

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>

How to Create Printed Circuit Boards

So you have that circuit designed and ready. You did some computer aided simulations and the circuit is working great. Only one thing left! You need to create a printed circuit board for the circuit so you can see it in action! Whether your circuit is a project for school/college or is a final piece of electronics in a professional product for your company, implementing your circuit on a PCB will give it a much better professional look besides giving you an idea of how the finished product will look like!

This article will show you the different methods by which you can create a printed circuit board for an electrical/electronic circuit using different methods suitable for small to large circuitry.

Steps

1. Choose a method to use for creating the PCB. Your choice will usually be based on the availability of materials needed by the method, the technical difficulty level of the method or the quality of PCB you desire to obtain. Here's a brief summary of the different methods and their main features that will help you decide:

1. Acid etching method: this method requires extreme safety measure, the availability of many materials such as the etchant and it is somewhat slow. The quality of PCB obtained varies according to the materials you use but generally, it is a good method for simple to intermediate levels of complexity circuits. Circuits involving more close wiring and tiny wires usually use other methods.

2. UV etching method: this method requires more expensive materials that might not be available everywhere. However, the steps are simple; it requires less safety measures and can produce finer and more complicated circuit layouts.

3. Mechanical etching/routing method: this method requires special machines that will mechanically etch away unnecessary copper from the board or route empty separators between wires. It can be expensive if you intend to buy one of those machines and usually leasing them requires the availability of a workshop nearby. However, this method is good if you need to create many copies of the circuit and also can produce fine PCBs.

4. Laser etching method: this is usually used by large production companies, but can be found on some universities. The concept is similar to mechanical etching but LASER beams are used to etch the board. It is usually hard to access such machines, but if your local university is one of the lucky ones having such machine, you can use their facilities if they allow it.

2. Create the PCB Layout of your circuit. This is usually done by converting your circuit's schematic diagram into a PCB layout using PCB layout software. There are many open source software packages for PCB layout creation and design[3], some are listed here to give you a head-start:

* PCB

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>

* Liquid PCB

* ShortCut

3. Make sure you gathered all the materials needed by the method of your choosing.
4. Draw the circuit layout on the copper coated board. This is only applicable in the first two methods. More details can be found on the detail section of your method of choice.
5. Etch the board. Look for the details sections for how to etch the board. This process removes any unnecessary copper from the board leaving only wiring of the final circuit.
6. Drill mount points. Drilling machines used for that are usually custom machines designed specifically for this purpose. However, with some adjustments a usual drilling machine will do the job at home.
7. Mount and solder the electronic components on board.

Acid etching method specific steps

1. Choose your etching acid. Ferric chloride is a common choice for an etchant. However, you can use Ammonium Persulfate crystals or other chemical solutions. No matter what choice for the chemical etchant, it will always be a dangerous material, so besides following the general safety precautions mentioned in this article, you should also read and follow any additional safety instructions that come with the etchant.

2. Draw the PCB layout. For acid etching, you need to draw the circuitry using an etchant resistant material. Special markers can be found easily for this specific purpose if you intend to do the drawing by hand (not appropriate for medium to large circuits). Laser printers' ink is the most commonly used material however. The steps to use laser printers for drawing the circuit layout is as follows:

1. Print the PCB layout on a glossy paper. You should ensure the circuit is mirrored before doing that (most PCB layouting programs have this as an option when printing). This only works using a laser printer.
2. Put the glossy side, with the printing on it, facing the copper.
3. Iron the paper using an ordinary clothes iron. The amount of time this will take depends on the type of paper and ink used.
4. Immerse the board and paper in hot water for a few minutes (up to 10 minutes).
5. Remove the paper. If certain areas seem particularly difficult to peel off, you can try soaking a bit more. If everything went well, you will have a copper board with your PCB pads and signal lines traced out in black toner.

3. Prepare the acid etchant. Depending on the etchant you choose, there might be additional instructions. For example, some crystallized acids require being dissolved in hot water, other etchants are ready to use.

4. Submerge the board in the acid.

5. Make sure to stir every 3-5 minutes.

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>

6. Take the board out and wash it when all unnecessary copper is eaten away from the board.

7. Remove the insulating drawing material used. There are special solvents available for almost all types of insulating drawing material used in drawing PCB layouts. However, if you don't have access to any of these materials, you can always use a sand paper (a fine one).

Ultra-Violet etching method specific steps

1. Draw the PCB layout on the special copper coated board.
2. Cover the board with a transparent sheet (optional)
3. Put the board in the UV etching machine/chamber
4. Turn on the UV machine for the specified amount of time depending on the specification of the board and machine.

Warnings

- * If you are using the acid etching method, you need to take the following precautions:
 - o Always store your acid in a safe cold place. Use glass containers.
 - o Label your acid and store it somewhere out of the reach of children.
 - o Do not dispose of your used acid in the home drains. Instead store it and when you have some amount of used acid, take it to your recycling center/dangerous waste disposal facility.
 - o Use gloves and air masks when working with acid etchants.
 - o Be extremely careful when mixing and stirring the acid. Do not use metallic objects and do not put the container on the edge of the disk.

By Engr. Mansoor Habib

Administrator: <http://www.helpingnotes.com/>