National Polytechnic Institute
Superior School of Computer Sciences

Analog Electronics.

Practice 10: PCB.

Hernandez Martinez Carlos David.
Cruz Medina Isaac Abraham.
Herrera Albavera Luis Enrique.

Group: 2cv3.

December 5, 2017

Contents

1 Introduction:

The PCB design and layout forms an integral part of the design of the whole product, and it can be the key to the success of the product meeting its performance requirements in many instances. PCB technology has progressed significantly in recent years. The design technology has improved with PCB CAD systems and PCB software to layout the boards better, and also analyses the performance under conditions such as the operation at high frequencies. In addition to this the technology for the manufacture of PCBs has improved enabling far smaller tracks to be used as well as incorporating features such as multilayer boards with blind vias, etc.

In the early days of PCBs, the design and layout was undertaken manually using tape that was placed onto a translucent film. These PCB designs were normally four times the required size and they were photographically reduced onto a 1 to 1 film for the PCB production process. Nowadays the PCB design process has been computerised and there are many PCB CAD systems and PCB software packages that enable the PCB layout and design to be undertaken more efficiently than before. The PCB software varies considerably in price. Budget, or even free software provides the basic functions, whereas the top end packages enable many more facilities to be incorporated into the design. Simulations, complex routing, and many more facilities are available.

The first stage in the development of a PCB design is to capture the schematic for the circuit. This may be achieved in a variety of ways. Circuits may be entered into a schematic capture tool. This may form part of the PCB design suite, or it may be an external package whose output can be exported in a suitable format. In addition to purely performing the schematic capture, simulations of the circuit may be undertaken at this stage. Some packages may be able to interface to simulation packages. For applications such as RF circuit design simulation of the circuit will enable the final circuit to be optimised more without building a prototype. With the schematic capture complete the electronic design of the circuit is contained within the file and can be converted to what is termed a "netlist". The netlist is the interconnectivity information and it essentially the component pins and the circuit nodes, or nets, to which each pin connects. essentially the component pins and the circuit nodes, or nets, to which each pin connects.

Before proceeding with the detailed PCB design and layout, it is necessary to gain a rough idea of where components will be located and whether there is sufficient space on the board to contain all the required circuitry. This will enable decisions about the number of layers needed in the board, and also whether there is sufficient space to contain all the circuitry may need to be made. Once a rough estimate has been made of the space and approximate locations of the components, a more detailed component layout can be made for the PCB design. This can take into account aspects such as the proximity of devices that may need to communicate with each other, and other information pertaining to any RF considerations for example. In order that components can be incorporated into the PCB design they must have all the relevant information associated with them. This will include the footprint for the printed circuit board pads, any drilling information, keep out areas and the like. Typically several devices may share the same footprint, so this information does not have to be entered for each component part number. However a library for all the devices used will be built up within the PCB layout design system. In this way components that have been used previously can be called up easily.

Once the basic placement has been completed, the next stage of the PCB design is to route the connections between all the components. The PCB software then routes the physical connections on the board according to the netlist from the schematic. To achieve this it will use the number of layers that are available for connections, creating via holes as required. Often one layer will be allocated for use as a ground plane, and another as a power plane. Not only does this reduce the level of noise, but it enables low source resistance connections to be made for the power. The routing can use a significant amount of computing power. This is particularly true for larger designs where there may be upwards of three or four thousand components. Where routing is difficult as a result of high component density, this can result in the routing taking a significant amount of time.

2 Objective:

- The student must know the different ways to create a *board-circuit*.
- By the end of the practice the student must know how to make *board-circuits* by using the *Acid Etching Method*.

2.1 Material:

- Etching Solution (Ferric Chloride).
- PCB Layout.
- PCB Board.
- Fine Typed Marker.
- Magazine/Glossy Paper.
- Sanding Paper.

2.2 Tools:

- Flat Iron.
- Laser Printer.
- Cutter.
- Wood Board.
- Plastic Tweezers.
- Ruler.

3 Development:

After starting the etching process to make our circuit board we need to cut the copper plate at the size of the *layout*. In our case we needed of *10mm* of height per *6mm* of width approximately. So, using the cutter we *mark* the copper plate several times like in Figure 3.0 until the copper plate split easily (Figure 3.1). The we can start the etching process.

Figure 3.0: Cutting the cooper plate.

Figure 3.1: Splitting the cooper plate.

3.1 Creating The PCB Layout:

For acid etching, we needed to draw the circuitry using an etchant resistant material. Special markers can be found easily for this specific purpose if we intend to do the drawing by hand (not appropriate for medium to large circuits). But laser printers' ink is the most commonly used material however. This is usually done by converting our 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 like *Proteus* or *Multisim* or *Liquid PCB*. In Figure 3.1.0 we can find the layout that we are going to work with.

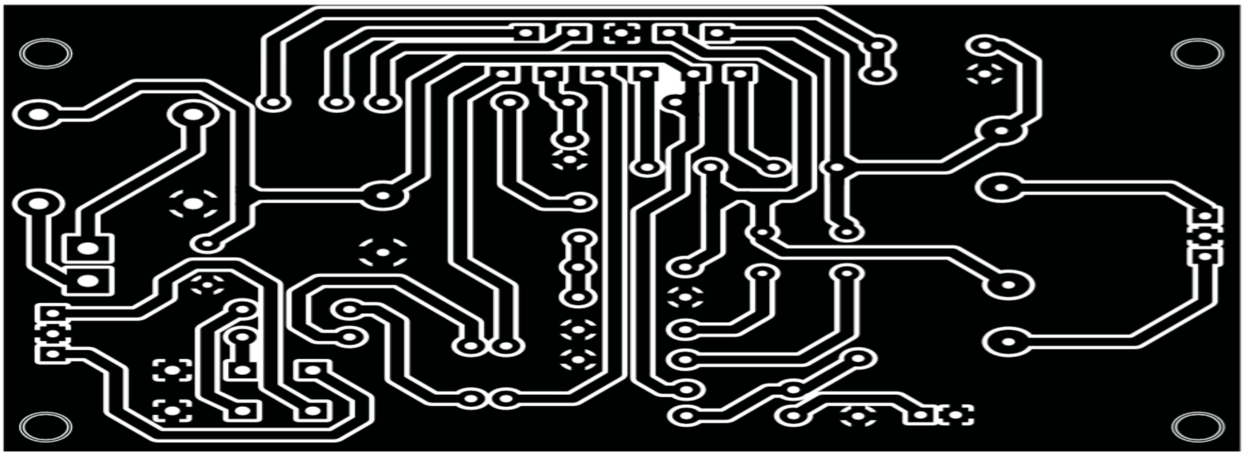

Figure 3.1.0: PCB Layout.

3.2 Printing The Layout:

Once we were happy with our schematic on our computer, we match the size of the diagram on the software so that both the circuit board and the paper will have the needed sizes. For our circuit the sizes are *9mm* of width and *5.7mm* of height.

We print it out on a glossy paper, such as magazine paper. We ensure the circuit is mirrored before doing this (most PCB layout programs have this as an option when printing). Once printed, we make sure to do not touching the ink part on the paper as it can get on our hands. Then, we align the circuit diagram on the paper with the circuit board (the diagram should be facing the copper part of the circuit board) as we can see in Figure 3.2.0 and Figure 3.2.1. We start up our iron and wait until it heats up. Once heated, we carefully place the iron for around 30-45 seconds on top of the paper which is on top of the circuit board (Figure 3.2.2 and Figure 3.2.3).

Figure 3.2.0: Aligning the circuit diagram and the circuit board.

Figure 3.2.1: Aligning the circuit diagram and the circuit board (real).

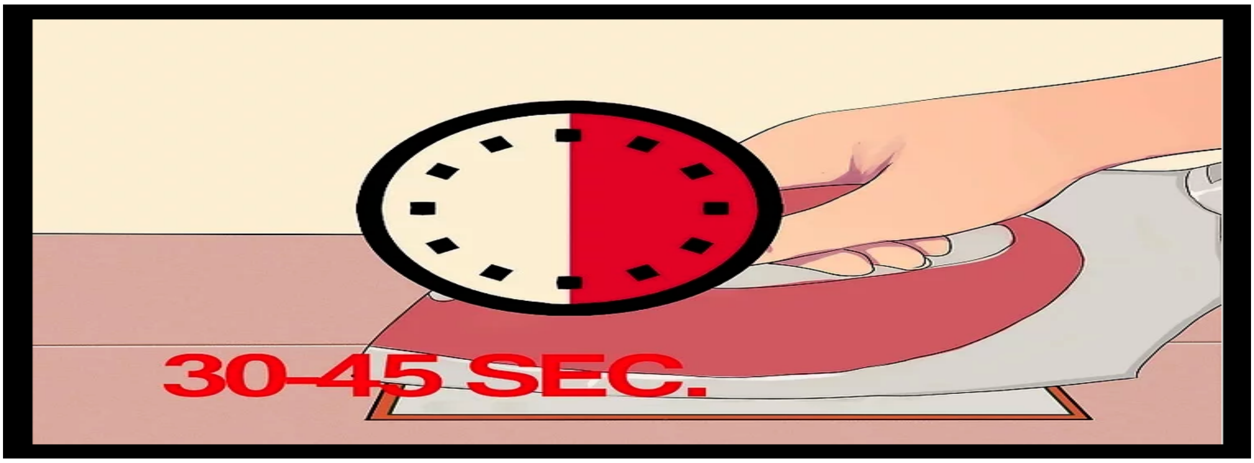

Figure 3.2.2: Placing the iron on the circuit board.

Figure 3.2.3: Placing the iron on the circuit board (real).

3.3 Cooling The Circuit Board:

After removing the iron from the top of the circuit board with our fingers we make some pressure all around the circuit board just to make sure that the ink it's successfully transfer (Figure 3.3.0) then, we take the circuit board to a source of water and we hold the circuit board below it. An alternative approach is to immerse the board and paper in hot water for a few minutes (up to 10 minutes like in Figure 3.3.1 and 3.3.2).

Figure 3.3.0: Pressing the circuit board to make sure that the ink it's successfully transferred.

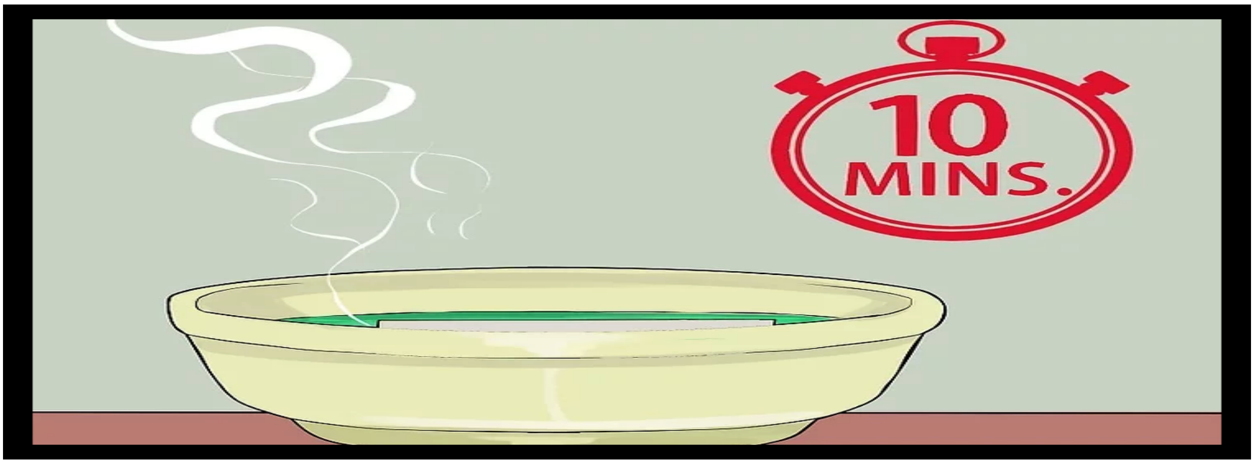

Figure 3.3.1: Immersing the circuit in water.

Figure 3.3.2: Immersing the circuit in water (real).

Slowly we start to take off the paper and soon all of the paper should come off. Finally, we have a copper board with our PCB pads and signal lines traced out in black toner like Figure 3.3.3

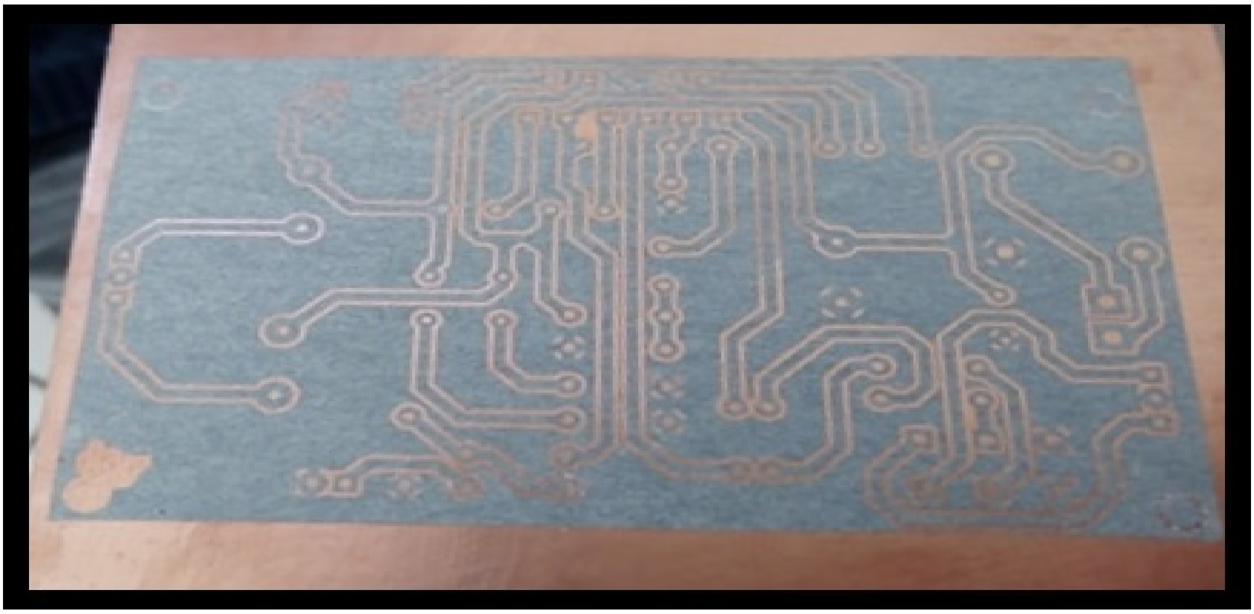

Figure 3.3.3: PCB with the ink placed.

3.4 Etching With Acid:

Ferric chloride is a common choice for an etchant (this is the one that we use). However, we can use Ammonium Persulfate crystals or other chemical solutions as we can see in Figure 3.4.0. No matter what choice for the chemical etchant, it will always be a dangerous material, so we need to be careful.

Figure 3.4.0: Acids for Etching.

Depending on the acid etch that we choose, there might be additional instructions. For example, some crystallized acids require being dissolved in hot water, but other etchants are ready to use. So, ones the acid were prepared, we submerge the board in it as in Figure 3.4.1 and we make sure to stir every 3 or 5 minutes like in Figure 3.4.2.

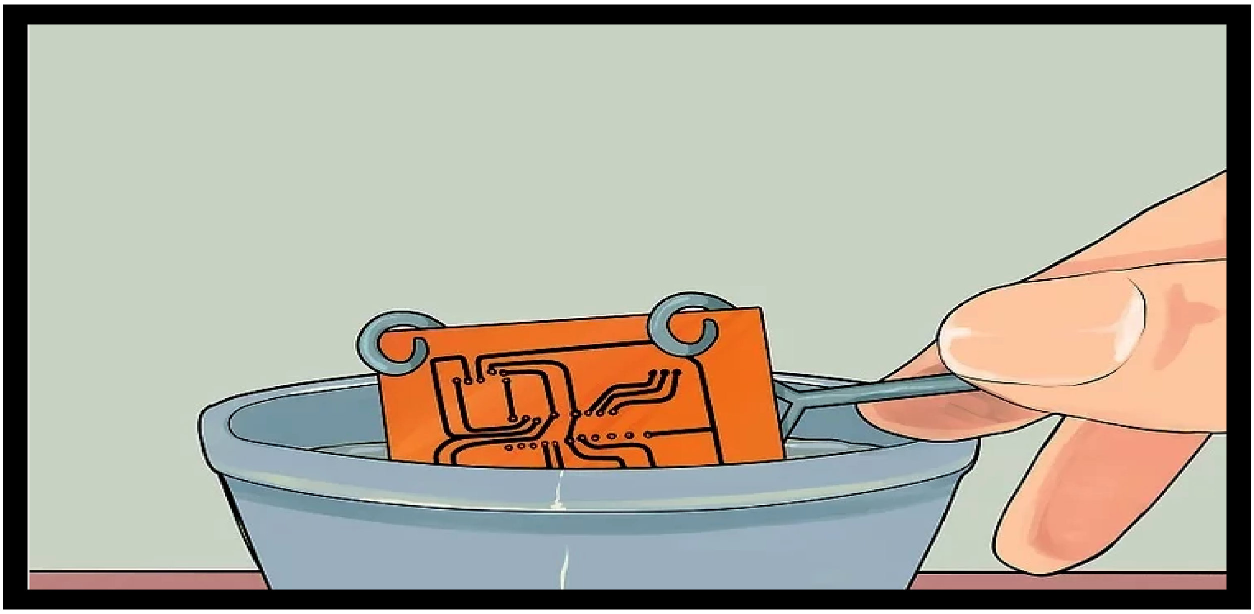

Figure 3.4.1: Submerging the board.

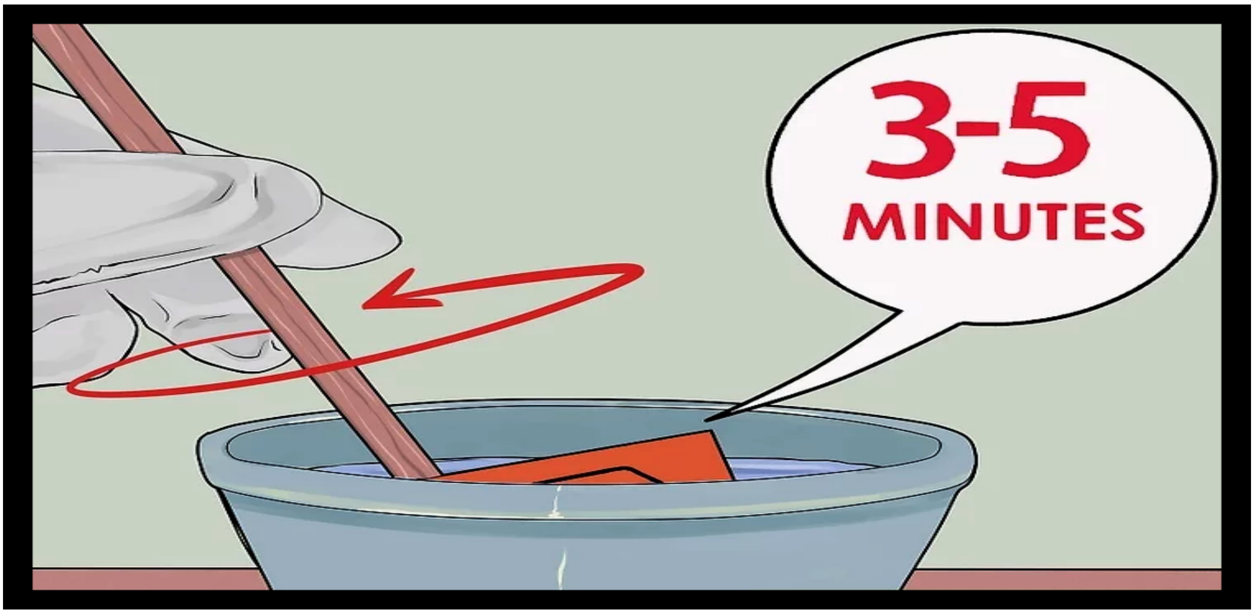

Figure 3.4.2: Stir every 3-5 minutes.

In Figure 3.4.3 we already submerge our circuit board.

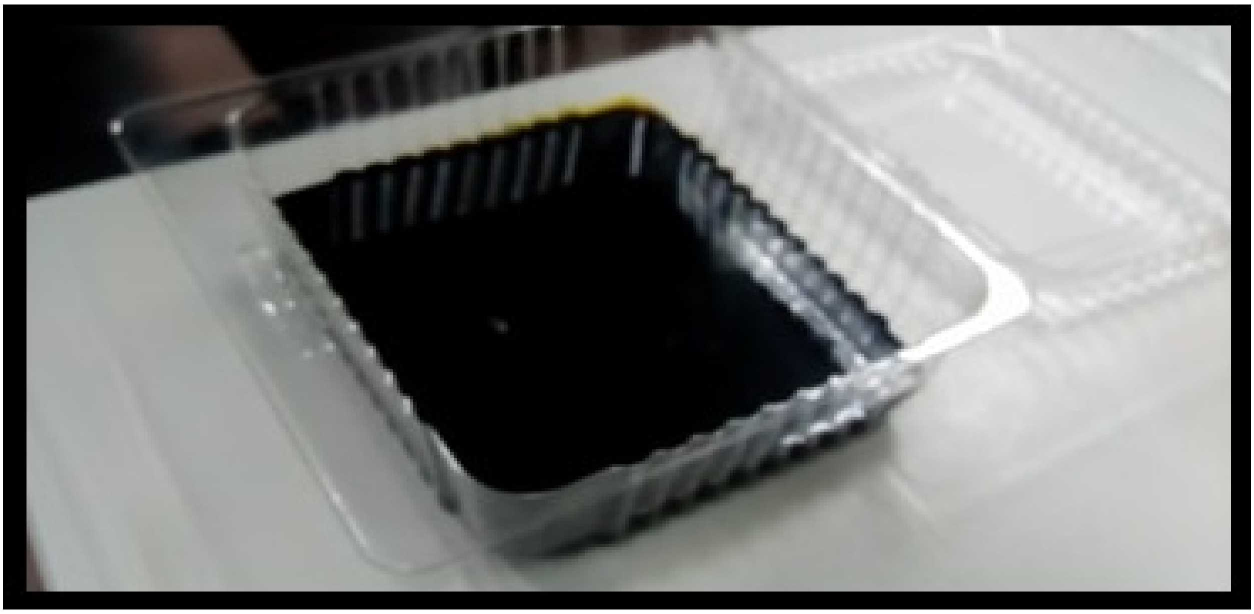

Figure 3.4.3: Circuit board submerged.

Finally, we take the board out and wash it when all unnecessary copper is etched away from the board (Figure 3.4.4).

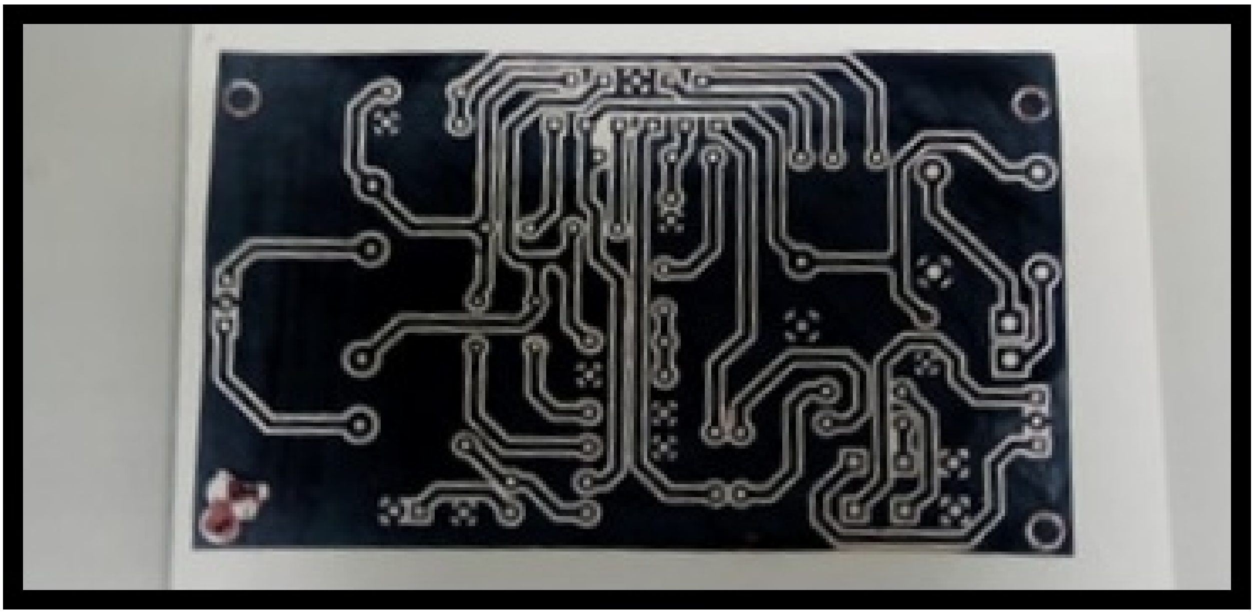

Figure 3.4.4: Circuit board cleaned after taked it out of the acid.

There are special solvents available for almost all types of insulating drawing material used in drawing PCB layouts. However, we use sandpaper and our PCB it's finish as we can see in Figure 3.4.5.

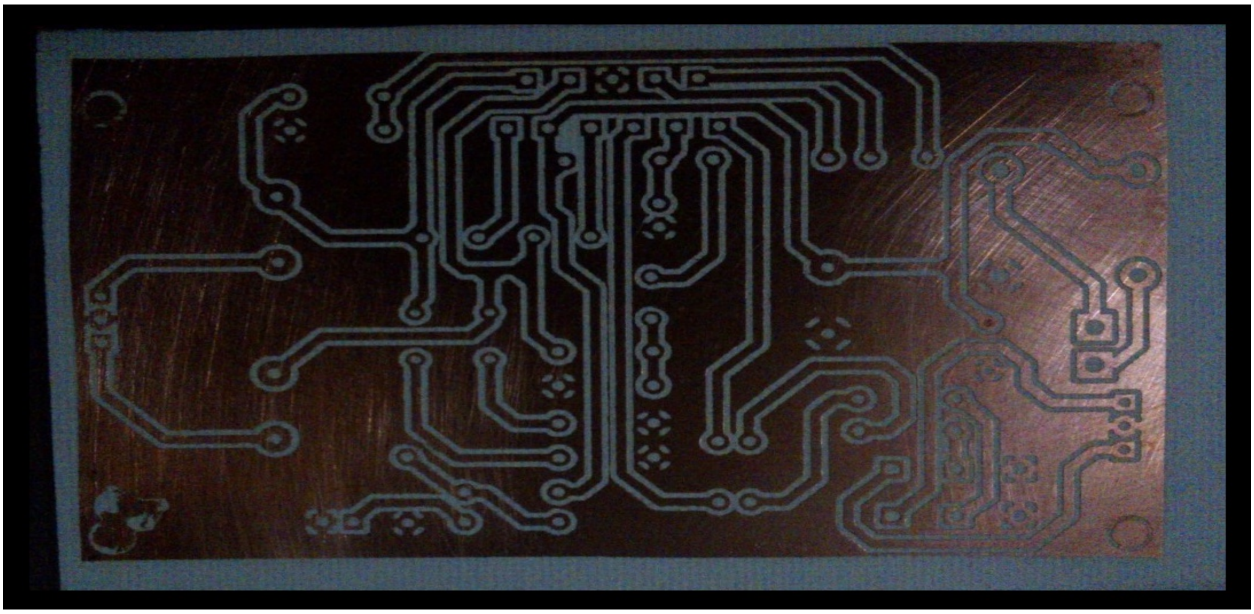

Figure 3.3.5: PCB finished.

4 Conclusion:

Today there is an upsurge in the number of electronic gadgets being used. Our world is practically surrounded by number of electronic devices each day and every one of these devices has a printed circuit board at its heart. So what exactly is this printed circuit board or PCB? It is nothing but a thick non conductive substrate sheet with copper tracks carved on it instead of wires. These tracks join at different points and nodes. This PCB is used to connect various electronic parts through the designed pathways and to provide physical support to the components. PCBs can be through-hole in which tiny holes are drilled on the nodes. The holes are used to fix the electrical component in position and are then joined on the opposite side and the copper tracks link them together forming a circuit. The board and the circuit are jointly called PCB assembly. This is an old technology and is used for low-priced circuitry applications and for supporting large components like transformers and high rating capacitors. Another type of PCB is on board SMD PCB or surface mounted device PCB. Instead of holes it has tiny conducting beds where the small SMD components are placed and soldered using special equipment. These boards have higher strength and both sides can be used for components.

Printed circuit boards have become an integral part of all kinds of electronic components. These devices have blended so well with our lives that we do not even comprehend how important they are, not only for our expediency but for our survival. PCBs can be used for absolutely everything. There's virtually no electronic device that cannot be built using PCBs. Whether it is battery operated toys, cell phones, music players, TVs or automated aircraft. PCBs are there in most of the electronics that we use in our everyday. Other uses of PCBs include military, medical and industrial components. The use of PCBs offers many benefits; first being its smaller size in comparison to other types of circuits. Moreover a printed circuit greatly reduces the need of wires.This in turn improves the appearance of final circuit and increases its durability. It's the cheapest method for mass producers ordering in large production as it offers huge cost benefits.

Some of their benefits are:

- PCBs are cost efficient and highly reliable.
- They are economical specialize for high volume production.
- Easy to install, some of they have color codes for different connections.

The design of printed circuit boards is very specific and of high quality. This is the reason why designing of these boards is a job of an expert and requires a lot of research and development to ensure that there are no flaws in it.

5 Bibliographic References:

[1] BOYLESTAD, Robert L. "Electronic Devices and Circuit Theory". Edit. Prentice Hall. 2009.