**PCB FABRICATION:**

In initial stage of PCB fabrication is to study the circuit diagram. Study of diagram means to check out the components required and examine them and also note their size, dimensions etc. To note the dimensions is necessary to us in the next stage where after preparing the schematic we have give the footprints of respective components before preparation of the layout.

Second stage is an important stage in PCB fabrication. Here we prepare the soft copy of the layout manually or with the help of any software. In case of our project we have used software named “PROTEL99SE” for the preparation of PCB layout.

We first prepare the schematic and give each component label and footprint. A footprint defines the size and shape of the particular component.After preparation of the schematic we create new PCB file, then define the size of PCB. Then we load net list and place the components at proper places and route to get the final PCB layout.

Now we have the layout ready with us. This layout is first checked for errors. If there are no errors then we go for preparation of negative of the layout. A photo is taken by Vertical Photographic Camera and a negative is taken which gets developed on the special paper of camera roll. This negative is then washed with clean water so as to remove the glueyness of the paper. The film is then developed in A-developer for 1minute where layout is gets appeared. Now to make the tracks transparent the film is passed through B-developer. Then

Wash the film with clean water and is then kept for baking in hot air oven and at last the film is cut in appropriate dimensions.

The copper clad on which the film is to be kept and the layout is to be printed is cleaned with steel wool. This cleaning is necessary to remove the

carbon and dust layer that gets formed on the clad.

After cleaning the clad a photo-resist layer is applied. This photo-resist is a liquid and needed to be dried. For this purpose the clad coated with photo-resist layer is kept for baking in hot air oven.

When the clad is baked in the hot air oven it is ready for printing of the layout on the clad. Hence for the same purpose the film prepared is now placed on the clad properly and this composition is kept in U.V. ray machine. The U.V. light passes only through the transparent part of the film and only tracks on the film are transparent as a result the U.V. light and photo-resist solution forms a chemical bond.

The PCB is developed in LPR developer and diablo ink is sprayed on whole PCB. The ink gets applied only on the chemical bond formed in the U.V. ray machine.

Now inspection is done and whenever the ink is faint by using marker we make it dark so that the copper doesn’t come out. After inspection drilling with appropriate drill bit is done.

At last after drilling etching is performed. Here for etching FeCl3 solution is used. Because of this solution CuCl3 bond is formed copper gets removed whenever the ink is not applied. The PCB is washed with water and ink is removed with ready PCB.