-sensitive *etch resist* is placed on the thoroughly cleaned board. The resist is first contact-printed from a photographic negative and then developed. In the most popular *negative-acting* systems, those portions of the resist that receive light harden and remain; those that did not will dissolve out. Etching is done in the usual manner.

Traditionally, the spray-on photoresist was used, such as a KPR product from *Kodak*. These days, though, it is far simpler and far better to use a dry-film photoresist, such as the *Riston* materials by *DuPont*, or any of the *Laminar AX* products from *Thiokol*. Those dry films develop in trisodium phosphate, the garage-floor cleaner found at your local hardware store. They are quite resistant to pinholes, eliminate dust and drying problems, have highly visible images, and are always at the right thickness. Once sensitized, the boards must be kept dry, cool, and in total darkness. They also have a one-year shelf life.

The third most stupid mistake that hackers make is trying to use KPR instead of the new and infinitely better dry films. Mistake number four, of course, is trying to coat their own boards instead of using pre-coated ones. Excellent dry-film pre-coated boards are stocked by *Kepro*.

The cost of the dry-film resist by itself is around a dollar per square foot. Unfortunately, a fancy laminator is needed to bond the resist to the board. I have a hunch that a Kroy Kolor machine or one of its imitators can be substituted here. The required temperature is 234 degrees Fahrenheit. Let me know if you pick up any experience along those lines.

Double-sided plate-through boards often use a combination of processes. Typically, you start with a double-sided board. The holes are first drilled, and then they are plated through by additive techniques. The holes get chemically activated, and then seeded with an ultra-thin palladium plating. Electroless copper is then built up on the conductive palladium to a medium thickness, followed by a heavy copper plating up to the final wall thickness needed. The rest of the board is then processed through the usual double-sided photographic steps. Key hacker printed-circuit mistakes are summarized for you in Fig. 2.



```
% requires printed circuit sampler code of figure five

100 100 1X 11 11 showgrid trace20 10 2 am 1 l 0.75 l- 4.5 l 0.75 l+
2 l 6 u 10 5 am 0.75 l 1.5 l- 1.25 l 0.5 l- 2 l 0.5 l+ 0.75 u 1.25 l+ 7 4 am
0.75 l 0.5 l- 0.75 l 5 5.5 am 1 l+ 1.5 u 5 4.5 am 2 l+ 2 u 2 6.5 am 1.5
u 7 3 am 0.5 l- 2.75 l 0.75 l+ 0.5 u 0.75 l+ 2 3.5 am 1 l 10 9 am 0.75 l+
2.75 l 0.5 l- 2.75 d 1 l 6 6.5 am 2.15 d 0.35 l- 1.15 l 2 l+ 1 l 5 6.5 am
1.5 u 7 2 dip16v 2 3.5 dip8v [{1 10 edgeconu} 1 5] xrpt 3 2 circpad2
3 1 circpad2 3 5.5 feedpad showpage quit
```

FIG. 6—A TYPICAL PC-TEST LAYOUT. Note how simple and short the code is. Omit the showgrid command to drop out the fine gray viewing grid. Be sure to preface this code with the PostScript routines of Fig. 5.

Printed-circuit resources

I've gathered some of the major PC-board resources into our first sidebar, as we've done in previous columns for other topics. Most of the products we have mentioned are available directly through those sources.

The best trade journal for printed circuits is *Circuits Manufacturing*. A few others are *Electronic Packaging and Production*, *Surface Mount Technology*, and their sister publication, *Electronic Manufacturing*. Be sure to let me know if there are other resources that you think should be added to the list.

The direct-toner method

There's a new process on the block for hacker printed-circuits which is ridiculously simpler, faster, and far cheaper than any of the above. All you really need is a word processor and an iron. This new scheme is known as the *direct-toner method*. And it is new and undeveloped enough so that you might play a major role in making it work and shaping its future.

Very simply, copier or laser-printer toner is outstanding as an etch resist. Two decades ago, *Xerox* even had a product that directly printed on your copper PC boards from a 2:1 artwork original. A ways back, a new hacker product known as *Meadowlake* did attempt an iron-on toner system. Early versions of the product didn't turn out reliable enough and lacked stability.

But there is a brand new type of *thermal-transfer toner* now carried by several laser-printer supply houses. While the toner is intended for making iron-on T-shirt images, it transfers to copper beautifully and smudge-free, and is thus a key secret to the direct-toner process. Three sources of a thermal-transfer toner are *Black Lightning*, *Lazer Products*, and *Don Thompson*. *Black Lightning* does offer a free sample. Cost of the toner ranges from $90 to $180 per cartridge, which translates to a dime per board. Several of the other new graphics toners should also work well. Your help is needed in pinning down which ones are acceptable and which are not.

Figure 3 summarizes the key toner-transfer steps; that is followed by some detailed instructions in Fig. 4.

The best way I've found to create initial artwork is by using a PostScript speaking laser printer and my word processor. In fact, I have a complete package that does just that for Apple, Mac, and IBM users. Note that the original image must be a 1:1 reversed positive. That means that left is where right belongs and black is where you want your foil to *remain*. Naturally, since your image is disk-based, it is easy to change, and super easy to build up from a library of suitable Post-Script dictionary routines.

The image gets printed, again as a 1:1 reversed positive, onto a laser-printable Mylar or polyester overhead projection sheet. Just for luck, I'll previously apply a very thin coating of MS-136 *Heated Mold Release Agent* from *Miller-Stephenson*. That may or may not help, but it sure seems like a good idea, at least for now. It also may be a good idea to anneal or remelt the toner for a few seconds in an oven, after the image is transferred.

Too much reheating, of course, would lower the resolution. Although ten-mil lines on twenty-mil centers should be possible, I'd stick with double that as an initial lower limit.

In theory, you could simply iron the toner directly onto a previously super-cleaned PC board. Instead, I modify the board by *sharpening* its leading edge, and run it through one of the imitation *Kroy Kolor* machines we looked at in **Radio-Electronics**, **November, 1988**. By the way, an improved and economical do-it-yourself version of that beast is in the works here at **Radio-Electronics**.

The benefits of the new way are obvious. You go from artwork to PC prototype amazingly fast. No cameras, chemistry, screens, or fancy equipment is needed. Without any fuss or bother. Just print and etch. And products such as *GoScript* and *Freedom of the Press* even let you fake PostScript on a dot-matrix printer, so "no printer" is no excuse.

The technique could also revolutionize running hacker PC projects. You show the PostScript code in the magazine and offer it downloadable off your BBS. Now, every hacker can end up with a precisely accurate original, rather than a third-generation copy.

Figure 5 shows some sample PostScript code from my PC layout stuff, while Fig. 6 shows a simple actual layout that you can use to test out your own version of the direct-toner board method.

As I said before, this is a brand new technique that needs some further experimentation to perfect. So, for our first contest this month, just tell me anything you find out on your own about making a direct-toner PC board. There'll be the usual dozen or so *Incredible Secret Money Machine* books for best entries, along with an all-expense-paid (FOB Thatcher, AZ) *tinaja quest* for the very best of all. Let's hear from you on this hot new topic.

New tech literature

The hacker buy of the month has to be the new $3.75 speech synthesizer available as stock number ICA from *All Electronics*. It even includes a load-power sensing detector. There is one very tiny gotcha, though—the main thing the synthesizer has to say is "Your ice cream is ready." Oh, well.

The *EG&G Reticon* folks have three brand new data books out. They are their *Image Sensing Products, Solid State Cameras*, and the *Analog Signal Processing Integrated Circuits*. The latter covers analog delay lines, switched-capacitor filters, and some very useful ap-notes.

The BU2911 is an interesting new melody chip from *Rohm*. It provides a pair of folk songs and six different chime and siren sound effects. Uses? How about some clocks, doorbells, or alarms? Miniature fluorescent lamps from *JKL* are offered in various colors at very low prices; they also carry EPROM erasing lamps.

The *SPIE* folks have great bunches of reprints involving nearly everything optical, and covering laser scanning, CD ROM, holography, fiber optics, and I-R detection. Get their list number 6.

Turning to mechanical stuff, free molded-gear samples are obtainable from *Winzeler*. And *Robert A. Main* has a unique catalog of *Hooks, Points, and Teeth*. A sharp outfit fer sure. Tellyawhat. Their catalog number twelve is so unusual, that we'll make us a second contest out of it. Just come up with some off-the-wall use for any of the hooks, points, or teeth shown.

If you want to explore our new direct-toner hacker PC process further, or want to draw your own first-quality electronic schematic diagrams, isometric drawings, or architectural perspective sketches (including all lettering!) using nothing but your favorite word processor, check into my *PostScript Show and Tell*, that has scads of working code and detailed examples in it. Available for Apple, Mac, or IBM, it does need a PostScript speaking laser printer (or a software emulator).

As always, this is your column and you can get technical help and off-the-wall networking per the *Need Help?* box. The best calling times are weekdays 8–5, *MST*. R-E

PROPERTY GUARD

continued from page 40

transducer is approximately 40-volts p-p. Tap 6 may be used for frequencies below 20 kHz with "on" times no more than 20 minutes; voltage is approximately 50-volts p-p. Tap 13 is intended for intermittent use where "on" times are less than 3 minutes and the frequency well below 20 kHz. Those times may be longer when operating at lower frequencies. Never allow the transducers to get excessively warm. That completes the testing and adjustment of the system.

Setup

The Phasor Property Guard is capable of operating in two modes. Mode 1 is at a frequency that is known to produce paranoia, nausea, disorientation, and many other physiological effects. Mode 2 allows using the system as an audible alarm to frighten off intruders or warn the user of an intrusion. Both modes may be used in combination, and are easily controlled by the user. Three separate jacks provide inputs for a broken trip wire or contact foil, a pressure or actuating switch, and a positive voltage pulse from other equipment.

The position of the transducers should be such that they direct their energy to the points of intrusion or access. They can be all directed to any target area, or be individually placed for multiple effect.

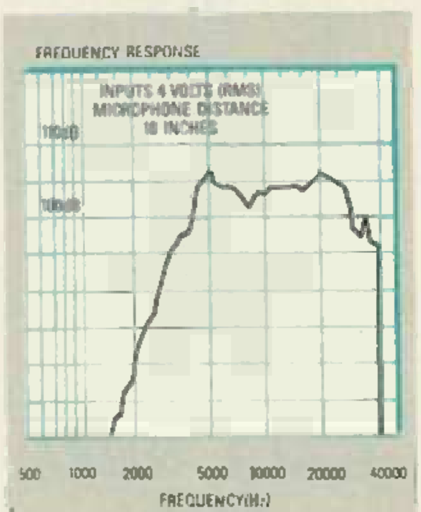


FIG. 7—FREQUENCY RESPONSE of the piezoelectric transducers.

The transducers used in this system are piezoelectric and are many times more efficient than the electromagnetic-type speaker. Their frequency response is shown in Fig. 7.

Ultrasonics

Ultrasonic is a gray area in many respects when the application involves the control of animals or as an intruder deterrent. It is always best to consult with local municipal and state laws before using this device to protect home or property.

Do not operate at continuous high output at frequencies below 20 kHz. Daily sound pressure exposures in excess of one hour at 105 dB may lead to hearing impairment. When properly used, this device provides a limited liability deterrent. It should not cause permanent damage or trauma.

There have been numerous requests for information on the effect of these devices on people. First of all, an ultrasonic device should not be used unnecessarily on humans, because of the possibility of acoustically sensitive people being highly irritated.

Remember that the Property Guard cannot stop a person with the same effect as a gun, club, or more conventional weapon. It will, however, produce an extremely uncomfortable, irritating, and sometimes painful effect in most people. Although not everyone will experience the effect the same degree. Younger women are much more affected than older men, due to being more acoustically sensitive. The range depends on many variables, but is normally somewhere between 10 and 100 feet. R-E