



**K. J. Somaiya College of Engineering, Mumbai-77**  
(Autonomous College Affiliated to University of Mumbai)  
**Department of Electronics Engineering**

**Batch: G3      Roll No.: 16010421063**

**Experiment / assignment / tutorial No. \_\_\_\_\_**

**Grade: AA / AB / BB / BC / CC / CD / DD**

**Signature of the Staff In-charge with date**

**PCB Workshop (2018-2019)**  
**Design and Manufacturing of Printed circuit Board (PCB).**

**Post Lab Subjective/Objective type Questions.**

**1. Describe the function of double sided UV Exposure unit.**

**Answer:** Double-Sided UV Exposure Unit:

- a) A double-sided UV Exposure Unit is an essential tool for the production of parts especially double-sided printed circuit boards (PCB).
- b) The vacuum which can be included in the UV exposure unit which makes it sustainable for processing UV sensitive flexible substrate materials.
- c) Double-sided exposure takes about 160 seconds to be completed.



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

(Autonomous College Affiliated to University of Mumbai)

**Department of Electronics Engineering**

**2. List the software's used for PCB layout design and explain how to design layout using DIPTRACE or EAGLE software.**

**Answer:** Software(s) used for PCB Layout Design:

- a) Eagle,
- b) Proteus,
- c) Altium Designer,
- d) Tiny CAD,
- e) Diptrage,
- f) ORCAD PCB,
- g) TINA,
- h) Zenith PCB,
- i) Auto TRAXDEX
- j) Mentor Graphics Xpedition.

Design Layout using EAGLE software

- Open PCB layout in the software
- In general window we find all the components, left click on the component and drag it onto the screen and drop. Right click on the screen then place the next components.
- To change the Label of component, right click and change the Label
- There are two sides of PCB that is top and bottom. At top, we see the components name and details and, on the bottom, we see the padding
- We can also rotate the components to arrange them in a systematic manner.
- After rotating arrange all the components and start drawing layout and complete all the connections.

**Department of Electronics Engineering**



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

(Autonomous College Affiliated to University of Mumbai)

**Department of Electronics Engineering**

- Layout is now ready to print. We can also place copper pour which is very useful for complex circuits to reduce the complexity as well as simplicity of layout. Then increase the clearance.
- After the completion of the layout, we can print it.

**3. Write and explain in short the steps for fabrication of PCB**

**Answer:** Fabrication of PCB:

Design and Output:

- Circuit boards are rigorously compatible with a PCB layout created by the designer using PCB design software.
- PCB software(s) like Altium Designer, ORCAD, Eagle etc. can be used.
- The software performs oversight algorithms on the design to ensure that no errors are detected.
- Designers also examine the plan with regard to element to track the width, board, edge spacing, tracing, hole size and spacing.

From file to film:

- Manufactures use a special printer called plotter, which makes photo films of the PCB's, to print circuit boards.
- The final product results in plastic sheet with a photo-negative of the PCB in blank ink.
- Each layer of PCB and solder mask receives it's own clear and black film sheet.

Printing the inner layers:

- The step in PCB manufacturing prepared to make actual PCB.
- It wants printing the figure on the film onto a copper foil.
- This step applies only to boards with more than two layers simple two-layer band skip to drilling.

Removing Unwanted copper:

- Just as the alkaline solution removes the resist, a more powerful chemical preparation eats away the excess copper.



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

(Autonomous College Affiliated to University of Mumbai)

#### **Department of Electronics Engineering**

- The copper solution removes all the exposed copper meanwhile the desired copper remains fully protected beneath the hard layer of photo-resist.

Layer alignment optical inspection:

- With all layers ready, the layers require an alignment punch to ensure they are all line up.

Layer up and bond:

- In this stage, the circuit takes a shape.
- The outer layer material consists of sheets of fiber-glass; a thin copper foil also covers a top and bottom of original substrate.
- Now it's time to bond them together on a heavy Steel cable with metal clamps.

Drill:

- Finally, holes are bored into the stack board.
- After the drilling, the additional copper that lines the edges of production panel undergoes removal by profiling tool.

Plating on copper deposition:

- After drilling the panel moves to plating. The process forces the different layers together using chemical deposition.

Outer Layer Imaging:

- In step 3, we applied photo-resist to the panel. In this step we do the same thing except with the outer layers of PCB design.

Plating:

- As we did in step 8, we electroplate the panel with a thin layer of copper. The exposed sections of the panel from the outer layer photo-resist stage receive the copper electro-plating.

Final etching:

- The tin protects the desired copper during this stage.
- The conducting area and connections are not properly established.

Solder Mask Application:

- The boards receive a blast of UV light, which passes through a solder mask photo film.
- The covered portion remains unhardened and will undergo removal.

Surface finish:

#### **Department of Electronics Engineering**



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

(Autonomous College Affiliated to University of Mumbai)

#### **Department of Electronics Engineering**

- To add extra solder-ability to the PCB we chemically plate them with gold and silver.
- This process leads to surface finish generation.

Silk screen:

- The nearly completed board receives ink-Jet writing on its surface which is used to indicate vital information about the PCB.

Electrical Test:

- As a final precaution a technician performs electrical test on the PCB.
- The test confirms the functionality of the PCB and its confirmatory to original design.

Proofing and v-scoring:

- The last step is cutting. Different boards are cut from the original panel.
- The method employed either centers an using router-ora V-Groove.

#### **4. Explain PCB in details.**

**Ans- Answer:** Printed Circuit Board (PCB):

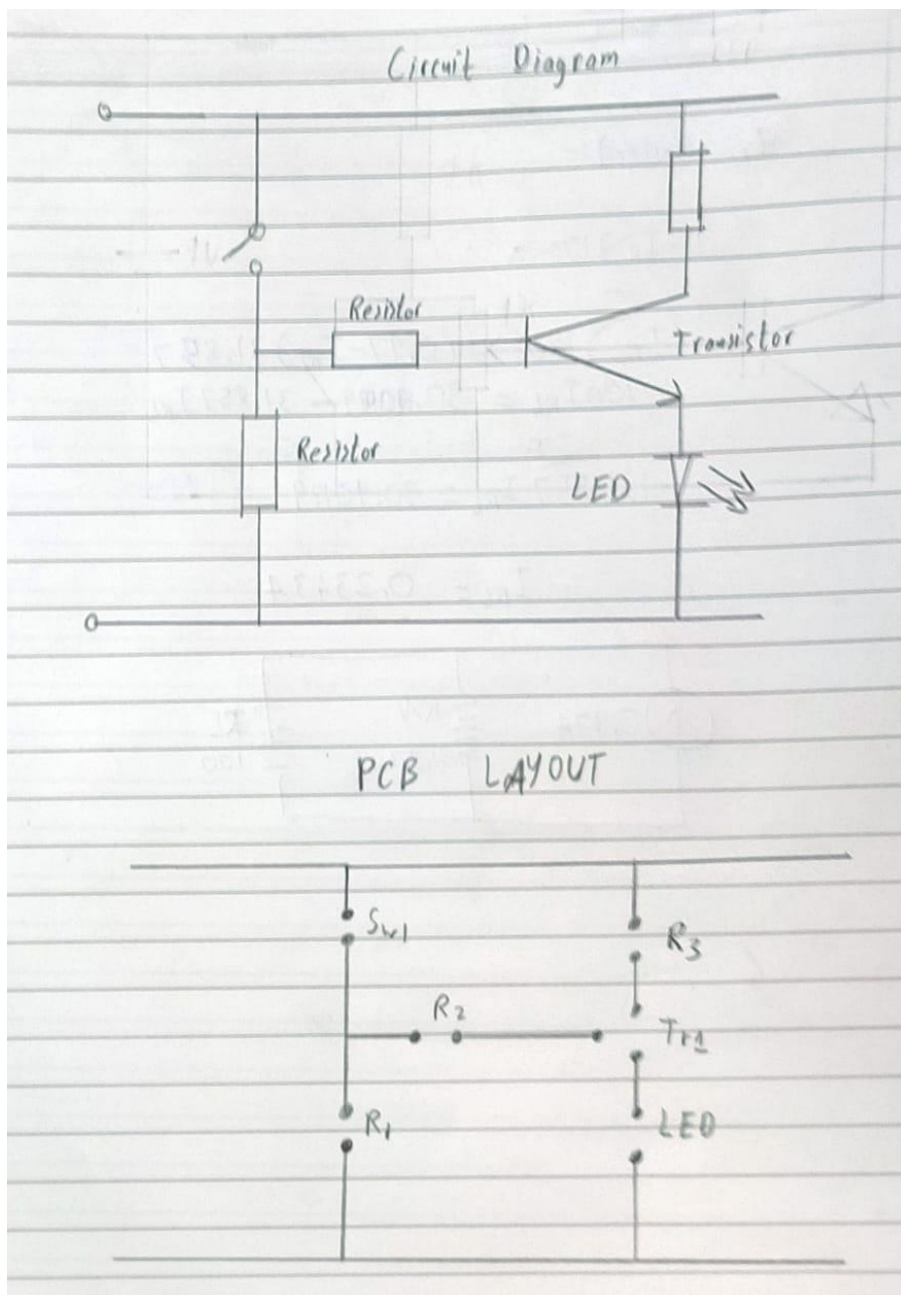
- A PCB mechanically supports and electrically connects electrical or electronic components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated with nonconductive material.
- It is built by altering layers of conductive copper with layers of non-conductive insulation materials and so on until printed circuit board is complete.
- Electronic components are added to the outer layer of the printed circuit board when all the layers have been etched and laminated together.
- Surface mount parts are automatically applied with robots and through hole parts are manually placed.
- During manufacturing the inner copper layers are etched leaving traces of copper to connect circuit compounds.
- Once etched copper layers are laminated to insulation material and so on until printed circuit board is complete.
- All the pieces are then selected and soldered onto the board using techniques such as reflow or wave soldering.

**Department of Electronics Engineering**

**K. J. Somaiya College of Engineering, Mumbai-77**  
(Autonomous College Affiliated to University of Mumbai)  
**Department of Electronics Engineering**

- The final assembly is placed after which solder mask and silk screen leg ending is applied.

**5. Draw any one electronic schematic diagram with its PCB Layout.**



**Department of Electronics Engineering**



**K. J. Somaiya College of Engineering, Mumbai-77**  
(Autonomous College Affiliated to University of Mumbai)  
**Department of Electronics Engineering**

**Department of Electronics Engineering**

PCB SEM I & II