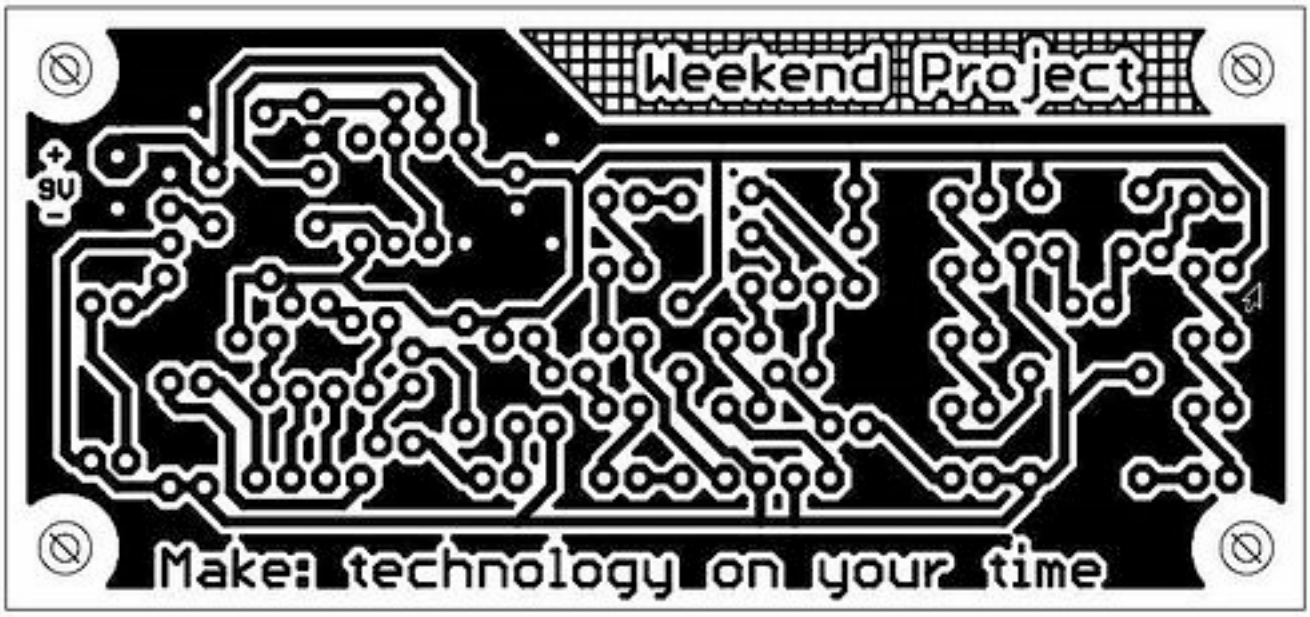
Weekend Projects

Make: technology on your time

Create Your Own PCB with a Laserjet Printer!



This weekend I teamed up with Xander Hudson to show how you can make your own printed circuit board (PCB)! It's a fun way to create your own circuit over the weekend.

To do this project, Xander started by designing it in Eagle CAD, a program stocked with the sizes and dimensions of tons of parts so that you can literally just drag and drop your parts onto the board and choose how they will fit together. Once you've got your design all set up, you'll begin making your board.

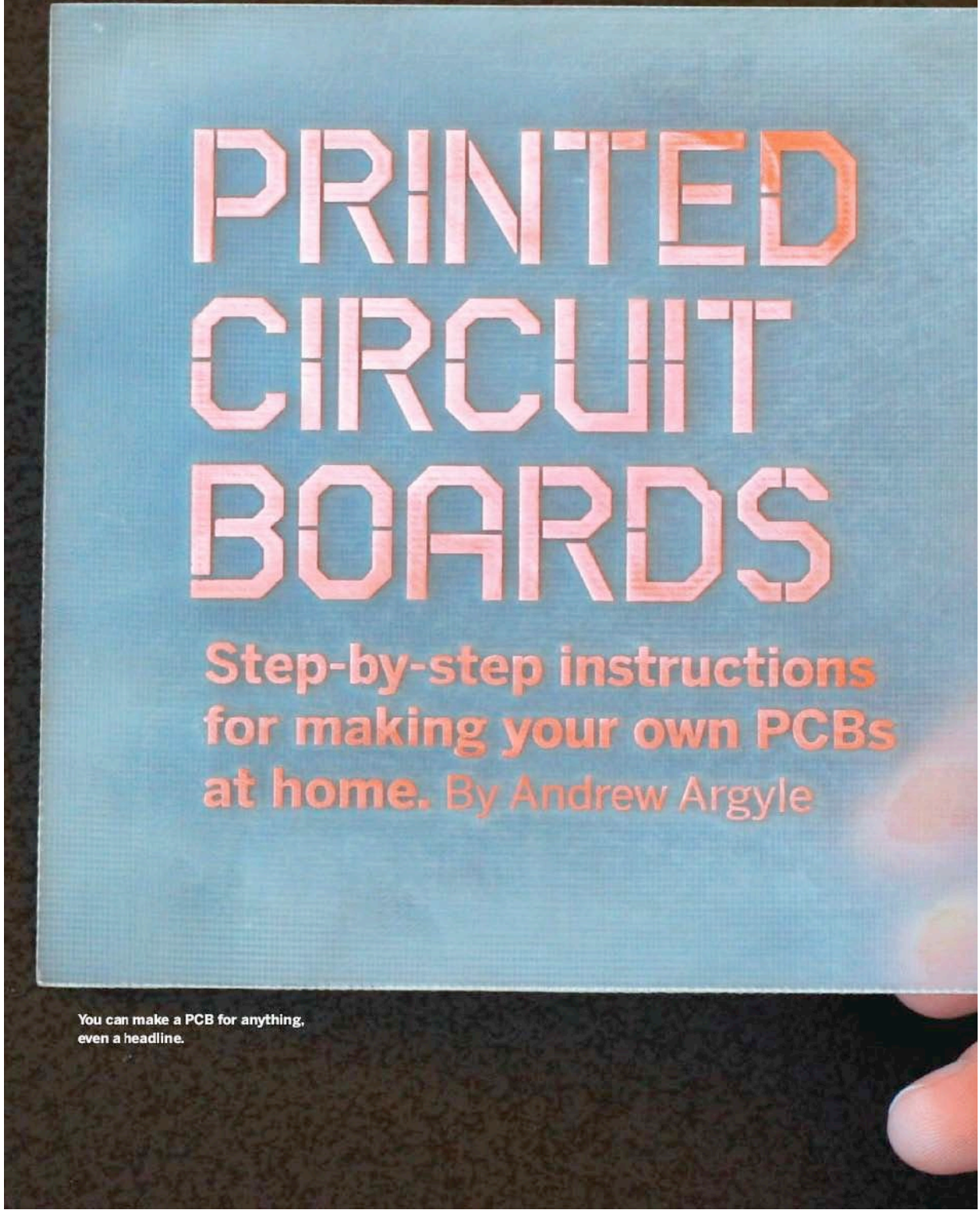
Below I've included instructions from Make: Volume 2 by Andrew Argyle. In the video we also reference Tom Gootee's website at http://www.fullnet.com/~tomg/gooteepc.htm which is where we heard about using Staples photo paper, the results will be the same, but the photo paper is cheaper!

You are welcome to make or modify Xander's fun blinking LED Make: light by downloading all the files here: http://cachefly.oreilly.com/make/makelightfiles.zip
Once you've made a PCB, make sure to upload pictures of your schematic, board design and finished project to the Make: flickr pool.

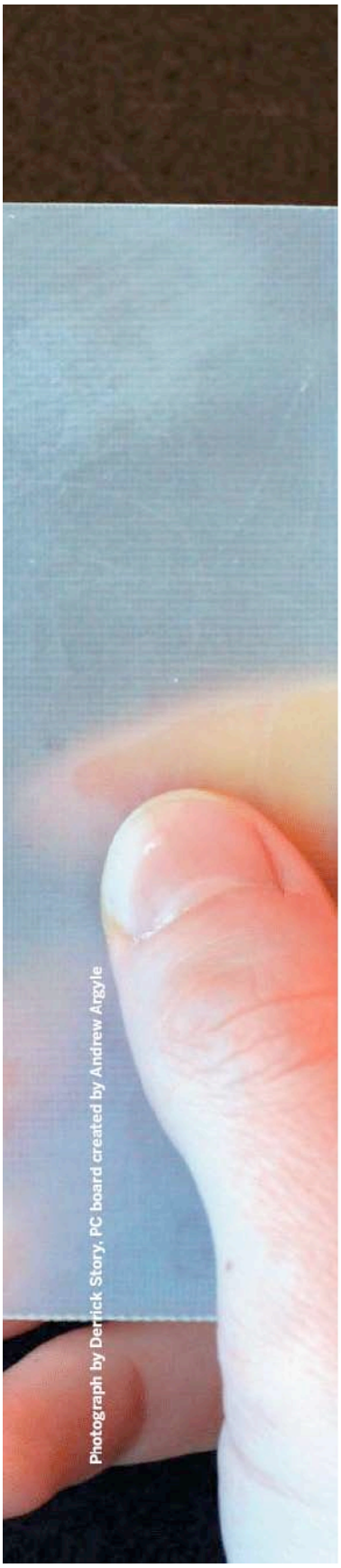
PRIMER

PRINTED CIRCUIT BOARDS

Step-by-step instructions for making your own PCBs at home. By Andrew Argyle



You can make a PCB for anything, even a headline.



Photograph by Derrick Story. PC board created by Andrew Argyle

Making your own printed circuit board (PCB) might seem a daunting task, but once you master the steps, it's easy to attain professional-looking results.

Printed circuit boards, which connect chips and other components, are what make almost all modern electronic devices possible. PCBs are made from sheets of fiberglass clad with copper, usually in multiple layers. Cut a computer motherboard in two, for instance, and you'll often see five or more differently patterned layers. Making boards at home is relatively easy, but limited to one- or two-layer designs. Here's how to create a one-layer board for a blinking LED circuit. For PCBs, this exercise is the equivalent of a beginner's "Hello World" program in software.

PCB Make vs. Buy

You can order custom PCBs through design houses, but they are expensive — and the per-unit cost is especially high for small batches. Also, when you change the design, a shop will typically take several days to turn it around. Since it's normal for a board to go through several revisions, this adds up to a significant delay.

MATERIALS:

TECHNIKS PRESS-N-PEEL PCB TRANSFER FILM (20 sheets for $30), techniks.com

AMMONIUM PERSULFATE CRYSTALS (1Kg for $20), mgchemicals.com

BLACK CORRECTION PEN (Lumocolor or fine-tip Sharpie)

PCB COPPER BOARD (9 x 12 x 0.062-inch, 1 oz., single-sided, $12), techniks.com

CIRCUIT COMPONENTS

TOOLS:

SCOURING PAD (do not use steel wool)

PHOTOCOPIER OR LASER PRINTER

ELECTRIC CLOTHES IRON

HARD WOODEN BOARD

LARGE POT AND PLASTIC TRAY for etching

HIGH-SPEED DRILL AND BITS

SOLDERING MATERIALS

PERF BOARD AND WIRE (OPTIONAL) for prototyping

NOTE: Techniks Press-n-Peel transfer film is a reasonable compromise between ease-of-use and cost. You could mask a board for etching with just a permanent marker, but this would take a long time, and it would be extremely difficult to make the lines sharp and accurate enough.

Advantages of PCBs

Circuitry built on custom boards is sturdier, more stable, and more compact than the same electronics strung together by hand. And since it takes less time to assemble, PCBs are also better for making multiples. They're cheap and easy to prototype, and you can find a variety of templates and projects online.

Process Overview

We'll design our printed circuit on the computer, as a plain black-and-white image. Black represents where we want the board to have copper connections between our circuit's components, while the white background corresponds to areas of plain, non-conducting fiberglass. Then we'll print the design onto special transparency film and iron it onto the copper side of a blank PCB (which starts out with copper completely covering one side). We peel the film back off, and the printed design remains on the copper. Next, the magic happens: when we bathe the board in an etching solution, copper dissolves away in the blank areas of our design, leaving conductive "traces" behind where we had drawn the circuit's connections. After this, we prepare the board for soldering, then finally go to town with the components.

Broken down step-by-step, the entire process can be summarized as follows:

1. **Create the design.**
2. **Print or photocopy the design.**
3. **Prepare the PCB.**
4. **Transfer the design to the copper.**
5. **Check and correct the traces.**
6. **Etch the board.**
7. **Clean and drill the board.**
8. **Solder the components.**

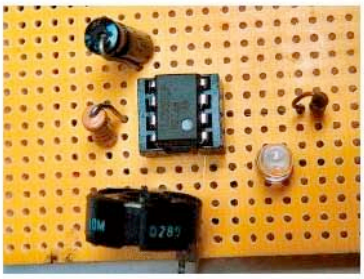
TIP: PCB design houses are well worth the price if you have a finished layout. But if your project is relatively simple, or the design is still evolving, then making your own PCB is the way to go.

Photography by Andrew Argyle

START »

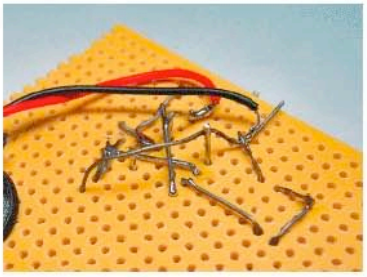
1. CREATE THE DESIGN

If you're building a circuit from scratch, rather than using a known design, you should prototype it first. One way is to string it up on a piece of perf board, as shown at right. Our "Hello World" blinker circuit has a 555 timer chip, a battery, a capacitor, an LED, and three resistors.



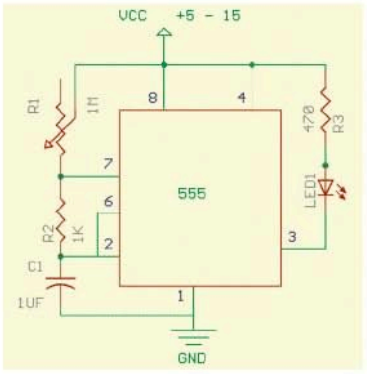
Prototype top.

The underside of our proto-board is a "spaghetti patch" of wiring that will inevitably lead to problems. Even with this simple example, it's easy for wires to touch and short out. With all wires shielded, it's still ugly — inelegant and difficult to decipher and debug. This is a powerful reason for making a PCB.

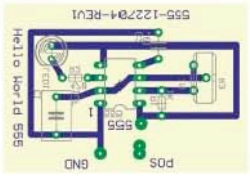


Prototype bottom.

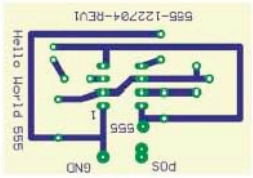
Start with your circuit's schematic diagram, then lay out the PCB design based on the actual sizes of your components. You can create your design with any drawing, paint, or design program. I use EAGLE CadSoft (www.cadsoft.de), which is shareware.



Schematic.



PCB with part placements.



PCB without part placements.

First, produce your layout with components drawn in life-size. Print this out double-size for later reference. Then generate your final layout by removing the parts outlines.

You're designing the underside of the board, so your layout should be a mirror image of the arrangement you'd see looking from above. Also, you'll turn the printing to face the copper you're etching, so lettering should be backward.

Indicate parts placement and the location of Pin 1 on any chip. For future reference, you should also list the design's date and a revision number.

And with a small design, make multiple copies side-by-side; this increases the odds that at least one will work.

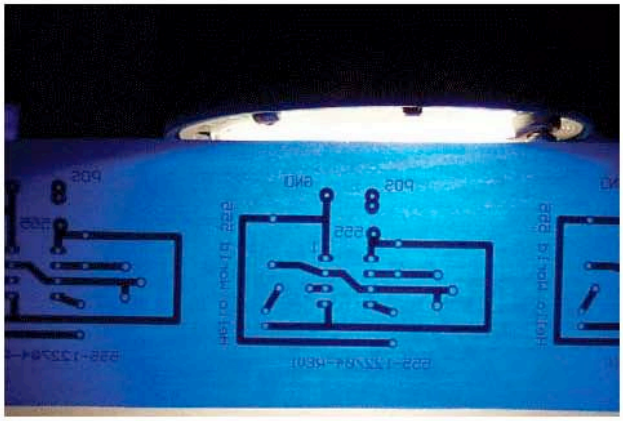
»

2. PRINT OR PHOTOCOPY THE DESIGN

Place the Techniks Press-n-Peel PCB transfer film in the tray of a laser printer or photocopier. One side of each sheet is shiny, while the other side is dull. Place the film so that the printing will come out on the dull side.

When using a laser printer, you want to minimize the possibility of damaging or curling the film. Use the "manual feed" feature to avoid making the film roll around the drum.

Once the film is printed, check it against a strong light for any defects. The film should be dark and no light should come through the traces.



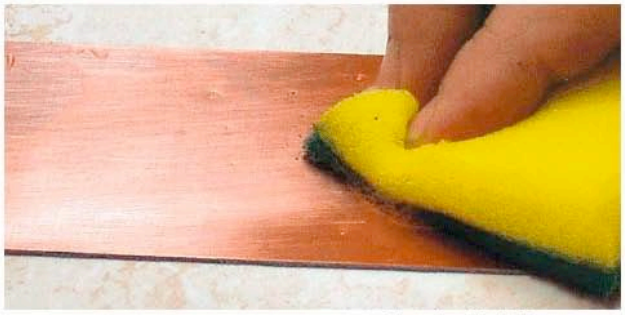
Checking for broken traces on the film.

3. CLEAN AND PREPARE THE PC BOARD

The PCB copper board should be as clean as possible so that the design adheres to it. Scrub it with dish soap and a scouring pad. Do not use steel wool, because it leaves tiny pieces of steel behind on the copper, which interfere with the etching.

Water should flow easily off the copper board. Dry the board to a polished finish with a lint-free cloth.

You should use the board as soon as possible after cleaning, because otherwise the copper will begin to slowly oxidize in the air, diminishing its ability to take the design transfer and to etch well. Make sure not to touch the copper with bare hands.



Cleaning the PCB.

NOTE: DO NOT TOUCH THE COPPER WITH YOUR BARE HANDS, SINCE SWEAT AND OIL LIKEWISE INTERFERE WITH THE TRANSFER AND ETCHING PROCESSES.

4. TRANSFER THE DESIGN TO THE COPPER

This is the most challenging step of making PCBs at home. First, cut the transparency to the same size as the PC board.

Find a fairly hard surface to iron on, such as a thick, wooden cutting board or a scrap piece of wood.

Set the iron to the polyester/rayon setting (medium heat on most irons). Once the iron has come up to temperature, lay it down directly on the wood to heat it. The warm wood will help with the adhesion and transfer of the film. Proper temperature control is critical; you will find it only through experimenting with your iron.

After about five minutes of warming the wood, lift the iron and lay the PCB down copper-side up. Align the film dull/printed-side down on the copper, and begin ironing firmly over the sheet, never lingering longer than four or five seconds in any one place. You want to avoid smearing the design.

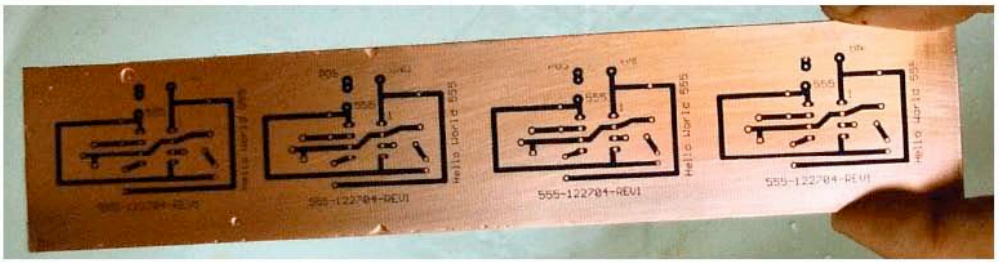
Continue ironing for about 45 seconds, or longer if the copper board is thick. Then remove the board and let it air-cool, or cool it faster by putting it under cold water.

Ironing the PCB.

NOTE:
WARM THE IRONING SURFACE FOR FIVE MINUTES BEFORE IRONING THE PCB.

Settings:
SET IRON ON POLYESTER/RAYON

Alignment:
CUT THE TRANSPARENCY TO THE SAME SIZE AS THE PC BOARD



Cooled PCB board.

Carefully pull the film away from the board. If the design stays behind and appears on the copper, it's transferred successfully. Congratulations!

5. CHECK AND CORRECT THE TRACES

Examine the board for obvious errors. You can fix broken traces by filling in the gaps with a permanent marker such as a fine-tipped Sharpie. Use a hobby knife to separate any traces that have smeared together. Take the time to do a good, thorough job with this step, which can make or break your final product. For this reasor, it's smart to print several designs, rather than pinning all of your odds on one.

6. ETCH THE BOARD

The chemicals used in etching are corrosive, and their fumes are irritating. Follow proper safety procedures, wear a mask (or work in a well-ventilated area), and protect your eyes.

There are different types of etching chemicals: dry and liquid. Dry chemicals are less expensive, but require dilution, as directed on the packaging. For this project, we recommend ammonium persulfate, available at lab and electronics supply stores. You will need about 1 quart of etching solution (etchant) to etch a 6" board.

Etching time is related to temperature. A small increase in temperature drastically reduces the time needed to etch a board, but too high a temperature makes the etchant too aggressive ard gives off fumes. Aim for a temperature of around 125 degrees Fahrerheit (52°C).

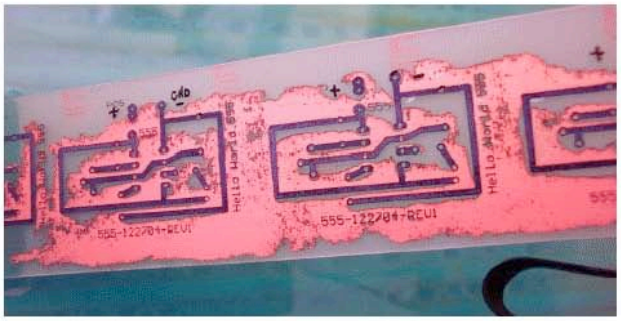
Desigr houses have temperature-controlled etching tanks. For etching at home, you just need a stove, a large pot, ard a plastic tray for the etchant.

Fill the pot halfway with water and heat to a simmer, but not a boil.

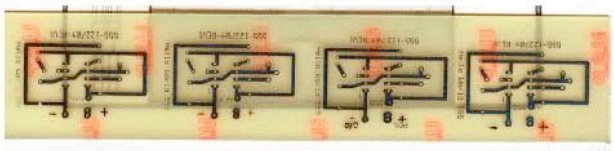
Put the PCB in the tray and place the tray on the simmering pot. Then slowly pour the etching solution into the tray.

The steam from the pot will slowly heat the etchant and increase the rate

Etch the PCB on the stove.



Etching in progress.



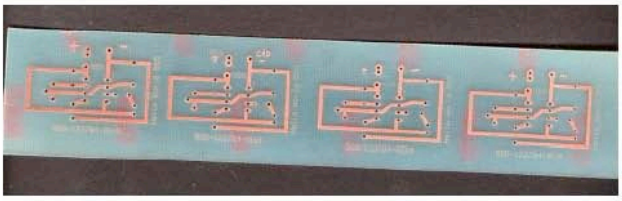
Etched PCB.

of etching. It should take no more than 15 to 30 minutes to do a proper etch. Gently agitate the tray during this time. This will remove bubbles, speed the etching, and make it more uniform. As shown in the middle picture on the facing page, you'll see copper dissolving away from the PC board.

When it cools, remove the board from the tray and thoroughly wash with water to remove all traces of the etchant. Wash again with a little dish soap to help neutralize any remaining acid.

7. CLEAN AND DRILL THE BOARD

Now the copper of the board has to be exposed for soldering. Use an abrasive pad or fine-grain (400 grit) sandpaper to scrub off the black film layer and reveal the copper underneath.



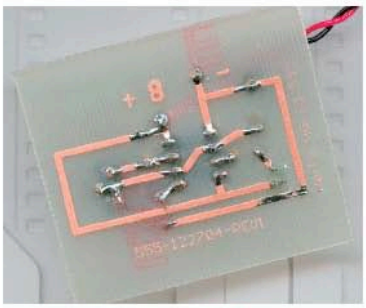
Etched and drilled board.

Drill the holes that the component leads fit into. If you're making many boards, it helps to use a drill press, or at least a power drill or Dremel, because hand-drilling can get tedious. The drill bits should have the same diameter as the etched holes.

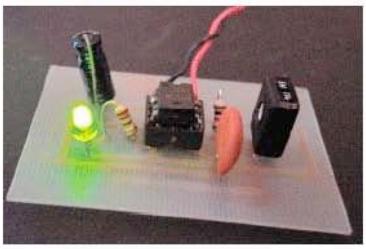
Finally, if you repeated your design, cut the boards apart. Choose the best board to solder the components onto.

8. SOLDER

Use the double-sized parts placement diagram you printed earlier, and solder the parts into the PCB. If the board is an early revision, I always use sockets for any chips, as shown in the bottom photo, so I can remove them without having to desolder. If all is done right, then you've got a complete, functioning circuit on a beautiful little board.



Underside of the completed board.



Homemade PCB blinkie circuit in operation.

FINISH

Andrew Argyle (andrew@argyle.org) has been intrigued with electronics since taking apart his father's power tools. He is currently building Nixie clocks and etching mysterious projects somewhere in Canada.

Advertisement

Learn how you can get 32-bit performance with 8-bit simplicity using Microchip's broad portfolio of 16-bit PIC® microcontrollers and dsPIC® digital signal controllers.

Register today for a half-day seminar near you.

This seminar shows you how the PIC24 and dsPIC33 devices can enable your next design and covers:

- An introduction to the key architectural and peripheral features of Microchip's 16-bit devices
- Software libraries and development tools that reduce your development time and get your product to market quickly
- How to combine peripheral features, architectural features and software libraries to create new designs more easily
- Microchip's three product families, PIC24F, PIC24H and dsPIC33, with detailed application case studies
- Training resources Microchip offers to help you get started solving your next application requirement

Who Should Attend:
Embedded systems designers not familiar with Microchip's 16-bit PIC24 and dsPIC DSC offerings who need to reduce their time to market and require 16/32-bit performance microcontrollers with well supported, easy-to-use powerful development tools.

Discounted Tools & Boards include:
MPLAB® ICD 2 Limited Edition - An all-in-one in circuit debugger/programmer solution the MPLAB ICD 2 Limited Edition is a low cost, real-time debugger and programmer for our entire 16-bit portfolio.

16-bit 28-Pin Starter Development Board - The 16-bit, 28-pin Starter Development Board serves as a development kit and evaluation tool for Microchip's 16-bit microcontrollers and digital signal controllers.

Also included in the standard package are:

- **Free** software libraries
- **Free** MPLAB IDE Integrated Development Environment

Cost:
$25 includes seminar attendance
$99 includes seminar attendance, MPLAB ICD2 Limited Edition and 16-bit 28 pin Starter Development Board

To register today for a 16-bit seminar near you, visit our registration site at www.microchip.com/seminars.

MICROCHIP TECHNOLOGY
EXPLORE
DESIGN
APPLY
REGIONAL TRAINING CENTERS

MICROCHIP

www.microchip.com/seminars