

Understanding the parts of PCB Layout

¹Pravin Bhadane, ²Suchita Bhangale, ³Aparna Lal

Department of Electronics, Nowrosjee Wadia College, Pune-1

Abstract: Different parts of PCB layout have been studied for understanding their functions and use in PCB designing. PCB is an essential part of electronic circuit board on which electronic components are mounted. The main studied parts are pads, tracks, layers, mask, and silk screen. Some issues about the physical design of PCB such as chassis dimensions, location of input/output connectors, chassis mounting points, component parameters – size, heat dissipation have been studied. Present study is helpful to the beginners as well as to the under graduate students of Electronics.

Keywords: footprint, heat sink, layer, pad, silk screen, track.

I. INTRODUCTION

PCB stands for Printed Circuit Board that connects circuit components. Blank PCB is called a copper clad which contains insulating substrate with conducting surface. Common materials for substrate is either paper phenolic or glass epoxy and it is coated with a conducting layer typically made up of thin copper foil of thickness less than 0.1mm. The unwanted copper is removed from the substrate by a chemical process known as etching, leaving only the desired copper traces or pathways also called as tracks. Generally etching is done by dipping the PCB in a chemical bath containing 20% solution of ferric chloride. Sometimes few drops of concentrated hydrochloric acid is added in a bath for catalysis purpose.[1]

II. PARTS OF PCB

Main Parts of PCB layout are pads, tracks, vias, footprint, copper layer, solder mask, silk screen. These parts corresponds to the actual electronic components of the circuit. The commonly required electronic components are connectors, discrete and integrated circuit devices. Main parts of layout are shown in TABLE 1. Pad is a rectangular or circular shaped metal surface and it consist of inner and outer diameter. It is a location that connects terminals of discrete components or pins of IC. Pads are connected to each other by tracks. Tracks are essentially the wiring of the PCB, equivalent to wire of conducting signals. Power and ground tracks are usually wider than signal track. Sometimes in multilayered PCB tracks cross the layer through via. Where via is a pad with plated hole connecting tracks from one layer of the board to other layer. Physically, via is a hole on a PCB which consists of two pads in corresponding positions on top and bottom layers of the board, that are electrically connected by a 'hole' through the board. It is made conductive by printing through hole (PTH) some metallic material usually tin by electroplating process. Each electronic component has more than one terminal or pin for external connections and to connect them to other components of the circuit some arrangement on PCB is required, such an arrangement used to physically attach a component to PCB is called as footprint

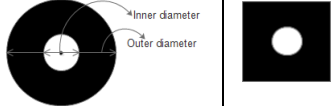

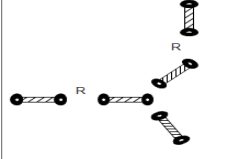
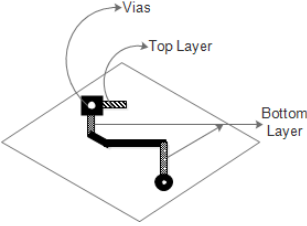

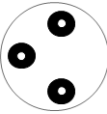
Layers are the top and bottom side surfaces of PCB. Components are usually mounted on the top side and soldered to the pads on the bottom layer of PCB. Such PCB is called as single sided PCB which has solder layer only on bottom side. Some PCB have solder layer on both sides which are called double sided PCB (DSP) and they are much more expensive than single sided one. Vias are required to join the tracks of two sides of DSP. Sometimes, signal wires overlap each other and make unwanted connection. Jumper is a piece of wire that is used to avoid the overlap. Jumpers are minimized by use of DSP. The tracks of the solder layer may be covered with polymer resist solder mask coating to protect them from corrosion. Printing readable information about component part number and functional blocks on the solder mask is called a silk screen. It is helpful for assembling, testing and servicing the circuit board. [2]

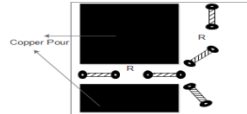
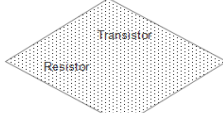
III. SOME ISSUE ON PHYSICAL DESIGN OF LAYOUT

The main design issues are chassis dimensions, location of input/output connectors, chassis mounting points, component parameters – size, heat dissipation, electric and magnetic field produced by them, etc. Chassis is a metal frame on which the circuit board and other supporting components are mounted. PCB dimensions should be appropriate so that it should fit in the chassis. The mounting holes on PCB should match with the chassis holes. The input connectors are generally mounted on front side or back side of the PCB, whereas output connector is usually on the front side. Some high power components dissipate more heat and to remove it safely there is a need of more surface area. The supporting element known as heat sink is a thick metal plate commonly used to provide more surface area to the components so that it dissipates heat off the component. The heat sink size depends on the amount of heat dissipated by components. Generally, heat sink is mounted on the back side of chassis.

Physical size of component must be considered for making PCB, which depends on wattage and voltage rating. Bulky and heavy components such as transformer and relay are generally not mounted on the PCB but they are placed on the chassis away from the PCB. Oscillators and clock are the common parts of any PCB. Sometimes they generate high frequency signal which induces some parasitic effect in the circuit board such as shunt capacitance, series inductance, inductive and capacitive coupling. The parasitic shunt capacitance bypasses the desired signal from the circuit. Similarly series inductance opposes the necessary signal. The capacitive and inductive coupling adds noise in the signal and reduces the signal to noise ratio. Hence PCB designer should take care of high frequency components and preferably he should place such components away from rest of the components. And if possible use technique such as shielding and grounding to minimize the effect of high frequency signals.[3]

TABLE I: PARTS OF PCB LAYOUT

Name	Shape	Use	Implementation
Pad		Provides space for soldering	Select pad of required size from the library and place it at destination
Track		Copper path for joining the pads	Select 'wire' tool and click to source pad and drag it to the destination
Layer		Flat plain for mounting and soldering	Select a layer of choice from menu
Via		Connects pads of two layers	Select 'wire' tool and draw some part of track on one layer and then switch to the another, continue to draw remaining part of track
Jumper		Connects pads in the absence of track	Use a normal multi strand or single strand wire
Footprint		physically attach a component to a PCB	Select footprint of component from library and place it at destination

Copper pour		Not used region of layer is filled with copper	Use inbuilt copper pour tool and draw the copper pour region on the layer
Silk screen		Text printed on the layer	Select 'text' option and write the abbreviated text

IV. CONCLUSION

PCB is a necessary part of electronic circuit design which works like a substrate for mounting the electronic components. It has advantages such as mass production and longer life. Pads, tracks, vias and layers are the parts required for making of PCB. Jumpers and mounting holes are also necessary parts of the PCB. The parasitic effects produced by high frequency components such as shunt capacitance and inductive coupling should be minimized by appropriate use of different parts of PCB.

V. FUTURE SCOPE

The present article is about the basic parts of layout which provides information to the beginners. This includes the common parts of layout which are used in almost all PCB layout softwares. Further we would like to study the use of some PCB software for making layout of electronic circuit. The procedure of making of PCB layout will be presented in the next part of this series of articles.

REFERENCES

- [1] Kraig Mitzner, "Complete PCB Design Using OrCAD Capture and Layout," Elsevier, Amsterdam, 2007, pp.2.
- [2] Walter C Bosshart, "Printed Circuit Board Design and Technology", 1st ed., McGraw Hill Education, 1984.
- [3] David Reusch and Johan Strydom, "Understanding the Effect of PCB Layout on Circuit Performance in a High-Frequency Gallium-Nitride-Based Point of Load Converter", IEEE Transactions on Power Electronics, Vol 29(4), 2014.