PRINTED CIRCUIT BOARDS

Prepared By:
Prof.Hiren D. Patel

Approved By:

1

INDEX

2

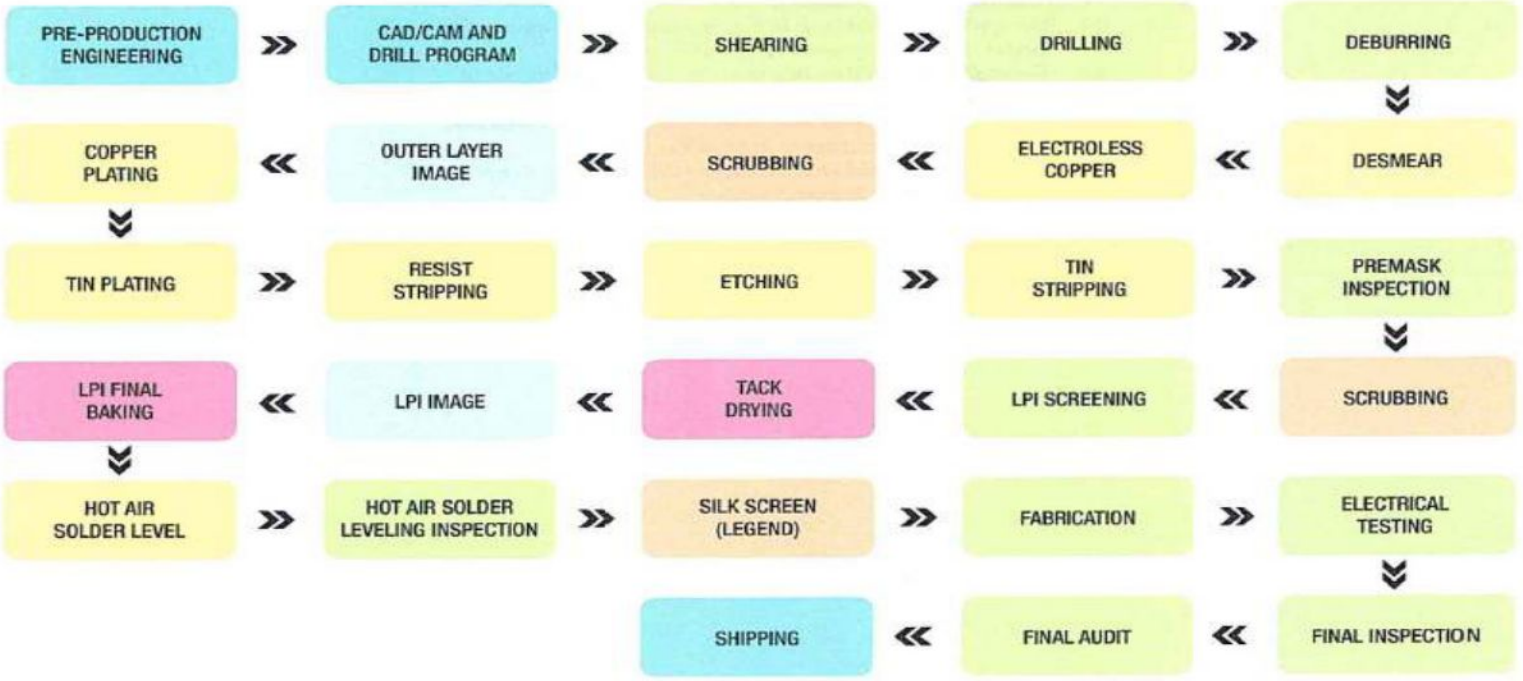
INTRODUCTION OF PRINTED CIRCUIT BOARDS

- Today, there is an increasing diversity of types of electronic modular assemblies. Many of them have been built with new kinds of basic materials and manufacturing processes and techniques.
- A firm understanding of these elements of construction will enable you to fully analyse the repair problem at hand and accomplish the task without further damage to the circuit board.

3

PRINTED CIRCUIT BOARDS MANUFACTURING PROCESS

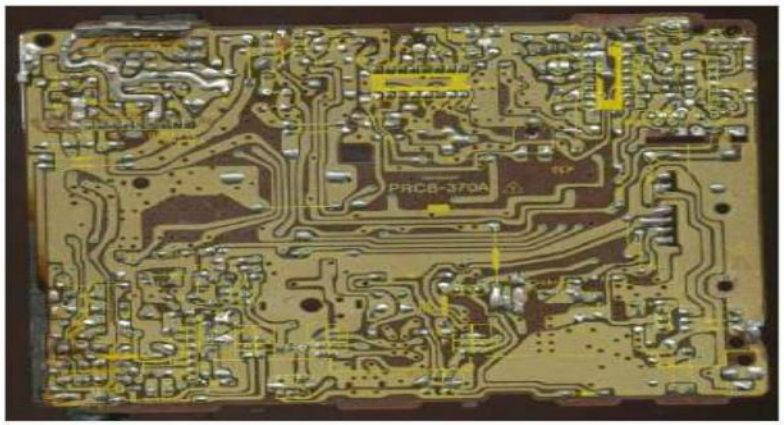
TYPICAL FLOW CHART FOR DOUBLE SIDED BOARDS



4

SINGLE-SIDED BOARDS

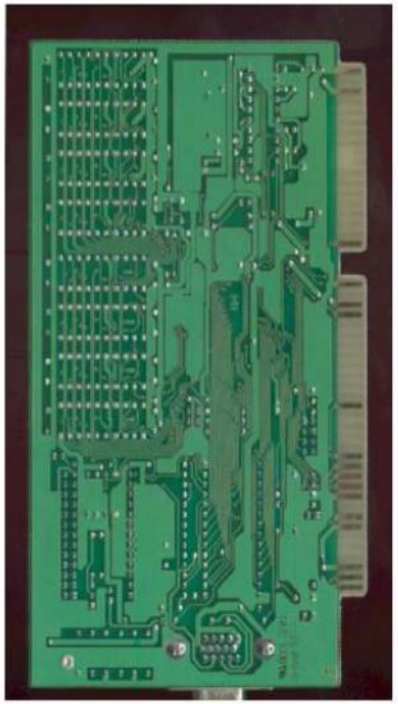
- The single-sided board has combinations of conductors, terminals (pads), and ground planes bonded to one side of the base material. The usual board also has a number of holes drilled through it to allow the mounting of components. This is shown in Figure.
- In a single-sided board, the pads are normally unsupported; there is no through-hole copper to support the pad to the board. The components are normally mounted on the back (non-circuit) side of the board, and the component leads are fed through the holes and soldered to the pads.



5

DOUBLE-SIDED BOARDS

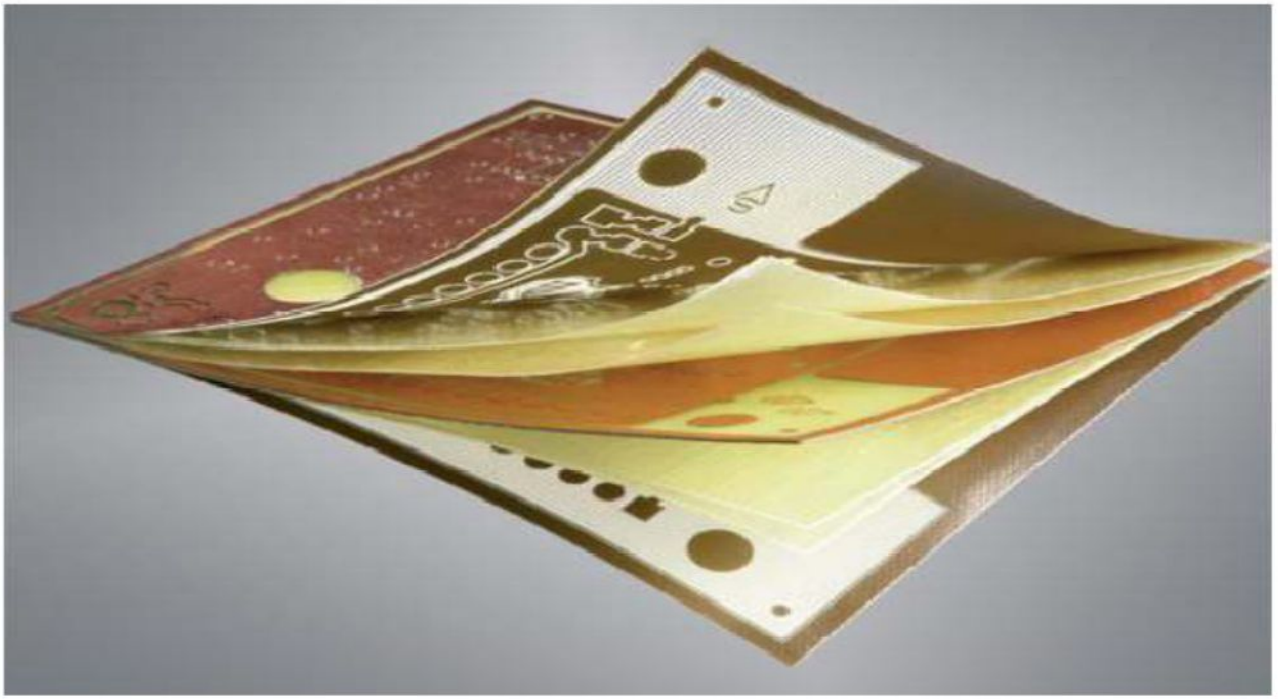
- The double-sided board has combinations of conductors, terminals and ground planes located on both sides of the board. In most applications, there is a continuous conductive path, or interconnection, between circuitry on both sides. This is usually via a plated-through hole, in which copper is plated to the hole walls and also to the pads on either side of the board. This is known as a supported hole configuration. In some boards, this through-hole support can also come from eyelets or funnelets that have been inserted into the hole and fused to the pads on either side.



6

MULTI-LAYER BOARDS

- Multi-layer boards, as shown in Figure 173, are similar in construction to the double-sided type of board, wherein a plated-through hole is utilised to provide a continuous electrical connection between the circuitry on both outside surfaces of the board. The multi-layer board, however, utilises one or more 'internal' conductor planes that are sandwiched in layers within the board. These are connected, in many locations, to the plated-through holes in the circuit board.
- It is very important to consider this internal construction for component removal, since the internal planes can act as heat sinks absorbing the heat applied to the surface solder and pad, and preventing it from completely melting the joint.
- Excessive heat at the hole/internal plane interface can cause damage and loss of continuity with the internal plane, often with no outward sign of damage. Since all plated through holes in a multi-layer may not interconnect with all (or any) of the internal planes, each plated through hole and its associated solder and lead must be considered independently with respects to thermal mass, heat transfer and dwell time on the joint.



COMPONENT TYPES AND MOUNTING TECHNIQUES

- Common components used in electronics are usually referred to as the discrete components or piece parts of a modular assembly. They can be separated into categories by function, such as resistors, capacitors, transistors, diodes, etc. In each category there are several sub-categories: for example, resistors may be carbon composition, vitreous enamel, wire wound or ceramic and may be fixed, variable or adjustable. The same may be true for other discrete components. In addition, a component may be classified as axial leaded, such as the resistor or multi-leaded, such as the transistor, dual in-line pack or LSI micro-processor style package.
- Each type of component has its own peculiarities and may be mounted in one of many different ways, depending on weight, shape, size, mechanical configuration, heat sensitivities and package size restrictions. Additionally, the component may be subject to electrical damage and require special precaution whiling handling.

The Straight-Through (Un-clinched) Lead

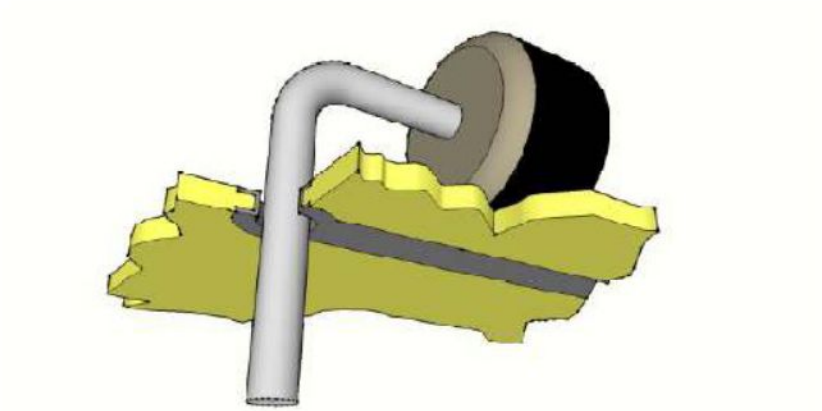
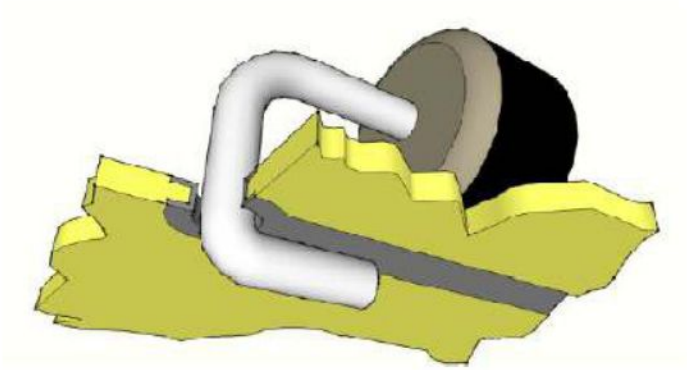


Figure 175: Straight through Component Termination Method

This type of termination, as shown in Figure 175, is the easiest to remove and replace because there are no bends in the leads after they pass through the holes.

10

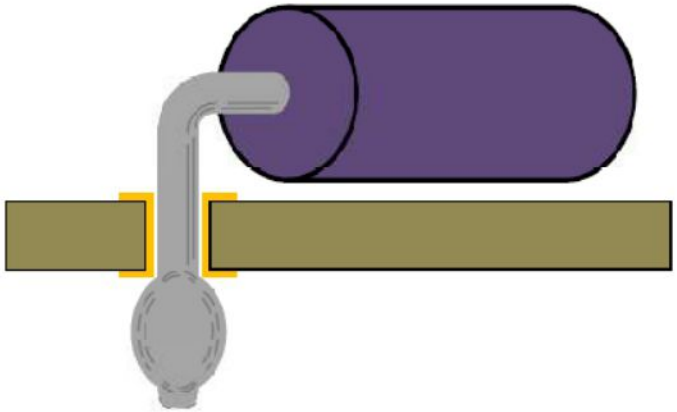
THE CLINCHED LEAD



This method is usually used with unsupported holes, but may be used with supported holes as shown in Figure. The component lead is inserted into the hole and then bent 90° onto the PCB conductor. This helps stabilise the component on the board so that it will not move during soldering.

11

THE SWAGED (SPADED) LEAD



The lead ends extending through the printed circuit board are swaged or flattened to increase their width beyond that of the hole, thus retaining the components in their proper position during the handling and soldering. This method required special procedures for removing the component. The swage is demonstrated in Figure

12

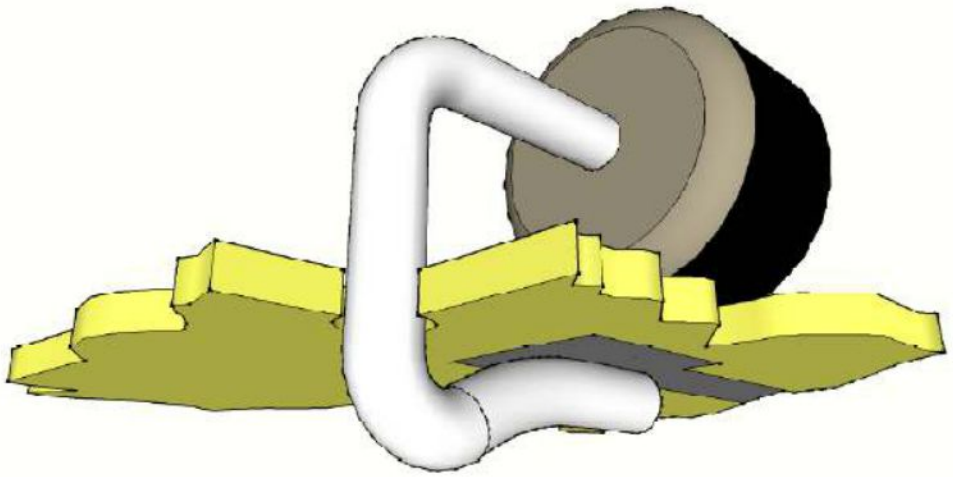
SURFACE MOUNTING

The leads of flat packs and other planar-mounted components are mounted and soldered onto the flat surfaces of the pads on the board, rather than passing through holes in the board. This is shown in Figure.

13

OFF-SET PAD MOUNTING

This is a variation of the Surface Mounting method. The lead is passed through a hole in the board and is bent so that it can be surface-mounted to a pad on the opposite side. This is shown in Figure. In some instances, the pad to which the lead is joined is some distance away from the hole.



14

CIRCUIT BOARD PACKAGING

- On some boards, the components are mounted widely spaced apart; this is known as low-density packaging. Boards with high-density packaging have many components mounted very close together.
- With low-density packaging, you have more space to work with and it is easier to get at the components in order to remove them. But with high-density packaging, circuit runs are much narrower and more closely spaced and components may be mounted so close together that some may be totally inaccessible.
- In this case, you may need to partially disassemble the board to get at them. This is where your knowledge of how the board was put together is useful; you have to 'de-manufacture' it to get at the parts, then 're-manufacture' it when you replace them.

COATINGS

- Conformal coating is a protective non conductive dielectric layer that is applied onto the PCB assembly.

These coatings are used to a variety of reasons:

- To provide electrical insulation.
- To cushion against mechanical shock and vibration.
- To prevent abrasion.
- To help support and bond the components.
- To function as a heat sink, or to protect against humidity or fungus.
- To protect from damage due to contamination, salt spray
- To protect against corrosion caused by harsh or extreme environments.

The general characteristics of coatings, for purposes of removal, are hardness, degree of transparency, thickness, solubility and thermal properties.

16

MAIN TYPES OF COATING MATERIALS

- **Varnishes** are hard, applied in thin coats, translucent or opaque and non-soluble in mild solvents.
- **Acrylic Lacquer** is hard, usually thin, transparent and soluble in lacquer thinner or xylene.
- **Epoxies** are hard, may be applied thickly or thinly, may be transparent or opaque and are non-soluble in mild solvents.
- **Silicone Varnish** is soft, may be applied thickly or thinly, is transparent and is soluble in most mild solvents.
- **Polyurethane** may be hard or soft, may be thickly or thinly coated, is always transparent and is non-soluble in mild solvents.
- **RTV** rubber (or silicone rubber) is opaque, soft and usually very thick. The colour is normally white, red-brown or black.

17

- Parylene is uniformly thin, non-soluble, transparent and very tough.
- Coatings are applied in a number of different ways:

 Dipping.

 Spraying.

 Brushing.

 Vacuum condensation.

- Spraying is used for thin applications, while the dipping method is used for thicker coating.
- Parylene is applied by vaporising the required quantity of material in a vacuum chamber ; the coating then condenses uniformly on the entire assembly. Masking is used over areas not requiring coating and is removed after the coating process.
- Once they are applied, coatings must be dried or cured. This is done in a number of ways, depending on the type of coating used. Air drying and heat curing are two of the most commonly used methods.

18