
Subject: Making printed circuit boards

Posted by [FloydV](#) on Tue, 18 Dec 2012 00:05:49 GMT

[View Forum Message](#) <> [Reply to Message](#)

I would like to make some printed circuit boards. The only PCBs I have done were a long time ago where you used a pen to trace the circuit over specially coated copper, then immersed the board in a solution to remove all the copper except where the lines were drawn.

Can someone bring me up to speed on newer processes. I would especially like to design the circuits in a drawing program and somehow get that drawing onto the board.

Thanks,

Floyd

Subject: Re: Making printed circuit boards

Posted by [Wayne Parham](#) on Tue, 18 Dec 2012 00:59:58 GMT

[View Forum Message](#) <> [Reply to Message](#)

That etch resist technique doesn't work too well, does it?

The "newer processes" are actually no different than the older processes when it comes to fiberglass PCB manufacturing. They have offered various products for years, from single-sided and double-sided to multiple layers, with or without a solder mask and/or silkscreen. The solder mask is the green coating that prevents the solder from sticking where it isn't wanted, mostly done for wave soldering but usually there on high-quality boards even if volume is low (and therefore will be hand-assembled). The silkscreen is there to help identify component locations. It's usually done in white, mostly part identification numbers and sometimes outlines of where some parts go.

What is newer is the CAD software used to deliver the artwork to the board house. It used to be done photographically, and your layout engineer would "lay tape". Literally. They used strips of colored translucent tape fastened to large clear sheets of plastic, one per layer. They often used different colored tape so when each layer was laid on top of each other, you could see the layers easily by color coding. But this was just for convenience while developing and checking the layout, because the final product of the layout engineer is a photographic positive done in black on clear at scale for each layer requested.

Now days, this is usually done electronically by sending a CAD model of the layout in a standard Gerber format. You can design the board using OrCAD or something similar, and then send the Gerber file to the board house for quotations. They usually have an inexpensive prototype trial run these days (which they didn't have in decades past), so you ask for a relatively small number. If you approve the quote, they'll use the Gerber file to make the boards and send 'em to you. The inexpensive trial runs are nice, because back in the day, you didn't have that luxury so you better get it right first time. Otherwise, you had "Engineering Change Orders" (ECOs) that essentially meant you cut a trace and if it needs to be rerouted, you soldered a small wire-wrap wire onto the board as a replacement conductor. It could be a bear to find a problem in a first-run too,

especially in complex designs.

Subject: Re: Making printed circuit boards
Posted by [FloydV](#) on Tue, 18 Dec 2012 04:32:40 GMT
[View Forum Message](#) <> [Reply to Message](#)

Thanks Wayne. By inexpensive trial run, how much is inexpensive?

To start with, I wouldn't make anything more complicated than say a crossover board. Or maybe a radio wave door bell to remote bright light in the theater room.

Is the old pen and mask still available to play with?

Floyd

Subject: Re: Making printed circuit boards
Posted by [Wayne Parham](#) on Tue, 18 Dec 2012 15:25:52 GMT
[View Forum Message](#) <> [Reply to Message](#)

Naturally cost depends on board size and options. Options include number of layers, holes with or without plate-through, solder mask or silkscreen.

For a crossover board, you probably only need a single layer and you don't need a solder mask or silkscreen. You can do the silkscreen if you want though. You can do the solder mask too, for that matter, but there would be absolutely no benefit except it looks pretty.

By inexpensive, I mean less than a thousand bucks. That is a lot of money compared to lunch, but it isn't a lot of money compared to how much it used to be.

They do make those kits to etch boards using a etchant resist pen. A step up would be the ones that use a film that you print on using a laser printer for the etchant resist. Might try that too.

Here's one guy's take on that method:

[How To Make A Basic PC Board By Direct Etching](#)

Subject: Re: Making printed circuit boards
Posted by [gofar99](#) on Wed, 19 Dec 2012 01:31:54 GMT
[View Forum Message](#) <> [Reply to Message](#)

Hi Something to add to all that is there are a number of software packages that will do board design....but if the output isn't a Gerber file the fabricator will probably have a fee for the the

conversion to a Gerber file. It can be more than the cost of the boards.

Subject: Re: Making printed circuit boards
Posted by [FloydV](#) on Mon, 24 Dec 2012 23:52:45 GMT
[View Forum Message](#) <> [Reply to Message](#)

I found these two links helpful:

<https://www.smallbearelec.com/HowTos/DirectPCBoards/DirectPCBoards.htm>

<http://www.robotroom.com/PCB.html>

I found the last link a bit more helpful, but the ExpressPCB program he mentions reminds me of AutoCAD. You can play with it for twenty minutes and not have a single useable thing on the screen.

Each link uses a slightly different chemical bath for an etching solution. Anyone like one more than the other?

Floyd

Subject: Re: Making printed circuit boards
Posted by [FloydV](#) on Mon, 24 Dec 2012 23:57:54 GMT
[View Forum Message](#) <> [Reply to Message](#)

I think I just listed your link in another post. I know I can get chemicals from someone like Scientific Products or Sergeant-Welch, but they probably won't sell less than 500 grams. Do you have any favorite supplier?

Floyd

Subject: Re: Making printed circuit boards
Posted by [gofar99](#) on Tue, 25 Dec 2012 15:15:04 GMT
[View Forum Message](#) <> [Reply to Message](#)

FYI, we use Express PCB to do the boards for our products (Oddwatt Audio). It works fine and the board maker will do the translation to Gerber for us. They generally are not up for doing much under 25 pcbs at a time though.
