

Process of PCB manufacturing

Maciej Pietras, Patryk Strankowski and Jarosław Guziński

1. What do we need?

- Coated copper clad
- NaOH or solution on the base of it
- UV lamp
- Transparency Sheets
- Drill
- UV Permeable piece of glass or plastic

2. Steps.

a) Designing the “paths” in Eagle and printing it.

This point requires skills of using the software. After creating the layout and introducing the frame around, it is printed.

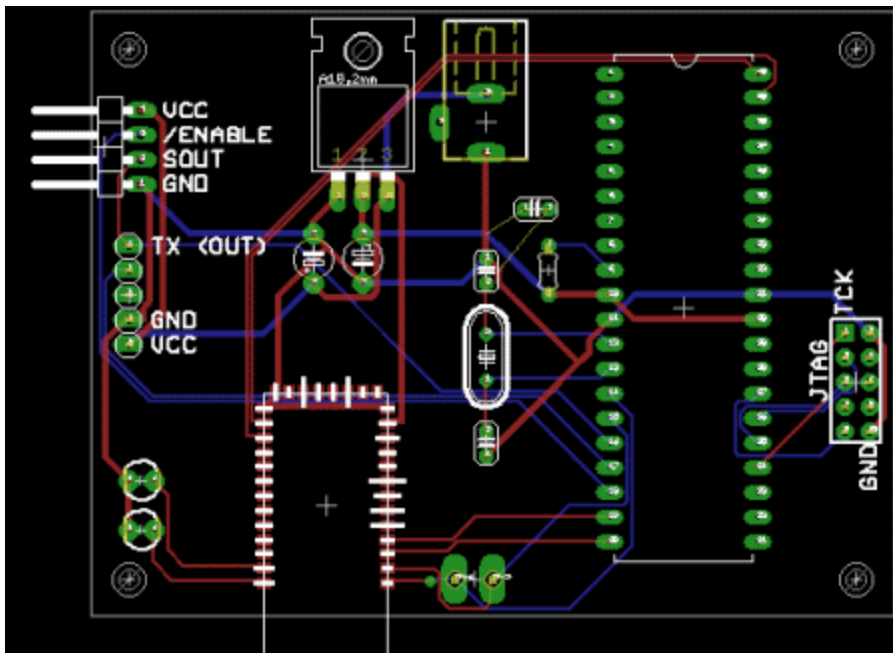


Fig.1: <https://www.build-electronic-circuits.com/pcb-design/>

Sometimes during aligning there is a possibility of light leaking in sideways and causing the edges of your copper to be “fuzzy” which can result in poor etching. This is the reason why during printing top layer selecting option ‘MIRROR’ next to ‘BLACK’ is advisable. Thanks to that, actual layer will be as close to the PCB as possible and the problem described above will be avoided. Later, while printing the second layer on different sheet, option ‘MIRROR’ shouldn’t be switched on.

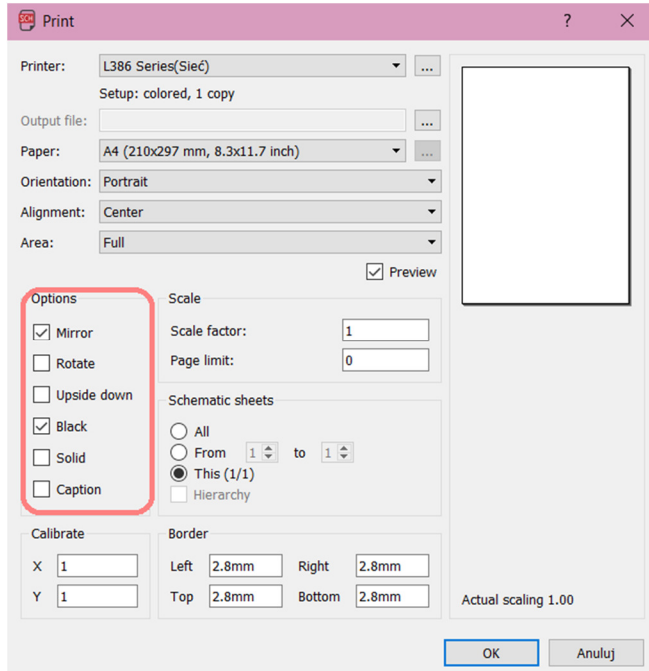


Fig. 2: View From Eagle program.

b) Preparing the board.

This step is about adjusting the size of the board to our needs. It's very important to be careful and not to scratch the PCB by the cutting tool. All process produces a lot of fiberglass powder so wearing a protective mask would be a good idea.

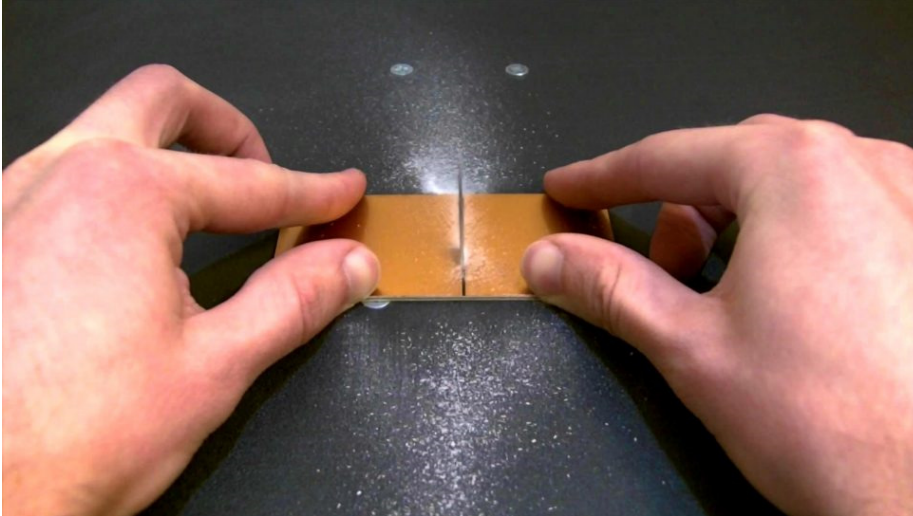


Fig. 3: <http://indexxit.com/pcb-cutter/stunning-pcb-cutting-jigsaw-table-diy-in-action-cutter-machine/>

c) Settling layers on the board.

The goal of this task is to line up the two printed layers. Theoretically this step requires a lot of precision and focus but drilling two reference holes through the artwork would be a huge facilitation. These holes can then be used to align the other layer. The farther apart they are, the easier it is to prevent rotational misalignments.

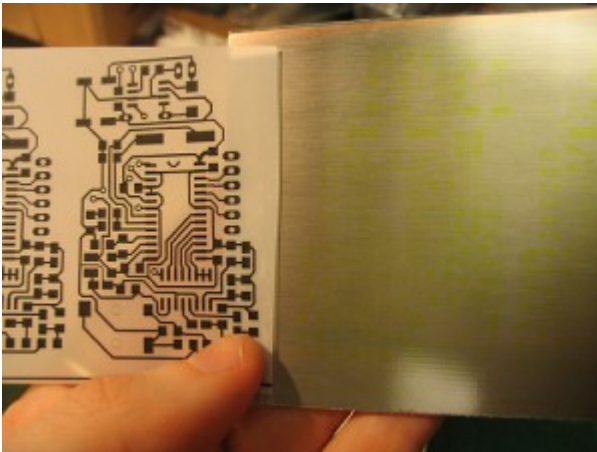


Fig. 4: <https://provideyourown.com/2012/make-your-own-pcb-part1/>

d) Exposure on UV light.

This step is undoubtedly the most time consuming one of all. The first step is to remove the protective layer of one side of the PCB. It's advisable to do this in dark place because contact with daylight or any other source of UV light would give some undesired effects.

Then, use some scotch tape to connect the layer with the PCB and place it under UV lamp and wait approximately 15 minutes. Later, do this process again on the other side, remembering about aligning the layer with the holes drilled before.



Fig. 5: <https://www.triplespark.net/elec/design/pcb/>

e) Developing the PCB.

This time we need to prepare a solution of PCB developer. Concentration of the solution depends on the type of NaOH carrier and producer of the plate. Having prepared the solution, put the plate there and rub it with a glove until copper vias will appear clearly.

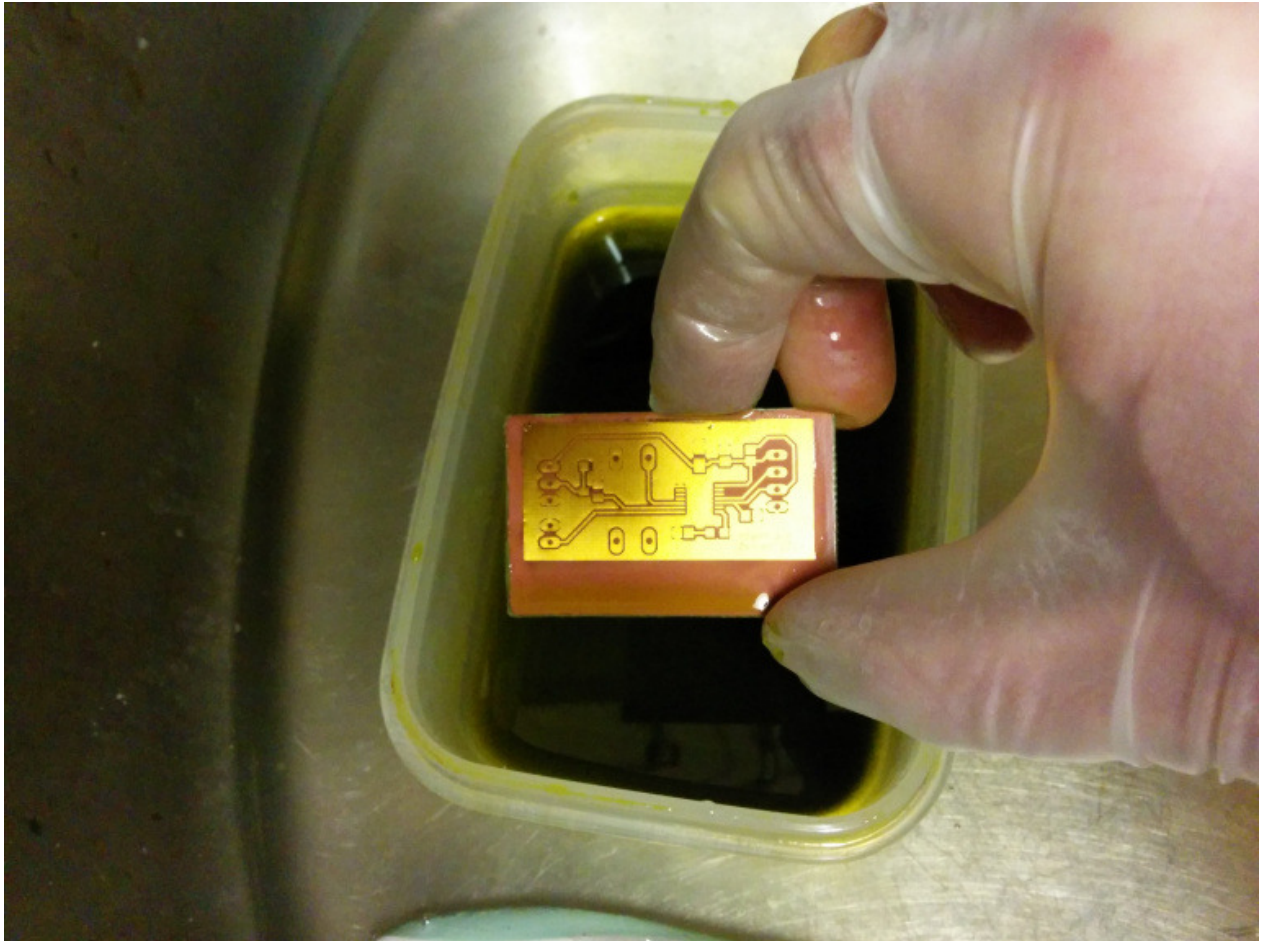


Fig. 6: <https://Oxfred.wordpress.com/2014/02/28/further-pcb-improvements/>

f) Etching

In this step, we can get rid of all unwanted copper. There are plenty of methods of etching and they require different machines and chemicals so mentioning all of them would take too much space.

Now, when we are almost finished we need to clean the board after etching with the water. Finally, the next step is drilling holes.

Now our job is done and the board is ready to solder out.

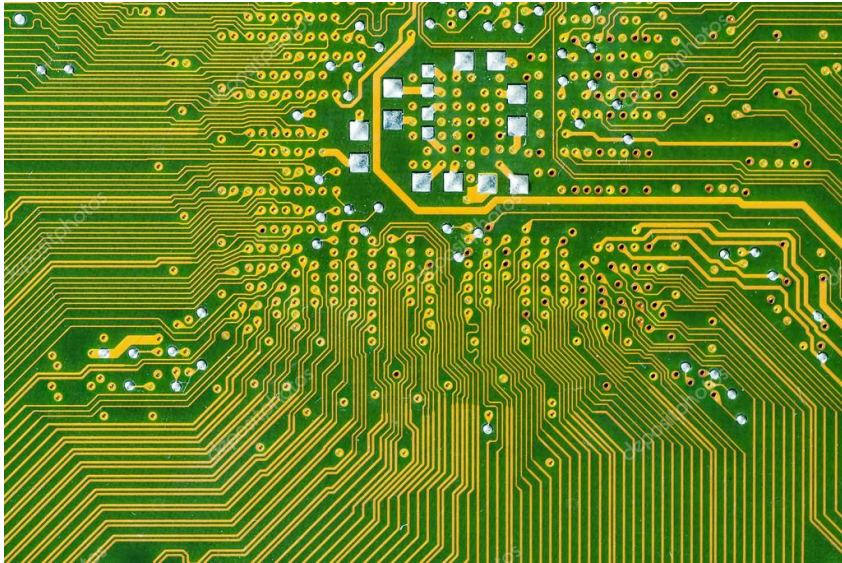


Fig. 6: <https://pl.depositphotos.com/38037577/stock-photo-pcb-motherboard.html>