Ketaki Mahajan

BATCH : PI-2

ROLL NUMBER: 16014022050

PCB Workshop (2022-2023)

Design and manufacturing of printed circuit board (PCB)

1. Explain PCB in details.

A printed circuit board (PCB) is an electronic assembly that uses copper conductors to create electrical connections between components. Printed circuits boards provide mechanical support for electronic components so that a device can be mounted in an enclosure. A printed circuit board design must include a specific set of steps that aligns with the manufacturing process, integrated circuit packaging and the structure of the bare circuit board.

Conductive features on printed circuit boards include copper traces, pads and conductive planes. The mechanical structure is made using an insulating material laminated between layers of conductors. The overall structure is plated and covered with a non-conductive solder mask, and a silk screen material is printed on top of the solder to provide a legend for electronic components.

The circuit itself is made by combining different sheets of non-conductive material, such as fiber glass or plastic, that can easily hold copper circuitry.

When the board only has copper traces and features no circuit elements, it is referred to as printed wiring board (PWB) or etched wiring board.

2. List softwares used for PCB layout design and explain how design layout using EAGLE PCB Design software.

Few of the softwares used for PCB are -

- Altium Designer (most famous PCB design tool in market).
- PCB Artist by Advanced Circuits (popular for its active customer base and best shipping record).
- SolidWorks PCB
- Ultiboard by National Instruments
- Dip Trace
- XCircuit
- KiCad EDA

To design layout using EAGLE PCB design software -

(i). A quick note on How Eagle Works.
(ii). The control panel.
(iii). Schematic window.
(iv). Board window.
(v). A couple of definitions before we begin.
(vi). Create a new project and schematic.
(vii). Add parts to schematic.
(viii). Connect the parts.
(ix). Label and name all the nets.
(x). Give some values to the parts.
(xi). Electrical rule check.
(xii). Board layout.
(xiii). Board layout 2 - getting on the right side.
(xiv). The ground plane.

(xv). Route the parts.
(xvi). Revisit thermals and orphans.
(xvii). The design rule check.
(xviii). DRC results.

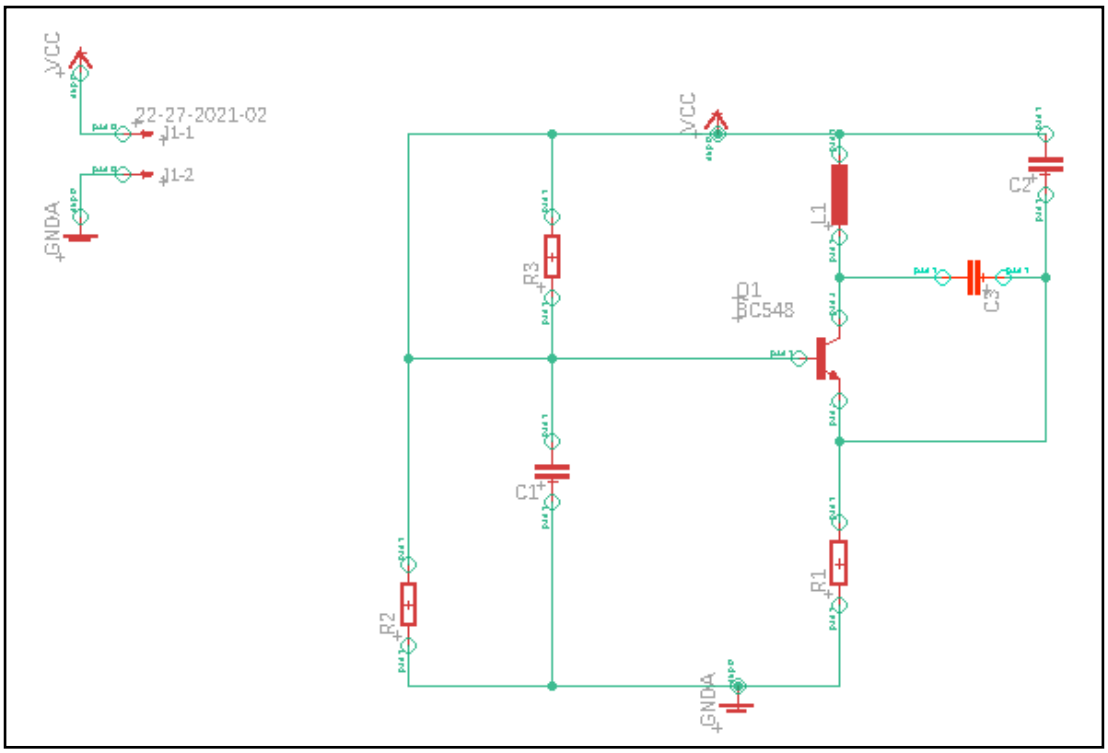
3. Write and explain in PCB fabrication process in short.
PCB fabrication is the process/ procedure that transforms a circuit board design into a physical machine based upon the specifications provided in the design package. This is achieved through following steps -
(i). Imaging desired layout on copper clad laminates.
(ii). Etching or removing excess copper from inner layers to reveal traces and pads.
(iii). Creating the PCB layer stackup by laminating (heating & pressing) board materials at high temperatures.
(iv). Drilling holes for mounting holes, through hole pins and vias.
(v). Etching or removing excess copper from the surface layer(s) to reveal traces and pads.
(vi). Plating pin holes and via holes.
(vii). Adding protective coating to surface or solder masking.
(ix). Silkscreen printing reference and polarity indicators, logos or other ~~marketing~~ markings on the surface.
(x). Optionally, a finish may be added to copper areas of surface.

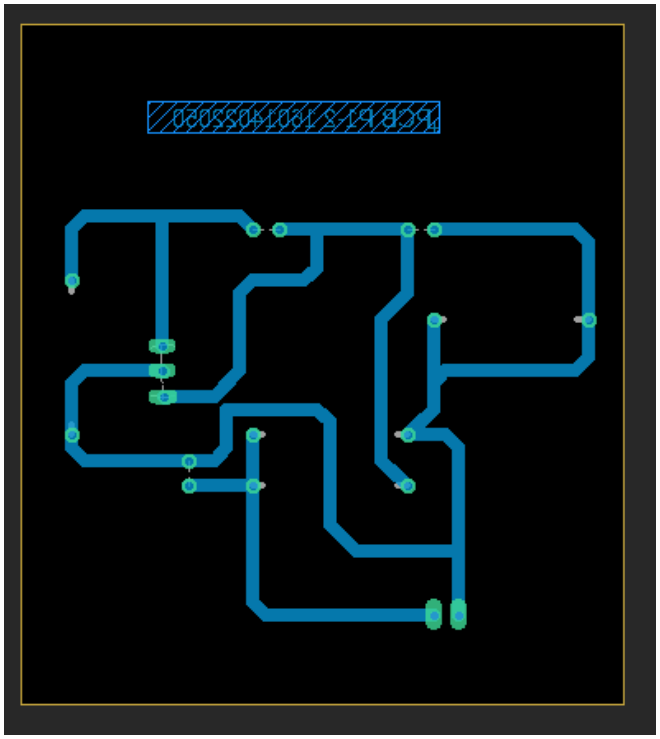
4. Draw a schematic diagram and its PCB layout of any electronics circuit.

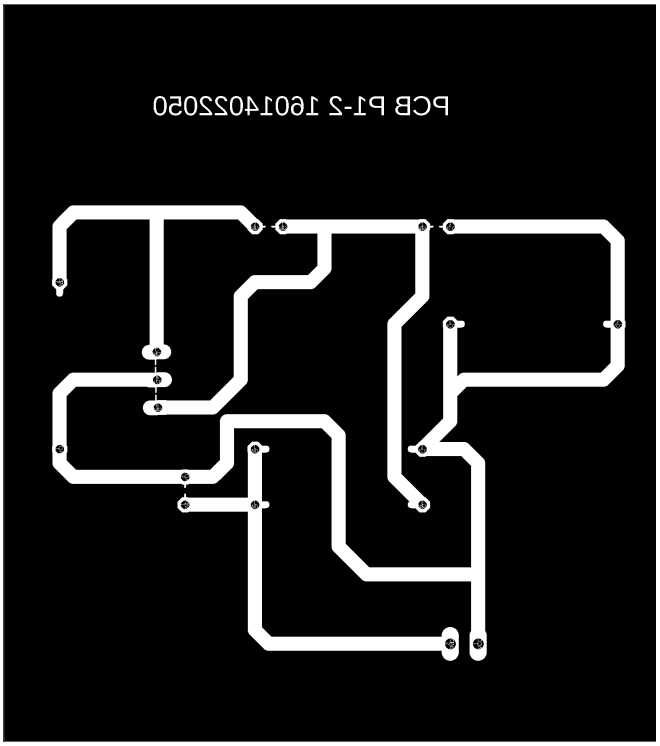
K. J. Somaiya College of Engineering, Mumbai-77

(Autonomous College Affiliated to University of Mumbai)

Department of Electronics Engineering







5. Describe the function of double sided UV exposure unit.

A double sided UV Vacuum exposure unit is an essential tool for the production of parts such as double-sided PCBs. After fixing layout-films exactly on both sides of the laminated board the light sensitive cover is exposed by an ultraviolet light in double-sided drawer exposure ~~light~~ unit with a vacuum system.

After exposure that part of the laminate, which is not protected by layout film becomes etch-resistant (if we use negative film & negative photo resist).