

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/267029241>

# A Laboratory Method for Printed Circuit Board manufacturing.

Chapter · April 2013

DOI: 10.13140/2.1.2221.3124

CITATIONS

0

READS

9,313

3 authors, including:



**Vasuki Soni**

Xilinx Inc.

4 PUBLICATIONS 8 CITATIONS

[SEE PROFILE](#)



**Rounak Singh Narde**

Rochester Institute of Technology

14 PUBLICATIONS 157 CITATIONS

[SEE PROFILE](#)

Some of the authors of this publication are also working on these related projects:



On-chip antennas [View project](#)



Millimeter-Wave Wireless Interconnect [View project](#)

# **Making Printed Circuit Board**

## **Laboratory Manual**

**Dept. Of  
Electronics & Telecommunication  
Engineering**

**National Institute of Technology , Raipur**

**Made By:**

Vasuki soni   S Vinayshree   Mordhwaj patel   Rounak singh narde

Batch : 2009-13

# I. Introduction to Printed Circuit Board

---

## WHAT IS PCB?

**A** Printed Circuit Board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or signal traces etched from copper sheets laminated onto a non-conductive substrate. It is also referred to as printed wiring board (PWB) or etched wiring board. Printed circuit boards are used in virtually all but the simplest commercially produced electronic devices.

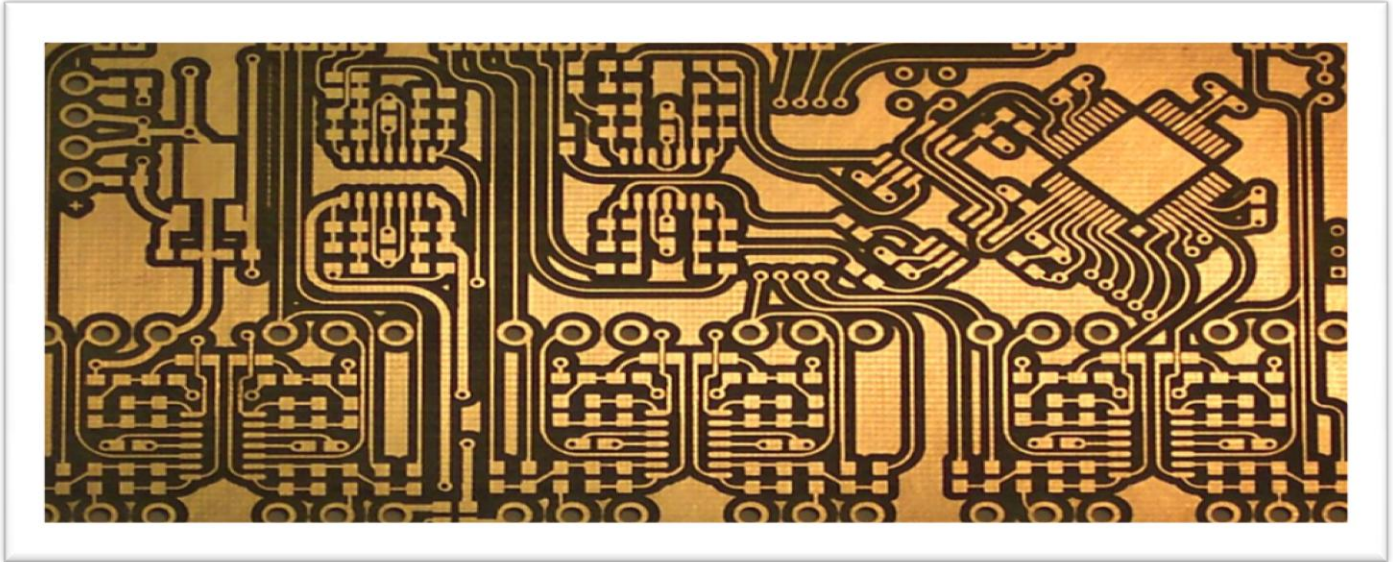


Figure – 1 Printed Circuit Board

A PCB populated with electronic components is called a printed circuit assembly (PCA), printed circuit board assembly or PCB Assembly (PCBA). In informal use the term "PCB" is used both for bare and assembled boards, the context clarifying the meaning.

### Process of Making a PCB:

1. Layout Design
2. Negative film development for the circuit
3. Copper Clad Preparation
4. UV-exposure of copper clad and its developing
5. Drilling
6. Etching

## II. Layout design of the circuit

---

1. Design the Layout of PCB, using **CAD** (Computer aided drafting) softwares like **EAGLE**, **PROTEUS** etc.
2. Take a print out of one Layer (generally bottom layer) Layout with pads & tracks.

NOTE: Actual size must be printed. Check the page setting and select A4 type. If the pcb is single layer and you are printing bottom layer of it, mirror of the layout is to be printed.

It is recommended to take a sample printout to check if all the dimensions are correct.

3. Take printout of the layout on the rough side of the trace paper. Rough side of trace paper gives you better quality.

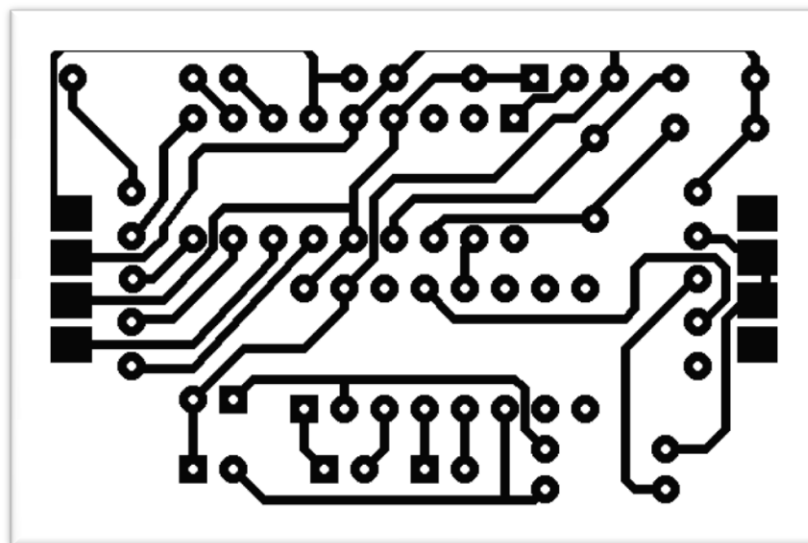


Figure – 2 Printed Circuit Board Layout



### III. Requirements of the laboratory process

---

#### 1. Chemicals Required:

- Chemical A – **Sodium Carbonate.**
- Chemical B - **Metol (monomethyl-p-aminophenol hemisulfate), phenidone (1-phenyl-3-pyrazolidinone) or dimezone (4,4-dimethyl-1-phenylpphenylpyrazolidin-3-one) etc are Hydrocarbons.**
- Photo Resist Chemical – **Poly Methyl Methacrylate (PMMA).**
- Developer (For Copper Clad Developing) - *The UV Exposed Copper Clad sheet in PCB Making process is developed with a 1% solution of sodium hydroxide NaOH. You can make this solvent by adding 10 gram of sodium hydroxide pellets to 1 litre of water and mix it until everything is dissolved.*
- Negative Developer Solution: *Prepare a solution of chemical A (1 spoon) + chemical B (1spoon) & 1.5 litres of water in a tray No. 1.*
- Water: *Fill 1.5 litres in tray No. 2.*
- Fixer Solution: **Sodium Thio-Sulphate.** *Prepare a solution of 2 cap of fixer & 1 litre of water in tray No. 3.*
- Blue dye.
- Thinner.

#### 2. Hardware required :

##### a. Printer



Figure –3 Printer

- ##### b. Permanent marker (for correcting the PCB printout or in conditions when the print of the design on the copper sheet gets rubbed off due to human errors).

##### c. Trace Paper



Figure – 4 Trace Paper

**d. Hand Gloves**



Figure – 5 Hand Gloves

**e. Plier**



Figure – 6 Plier

**f. Scissor**



Figure – 7 Scissor

**g. Copper Clad Sheet (For PCB Making)**



Figure – 8 Copper Clad Sheet

**h. Proto Contact film making machine**



Figure – 9 Proto Contact Film Making Machine

**i. Heater**



Figure – 10 Heater

j. Photo Resist dip coating machine



Figure – 11 Photo Resist Dip Coating Machine

k. Ultra Violet exposure chamber



Figure – 12 Ultra Violet exposure chamber

l. Art work table (For intermediate troubleshooting)



Figure – 13 ART WORK Table



**m. Drilling machine**



Figure – 14 Drilling machine

**n. Cutting machine (for cutting the copper clad sheet)**

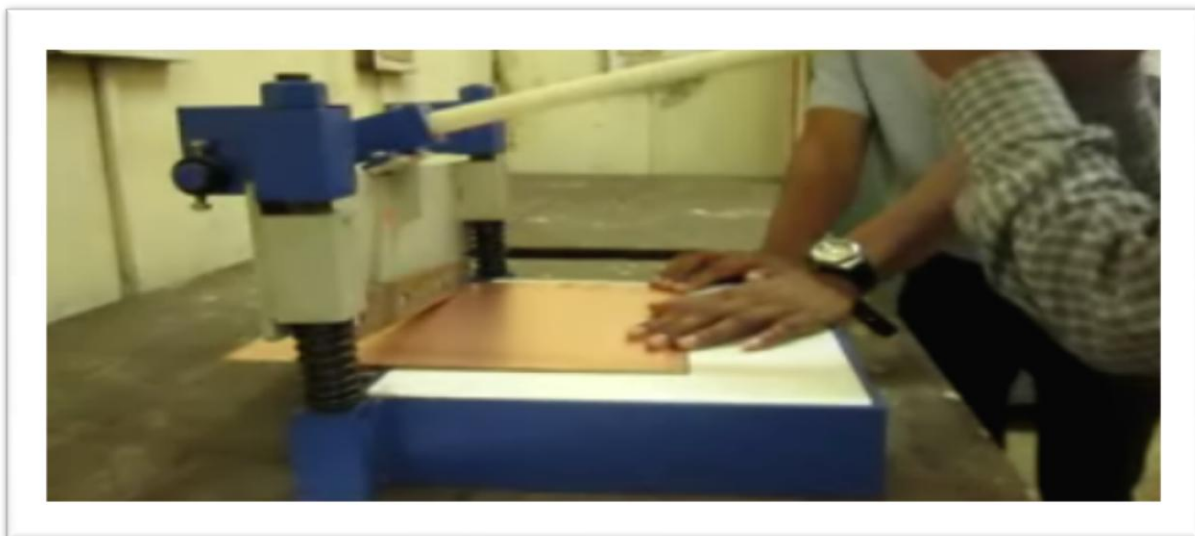


Figure – 15 Cutting Copper Clad Sheet with the

## IV. Negative development of the circuit (Dark Room Process)

### a) Introduction:

The process of exposing the photo-lith film to yellow light in the presence of the trace paper and its development to create negative film of the PCB design is done in dark room. Use only Red Light throughout this process.

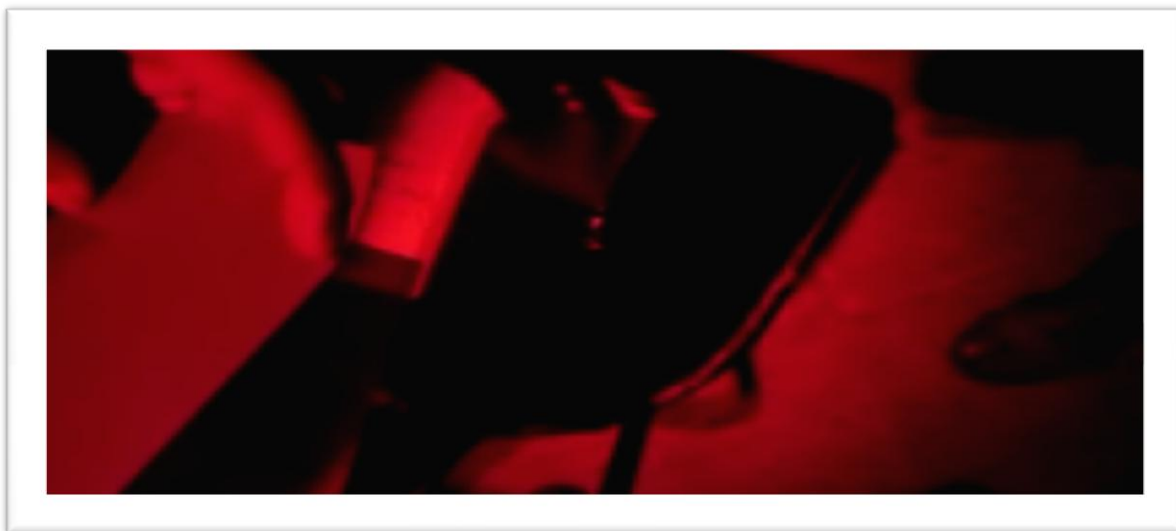


Figure – 16 Dark Room process  
[ Taking out the Photo Lith Film from envelope in RED Light ]

### b) Chemicals Required:

- Chemical A – Sodium Carbonate.
- Chemical B - Metol (monomethyl-p-aminophenol hemisulfate), phenidone (1-phenyl-3-pyrazolidinone) or dimezone (4,4-dimethyl-1-phenylpphenylpyrazolidin-3-one) etc are Hydrocarbons.
- Negative Developer Solution: Prepare a solution of chemical A (1 spoon) + chemical B (1spoon) & 1.5 litres of water in a tray No. 1.
- Water: Fill 1.5 litres in tray No. 2.
- Fixer Solution: Prepare a solution of 2 – cap of fixer & 1 litre of water in tray No. 3.

### c) Procedure :

1. Take out the negative film from the black envelope. Cut the negative film into the desired size of the layout. It is to note that negative film has two different sides, one is white & other is black. Black side is used for exposure purpose.

**Caution** : Don't open the black envelope and photo lith film in daylight or normal light. Otherwise, the negative paper will be exposed and no further process can be carried out with it.

2. Use tape to fix the circuit printed trace paper on the photo lith film, black side of film facing the trace paper. Now keep this on glass of Proto Contact Film Maker, trace paper facing downward. Close the flap TIGHTLY. Turn the machine on and expose the film by pressing the push switch for **10-12 seconds**.

**Caution**: Do not turn on the machine before placing the negative paper (photo sensitive film) over on it.

## Making Printed Circuit Board

3. Take out the exposed film. Put it in tray No. 1 (Developer Solution) for **2-3 minutes** & stir solution gently. The film will change its color first to white and again to black, at this time the circuit will appear on it clearly. Now without any delay take it out immediately.

### d) Precautions:

- Do not touch the negative film where the circuit is exposed. Use the extra outer part to handle it.
- When circuit appears immediately take it out, otherwise, it will be over-developed (whole film will be blacken).
- This process should be carried out evenly for the film otherwise it may lead to unequal development of negative.

4. Now, wash in tray No. 2 (water) for **1- minutes** & stir it gently. DO NOT touch it.

5. Now put it in tray No. 3 (Fixer) for **1-2 minutes** & stir it gently.

6. Wash the film in tray No. 2 (water) & then dry it.



Figure – 17 Tray -1 , Tray – 2 and Tray -3 Solutions

7. Now you can turn on the lights.

## V. Copper clad preparation [Photo Resist Dip Coating]

---



Figure – 18 Copper Clad hanging on the clip ready for the Photo Resist Dip Coating

### a) Introduction

Photoresists are classified into two groups: positive resists and negative resists :

1. A **Positive resist** is a type of photoresist in which the portion of the photoresist that is exposed to light becomes soluble to the photoresist developer. The portion of the photoresist that is unexposed remains insoluble to the photoresist developer.
2. A **Negative resist** is a type of photoresist in which the portion of the photoresist that is exposed to light becomes insoluble to the photoresist developer. The unexposed portion of the photoresist is dissolved by the photoresist developer.

### b) Chemicals Required:

1. **Photo-Resist Solution:** Fill the Dip coating machine with Photo Resist Chemical – **Poly Methyl Methacrylate (PMMA)** solution. If it is too viscous, dilute it with Thinner.

### c) Alternatives for Chemicals Used

- Poly(methyl methacrylate) (PMMA).
- Poly(methyl glutarimide) (PMGI).
- Phenol formaldehyde resin (DNQ/Novolac).
- SU-8.

### d) Procedure:

1. Take out the Copper Clad & cut it in desired size with the help of Cutter machine (See Figure - 15).
2. Now, Clean the Copper side with metal scrubber and wash it.
3. Turn On the Photo Resist Dip Coating Machine. Put the Copper Clad Sheet on the hook of Photo Resist Dip Coating Machine. Move lever DOWN & keep it inside the solution for **1 minute**. Now, take it out & dry it in air.

## Making Printed Circuit Board

**NOTE:** If the size of the board is large to coat in one time, then repeat the procedure for uncoated part.

4. Put it in oven. Set the timer for 2 minutes & temperature nearly 70°C.

**NOTE:** Turn on the oven, 10 minutes before using it. It takes time to reach desired temperature.



## VI. Ultra Violet exposure & developing



Figure – 19 Film stuck over the photo resist coated copper clad sheet ready for the Ultra Violet Exposure

### a) Introduction :

In this process , the Copper Clad + Film pair is exposed to the Ultra Violet Rays in the UV Chamber , so that after developing the UV Exposed Copper Clad sheet, the exposed part (Black Tracks / lines of the Circuit in the PCB design shown in Figure - 2 ) gets rough compared to the other part which was not exposed to ultra violet. This rough surface as we know is the required part of our Printed circuit board. So in the further process, we apply the Blue ink/ dye so that the blue ink gets stuck only to the rough part.

### b) Chemicals Required:

1. **Developer:** Fill Developer Solution (Developer (For Copper Clad Developing)) - *The UV Exposed Copper Clad sheet in PCB Making process is developed with a **1% solution of sodium hydroxide [NaOH]**. You can make this solvent by adding 10 gram of sodium hydroxide pellets to 1 litre of water and mix it until everything is dissolved* in a jar or tray.

2. **Water:** Fill 1.5 litres in a tray.

3. **Blue Dye.**

### c) Procedure:

1. Fix the negative on the copper side of Copper Clad using tape. (Place the negative correctly facing the clad otherwise your circuit will be Mirror of what you want to get.)

2. Put it in UV-Exposure machine between the glasses with Copper layer upside. Set the timer for **6 minutes**. Use TOP for single layer PCB and BOTH SIDE for double layer one.

**Caution:** Watch for any air gap between negative & Copper Clad. It should be tightly pressed between the glasses.

3. Remove the negative from the copper clad & put it in Copper Clad LPR developer for **2-3 minutes**. Stir it gently.

## Making Printed Circuit Board

**Caution:** After every minute, take it out & check if circuit appears on the clad. Inside the developer, the circuit cannot be seen. So it is required to take it out and observe.

4. Wash the Copper Clad gently with water without touching it.

5. Now, Apply Blue Dye over the Copper Clad softly with cotton.

**Caution:** DON'T RUB IT, otherwise the ink will not stay on the required part.

6. Keep for 5 to 10 second and wash it in water in tray without rubbing it. Now dry it in air. If you find any fault in the circuit of the dyed clad sheet correct it using a permanent marker, Please clean the excess of ink/dye if present.

7. Put it in oven for 2 minutes at 70°C.

## VII. Drilling process

---

### a) Introduction :

Drill the PADS in the blue coloured trace on Copper Clad after the Blue Ink was applied on the developed copper clad; using required drill bit size(Usually 0.8mm). Use the machine in 5000 RPM and drill it slowly.

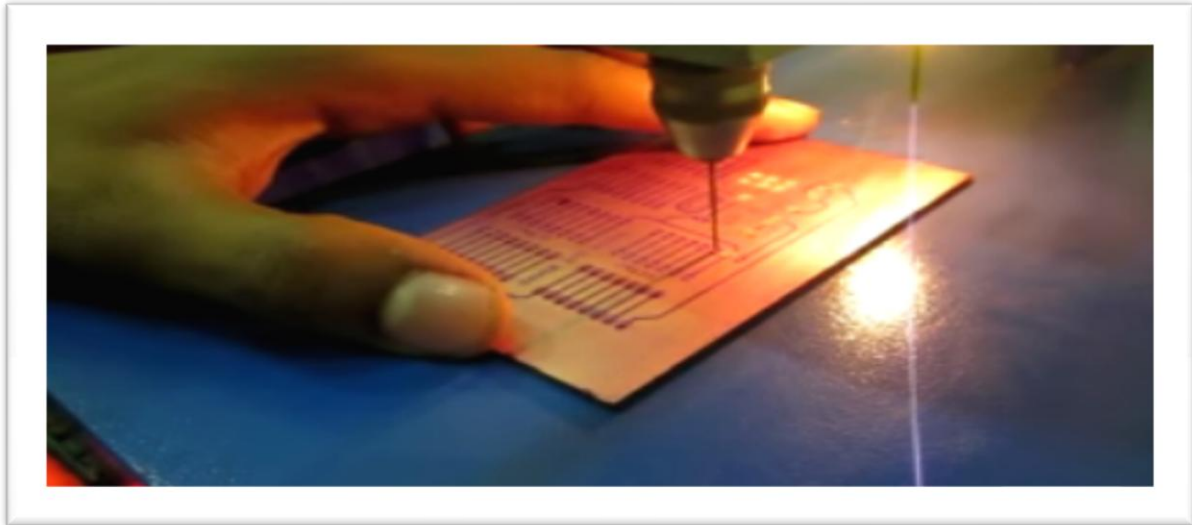


Figure – 20 Drilling Process

After this process, we then proceed to the Etching process as a final step.

## VIII. Etching process

---

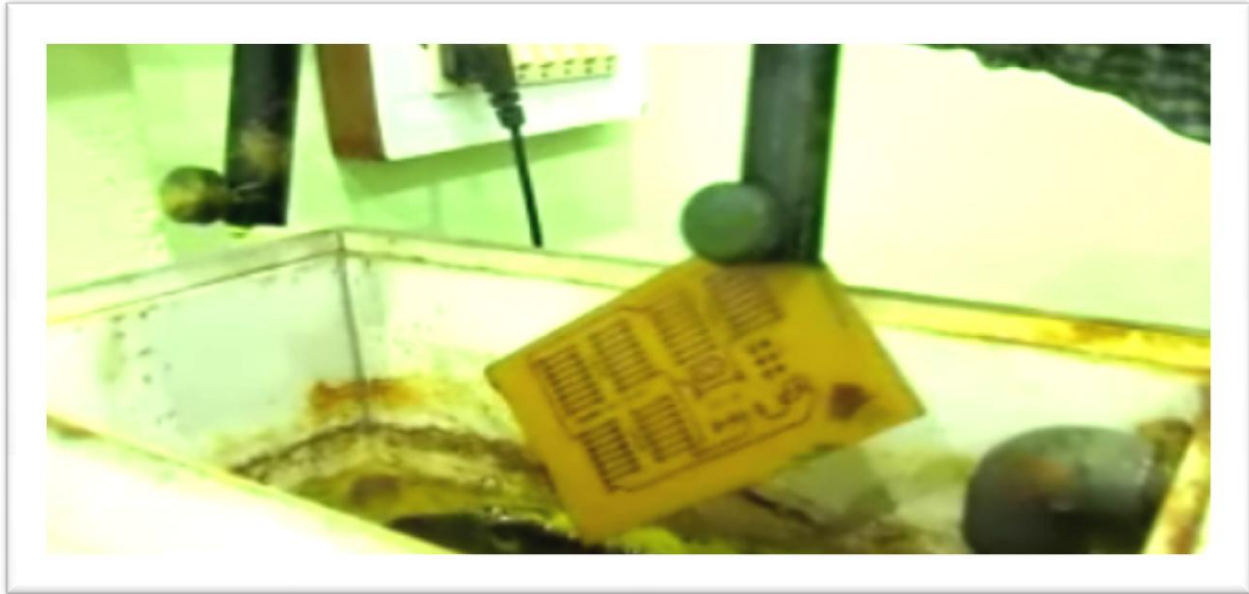


Figure – 21 Processed Printed Circuit Board after Etching  
i.e. the copper remains only on the printed portion ; as is visible in the figure

### a) Introduction :

In this process, we hang the Copper clad sheet having the design of PCB printed on it with a Blue ink as in the previous step. So now after dipping the sheet in Ferric Chloride solution , the part of PCB which is covered with the Blue ink remains as it is but the part which has no ink on it, dissolved in the solution of Ferric Chloride after undergoing chemical reaction. Hence we are now left with the Printed Circuit Board as required; with the design printed on it in the form of Copper Tracks.

### b) Chemicals Required:

1. **Ferric Chloride** in the etching machine.
2. **Water:** Fill 1.5 litres in a tray 2.

### c) Procedure:

1. Start the etching machine heater, 15 minutes before using it so that the Ferric Chloride solution reaches and maintains itself at the operational temperature during etching and also the process takes less time.
2. Put the clad in the etching machine and turn the timer on. This process should be made to happen for about 6-9 minutes or the user can check whether the etching has been done completely or not by removing the flap cover of the  $\text{FeCl}_3$  container.

NOTE : The blower starts only when the timer is set through the circular knob.

**Caution:** Check the clad between 4-6 minutes for the progress. Don't keep it inside too long; otherwise the required copper can also get etched.

3. When the etching is done wash it with water, till the dye color (i.e. Blue Color) is removed. Hence now we have our Printed Circuit Board ready to be used.

## IX. References

---

1. Making Printed Circuit Board (Video): <http://www.youtube.com/watch?v=khRay6Ooew4> .