

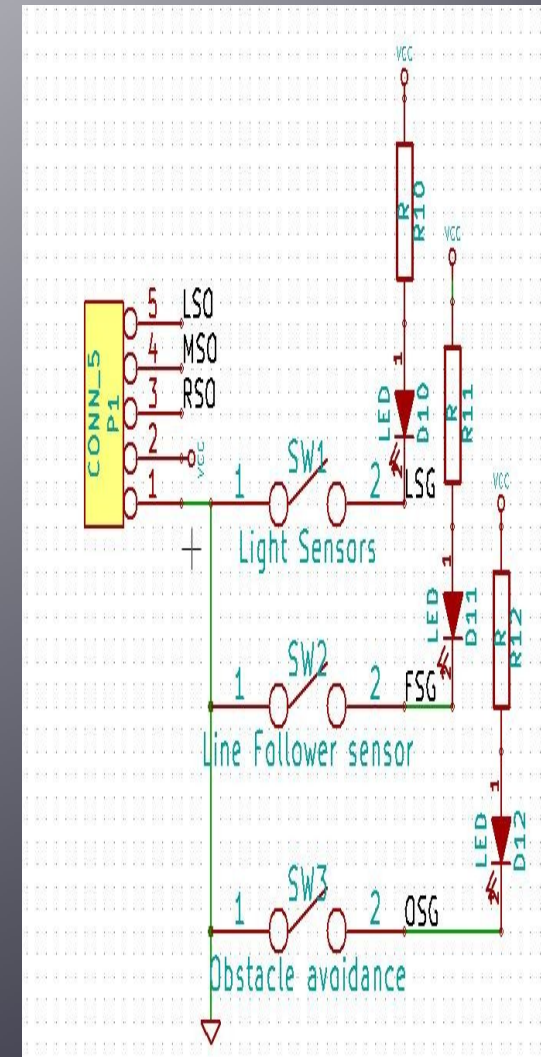
A presentation on:

PCB DESIGN & MANUFACTURING PROCESS

Xplore Labz

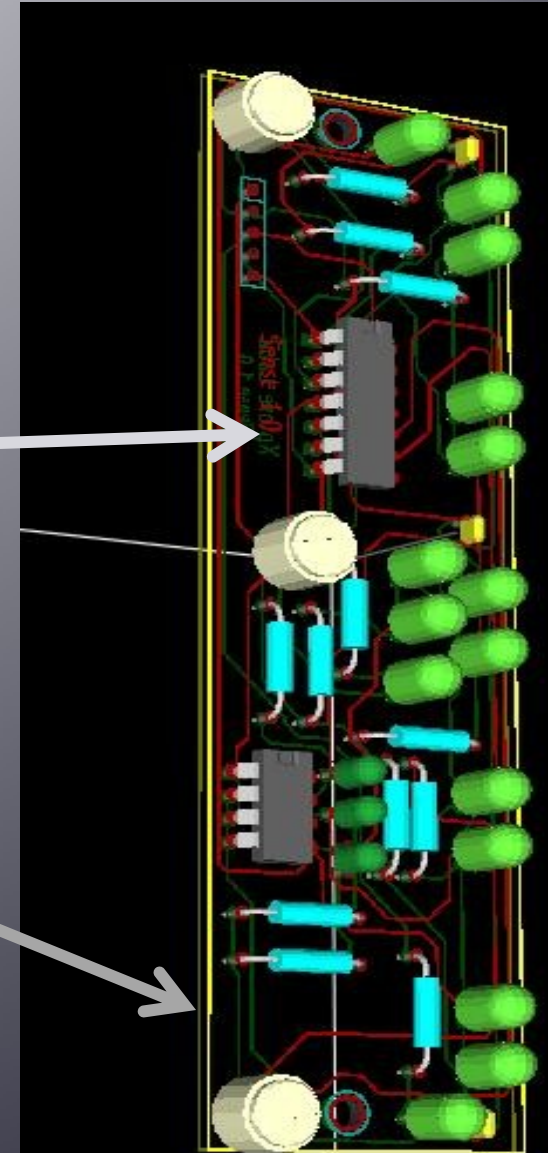
Step 1: Circuit Design

- The circuit design is done with popular Electronic Design Automation (EDA) tools.
- Examples are Eagle Cad, Kicad, Pspice, Express PCB etc
- It is very important to simulate and test the circuit before you start designing printed circuit board(PCB)
- In most cases the circuit is first designed and simulated using EDA tools and then tested on a bread board.
- **Note:** Some of the EDA tools are *open source* and would be a good starting point for amateurs and students. Example: Kicad



Step 2: PCB Design

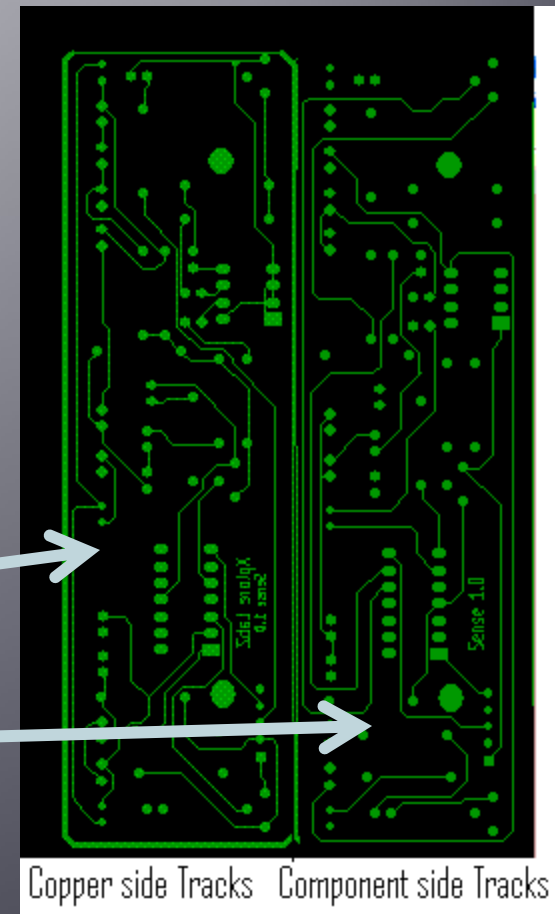
- The circuit designed in step 1, is used to generate a net-list
- This net-list is imported to a PCB design software.
- The circuit symbols in step 1. are replaced by footprints of actual components.
- The components are spread out according to the physical form factor required. i.e., orientation of ports etc.
- Now depending on complexity of circuit a single or multilayered design is chosen.



Step 2 contd. .

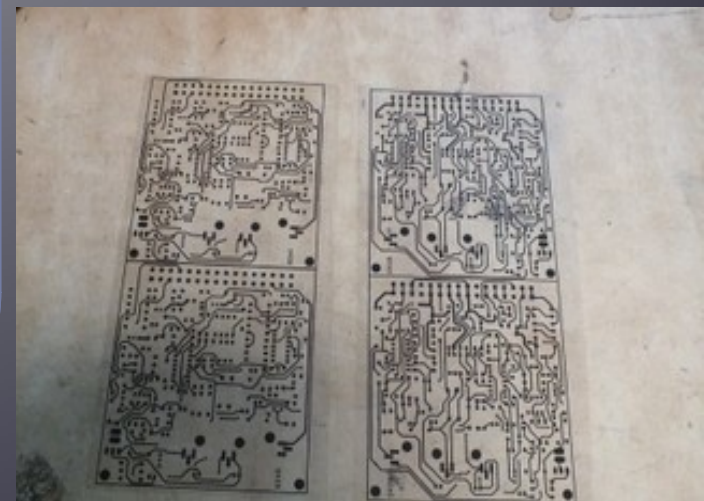
• While designing PCB layout a few important files are generated which are used for actual fabrications. The name of the files differ with the PCB suite that you use. The important ones are:

1. **Drill File:** Specifies drill and pad sizes for all the components in the design.
2. **Layer files:** Shows physical tracks. For a double layered PCB there will be two files one for the copper side tracks and other for component side tracks.



Step 3 Photo plotting

- The actual fabrication starts with this step. The first step is to photo plot all the design files discussed in the last step.
- The top layer, bottom layer, solder mask and legend files are photo plotted on a transparent sheet
- Picture on the Top shows a photo plotted PCB
- Usually multiple designs are duplicated and plotted at once for easy of mass fabrication.



Step 4 : PCB Drilling

- The drill file is now used for drilling the entire PCB board
- The drilling can be done using a CAD/CAM tool which directly takes the drill file as input and drills accurately.
- Drilling can also be done manually in cases where quantity is small or for test or prototype PCBs



Step 4: Electrolysis Process

- For a double sided Printed circuit board, when using plated through hole components (PTH), we need connectivity through(via) the holes.
- Hence after drilling the Copper clad, it is immersed in different chemicals usually known as catalysts. These will allow plating of tin or any other conductor inside the holes.
- Usually 2 to 3 microns of copper is required to be built for proper electrolytic plating.

Step 4: Electrolysis Process contd.

- The next steps is to provide a coat of conductive material usually palladium inside the holes.
- This is achieved by immersing the board in a catalyst(Palladium) chemical tank and steered.
- Usually 2 to 3 microns of copper is required to be built for proper electrolytic plating.

Step 5: Electro Plating

- After electrolysis plating we perform electro plating
- In Electroplating we build 8 to 10 microns of copper on both surface and inside the holes.



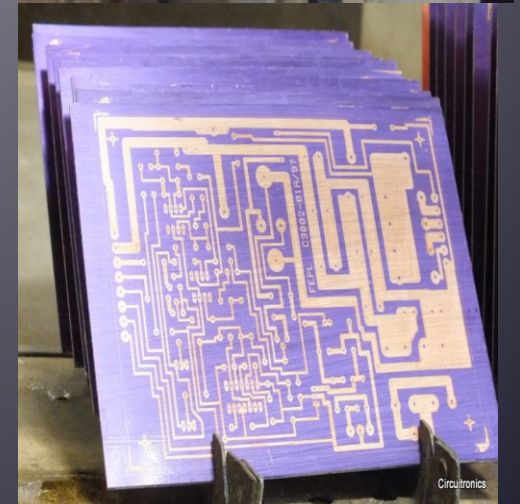
Step 6: Photo Resistive lamination

- Up to this steps we have the copper clad with conducting holes and now is the time to transfer the actual layout on the copper clad.
- For two layered PCB, the photo plotted design files in STEP 3 are laminated over the copper clad on either sides.
- The photo resistive adhesive film is 1.5 mil thick and is laminated on the copper clad with heat.



Step 7: Pattern Exposure

- The process of exposing the laminated board to UV radiation for sustained period is known as pattern exposure
- Now the supplied pattern is aligned with the drilled panel and it is exposed to UV radiation.
- Once the pattern is transferred on the copper clad, it is examined carefully to check if the exact layout has been transferred without any tracks or marks missing.
- The board is dipped in sodium carbonate solution for developing the exposed pattern.
- After this process, the we will see the impression of the required layout. This is known as image transfer.



Step 9: Copper and Tin Plating

- After the pattern is transferred to the copper clad, we need to build 8-10 microns of copper and is known as pattern plating.
- Minimum of 25 microns of copper has to be built inside the plated through holes.
- The panel is then taken for tin plating. Tin provides resistance for ammoniac etching.



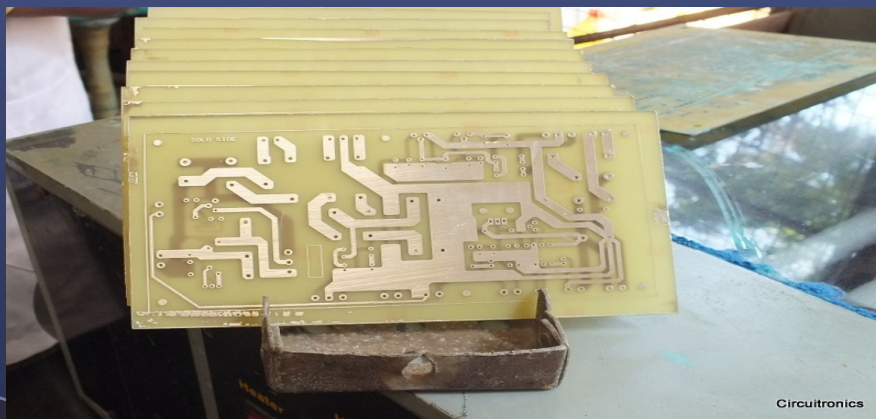
Step 10: Stripping the dry Film

- After the plating the film is removed, and we find the tin and copper plated on the tracks
- Images show the stripping process.



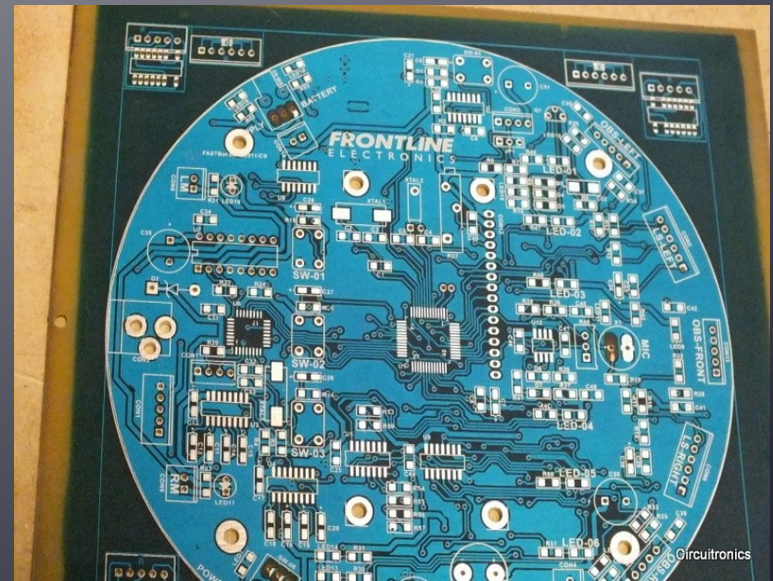
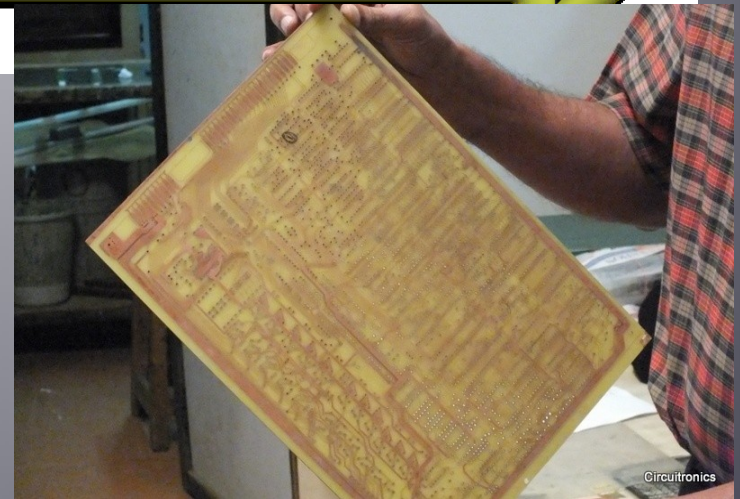
Step 11: Ammoniac Etching

- The final steps is to remove unwanted copper from the PCB.
- After etching the required layout remains the PCB is ready.
- This might be followed by solder masking and legend printing.



Step 12: Solder Masking and Legend Printing

- Solder masking is done for environmental protection.
- This prevents the copper from oxidation.
- It is followed by printing of labels and values on the board is known as Legend Printing.





Thank you!



*Team at Luminique Innovations
Pvt Ltd our Fabrication Partners*