

Inexpensive Printed Circuit Boards

By - lawrence.mazza@stratos.net

After I design a circuit, I like to fabricate it using a printed circuit board because a board offers a lot of advantages. For one, the solder connections are more stable, and for another, well, they look so nice that you want to show them off. There are several ways to make boards including etching, direct plating and copper removal. For the hobbyist, etching is probably the easiest and most cost effective. Etching is the process of chemically removing the unwanted copper from a plated board. You must put a mask or resist on the portions of the copper that you want to remain after the etch. These portions that remain on the board are the traces that carry electrical current between devices. This article will discuss an alternative approach to placing the circuit pattern on the copper clad board.

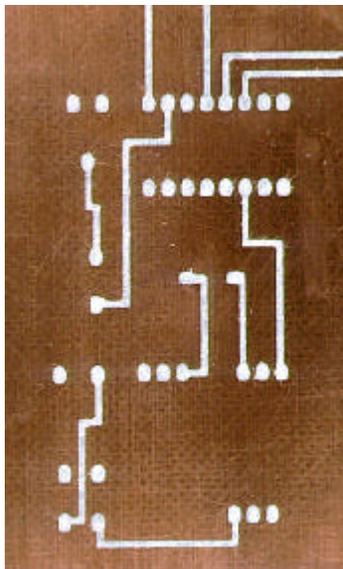
One way to put a pattern on the board is the direct draw approach. A resist pen is used to draw the circuit onto the copper surface. The pen deposits an ink that resists the etchant thereby creating the desired circuit on the board. This is the quickest way to get a circuit pattern on the board, but it is difficult to position the traces accurately, especially if you are using any IC packages in your design. Plus, since the ink doesn't apply uniformly, there is a risk that the traces will be etched away since the etchant can get to the copper through an extremely thin layer of resist.

Another approach is to use a photo sensitive resist. In this case, a board is covered with a resist material that sets up when exposed to Ultra Violet light. To make a board this way, you must make a photo negative of the circuit which is clear where you want a circuit trace, and opaque where you don't want a trace. After the photo negative is made from your art work, it is placed onto the photo sensitized board, and is exposed to the UV. The UV light transmits through the clear portions of the negative and cures the photo resist. After that, the board is submerged into a developer bath that develops the traces on the board. The resist that is left is in the shape of the art work that represents your circuit. The advantage to this approach is accurate and neat traces, and once you make the photo negative, it can be used over and over to make additional boards. However, you need a photo lab to do this, and the entire process takes quite a bit of time.

Now that laser printers are affordable to the common folk, creative people have developed methods of using them to produce the resist pattern. One method uses a specially coated paper and is sold through many electronics outlets. The circuit pattern is printed onto this paper and the paper is ironed onto the copper clad board. The board and paper are placed into water, and after a while, the resist pattern lifts off the paper. This process works a lot like putting decals on model cars, planes and boats. And although I have had mixed results using this paper, it can also be used to make custom decals using a color printer.

I have experimented with other types of paper in search of a replacement for this process. In doing so, I have discovered another, fairly consistent paper that I now use to make all of my printed circuit

boards. This paper is similar to what is used in magazines. It is a white, glossy paper that prints well and costs about three cents per sheet. What makes the paper glossy is also what makes it work in this process. And although it doesn't work exactly like the 'decal' type paper, I find it just as easy to use. The reason the paper is glossy is because it is covered with clay. I'm not sure how it's made, but there is a fine layer of clay that is pressed or worked into the surface of the sheet. You can find this paper at office supply stores - just ask for 60 pound coated paper.



After the pattern is printed onto the paper (I use AutoCad to draw the circuit, by the way), it is ironed onto the copper clad board using a hand iron set to maximum temperature with no steam. This process melts the toner and allows it to stick to the copper. Note that the copper board needs to be very clean in order to make the toner stick (this is true of the decal paper as well). The ironing process only takes a few minutes, and it helps to keep heavy pressure on the iron to keep the toner in contact with the board.

When the board is cool, it is placed into hot water just like the decal stuff. Little by little, the paper saturates with the water, and starts to disintegrate. With a little work, you can gently rub the paper off, and what's left is the toner resist, with a little of the clay also attached to the toner. It is the separation of the clay from the paper that allows the toner to remain on the board. In the picture, you can see the white clay that is still attached to the toner. At this point you can examine the board and fix any holes or cracks using a resist pen. Then the board is etched with Ferric Chloride or Ammonium Persulfate as usual.

This is just another approach to making PC boards that I find the easiest and most cost effective. Try it the next time you make a board, and let me know your experience.

-L