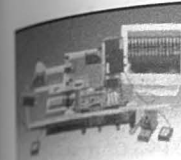


- **Add Test Points**
 - Set-up design for test audit rules and preferences.
- **Perform design rules checks**
 - Check for clearance, connectivity, hi-speed, planes, test points and fabrication.
- **Process Gerber, Drill and Assembly Data**
 - Numerically sort out the drill table.
 - Organize the drill sizes from smallest to largest by synchronizing the lowest drill symbol to the smallest drill size. Continue until all drill sizes have been defined.
- **Generate Gerber Data**
 - Select the Gerber files except the top and bottom assembly drawings.
 - Compare the ASCII netlist to the design netlist.
- **Generate Assembly and Drill Drawings**
 - Select pen, assembly top, bottom, and drill drawings and press.
 - Create the zipped file needed for fabrication and assembly.
 - Archive all data in the source safe and send the appropriate files to the fabrication and assembly houses.



5.15 Useful Standards

- *IPC-2531: Standard Recipe File Format (SRFF) Specification*: Outlines the requirements that an SRFF file must meet; describes the file format, outlines the file sections and indicates how data should be represented through objects which can either be vendor-independent or specific.
- *IPC-2547: Sectional Requirements for Shop Floor Equipment Communication Messages (CAMX) for Printed Circuit Board Test*: Describes event message content and an XML encoding scheme, which enables a detailed definition of messages in the domain of electronics inspection, test and repair/rework i.e. product and process quality.
- *IPC-2571: Generic Requirements for Electronic Manufacturing Supply Chain*: Defines an XML encoding scheme to facilitate supply chain interactions; the scheme is defined for bill of materials, approved manufacturer list, changes in engineering, manufacturing or product.
- *IPC-2511-A: Generic Requirements for Implementation of Product Manufacturing Description Data and Transfer Methodology*: Identifies the generic requirements for implementation of product manufacturing description data and transfer; helps users to transfer design requirements and manufacturing expectations from computer-aided design systems to computer-aided manufacturing systems for printed board fabrication, assembly and test.
- *IPC-2511B: Generic Requirements for Implementation of Product Manufacturing Description Data and Transfer XML Schema Methodology*: Specifies the XML scheme that represents the data file format to describe printed boards for tooling, manufacturing, assembly inspection and testing requirements.



Copper Clad Laminates

6.1 Anatomy of Laminates

The basic function of the laminate is to provide mechanical support for electronic components and to interconnect them electrically. Laminates for PCBs are composite materials. They can be simply described as products obtained by pressing layers of a filler material impregnated with resin under heat and pressure. The resulting thin insulating material, which is the mixture of filler (reinforcement) and resin on which all conductors and components are mounted, is called *base material*. This can be either rigid or flexible material.

6.1.1 Fillers (Reinforcements)

Fillers are meant to provide mechanical strength, stability and rigidity to the laminate. The commonly used fillers are a variety of papers, cotton fabric, asbestos sheet, glass in various forms such as cloth and continuous filament mat, ceramic material, molybdenum, etc. However, the most common materials used are paper and glass fibre.

Paper has been used as reinforcement in a vast majority of printed circuit boards. They are low priced and are easily machinable. However, they have a tendency to absorb a lot of moisture.

Fibre-glass as reinforcement has gained popularity because of its high tensile strength and dimensional stability. It offers a high resistance to temperature variation and has a low moisture absorption property.

- *IPC-TR-485: Results of Copper Foil Rupture Strength Test Round Robin Study*: The test report evaluates rupture strength testing as a means of determining the mechanical properties of electrodeposited and rolled copper foil.
- *IPC-4412: Specification for Finished Fabric Woven from "E" Glass for Printed Boards*: Covers the classification and requirements for finished fabrics woven from E glass fiber yarns.
- *IPC-4130: Specification and Characterization Methods for Non-woven E Glass Mat*: Defines the nomenclature, definitions and requirements for materials made from non-woven E glass fibres; includes specification sheets for selecting and purchasing these materials.
- *IPC-4110: Specification and Characterization Methods for Non-woven Cellulose Base Paper for Printed Boards*: Defines the nomenclature, chemical and physical requirements of paper made from cellulose fibres for PCB fabrication.
- *IPC-4411-K: Specification and Characterization Methods for Non-woven Para-aramid Reinforcement*: Includes the nomenclature, definitions and requirements for reinforcement made from non-woven para-aramid fibres.

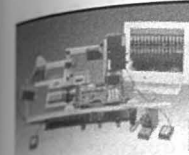


Image Transfer Techniques

7.1 What is Image Transfer?

Image transfer basically involves the transfer of the conductor pattern from the film master on to the copper clad base material or any other metal clad laminate. In the fabrication of the PCB, the two methods common for image transfer are:

- Photo printing method; and
- Screen printing method.
- **Photo Printing**: This is an extremely accurate process, which is generally applied to the fabrication of semiconductors and integrated circuits wherein the conductor widths are typically in the region of a few microns. Although such a precision technique is not required in the production of general purpose PCBs, yet where conductor widths of 100 μm are required and for PCBs for professional applications, the photo printing process is resorted to.
- **Screen Printing**: Although less precise than the photo printing process, screen printing is a comparatively cheap and simple method. The majority of PCBs produced worldwide are screen printed.

7.2 Laminate Surface Preparation

Copper surface plays a major role in the success or yield of the image transfer process. It demands that the surface should be carefully inspected for pits, drilling burns and any other types of irregularities. If unacceptable defects are observed, the image transfer process should not be carried

abrasive materials are impregnated into a nylon or similar plastic matrix. The resulting brush construction can be either a compressed lamella or a filament type. The silicon carbide 320-grid abrasive filament brush is generally used for cleaning the copper surface prior to lamination of the dry film resist. There are different types of mechanical cleaning operations. They are:

- Polishing;
- Brushing;
- Buffing or sanding;
- Deburring or scrubbing; and
- Scrubbing

Polishing

This process involves smoothening of the metal surface to improve its appearance. To do this, polishing wheels (muslin) are generally used.

Brushing

This process is a metal smoothening operation which involves rotary non-metallic wire brushes such as nylon brushes. The wire brushes are generally run wet. For very dull finishes, pumice slurry of grade 3F or 4F with water is used. Although the cleaning performance with respect to heavy oxides and other metallic soils as well as the removal of particulate matters is excellent, the removal of organic soils such as oils and greases is not complete. In fact, some of these soils are retained in the brushes. Therefore, the de-greasing step with a solvent is recommended before brushing. Figure 7.3 shows the scheme of a typical cleaning process with abrasive brushing for high volume PCB production with plated through-holes.

Buffing or Sanding

Buffing implies that the metal surface is buffed with the muslin wheel, to obtain low gloss finish to high gloss finish. The method removes heavy inorganic soils and particulate matter, and also helps to remove the burns around the drilled holes. Buffing can be carried out with powered hand vibrators.

Deburring:

After the board is drilled or punched, burr formation is noticed. Burr is the extra projection of metal when a hole is drilled through the PCB. Even with the best drill bit and drilling equipment, the formation of certain amount of burrs cannot be avoided, especially on the side where the drill bit comes out from the base material. For reliable plated through-holes, these burrs must be completely removed. This is usually done by abrasive cleaning. The machine used for deburring consists of a deburring wheel (fused silicone carbide), which is used to remove the burr on the drilled laminate. This process is carried out only in the

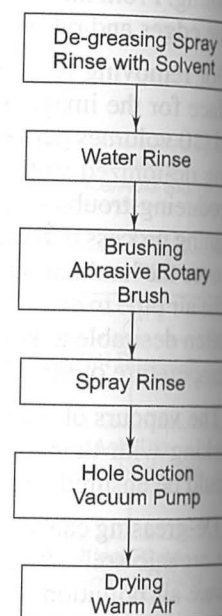


Fig. 7.3 Cleaning process with abrasive brushing for high volume PCB production with plated through-holes

presence of wet conditions. The sanding process (wet deburring) involves a water-resistant abrasive of 280 to 600 grid.

In all the above methods, after the laminate is cleaned, the boards are rinsed with water at high pressure (> 60psi) and dried with hot blow air. Normally, all mechanical processes involve automatic or semi-automatic machines. During these processes, the base material may be removed which, in turn, degrades the electrical properties of the boards. Hence, strict process monitoring is required.

Scrubbing

Scrubbing is carried out with fast rotating brushes made of fine plastic brushes which are continuously fed with abrasive slurry. The slurry is prepared by adding pumice powder to water and is generally re-circulated. The method includes a final water brush – rinse cycle to ensure the complete removal of particulate matter. Such machines provide an excellent surface finish on the PCB surface.

7.2.3 Test for Cleanliness

In order to test the adequacy of the cleaning process, an in-process water break test can be used. The cleanliness of the copper surface can be expressed in terms of the contact angle and wettability.

- **Contact Angle:** A clean copper surface should have a very low contact angle.
- **Wettability:** The clean copper surface should hold a continuous film of clean water of 15 to 30s. The water film should not break into droplets nor show areas of de-wetting during this test. If water breaks, it indicates that the cleaning process is not adequate.

7.3 Screen Printing

The photographic image is transferred to copper clad laminate by the screen printing technique. This technique has been used for a long time for printing cloth, panels and so on, and reaches its maximum degree of accuracy when applied to PCB manufacture. The technique is particularly adopted for low cost print and etch and plated printed boards, when the ultimate resolution and definition are not very exacting.

A screen comprises an aluminium frame, mesh, emulsion and adhesive bonding. The emulsion is removed by a photochemical process where deposition is required. A specification of 1:1 ratio of open area to board pad area is typical, but $\pm 10\%$ variations are not uncommon. The flexibility of the emulsion creates a good gasket against the PCB and aids print definition.

It is basically a stencil operation, which depends on the transfer of resist to the copper clad laminate surface using a stencil image of the circuit design. The stencil is firmly attached to the surface of a silk, nylon or stainless steel screen. A liquid resist material is forced through the open areas of the screen mesh that are not protected by the stencil onto the copper clad substrate by the pressure of a squeegee wiped across the top surface of the screen. The squeegee pressure deflects

the screen downward in point contact with the substrate. As the squeegee passes a given point, screen fabric tension snaps the screen back, leaving screen ink behind. Figure 7.4 illustrates the screen printing process.

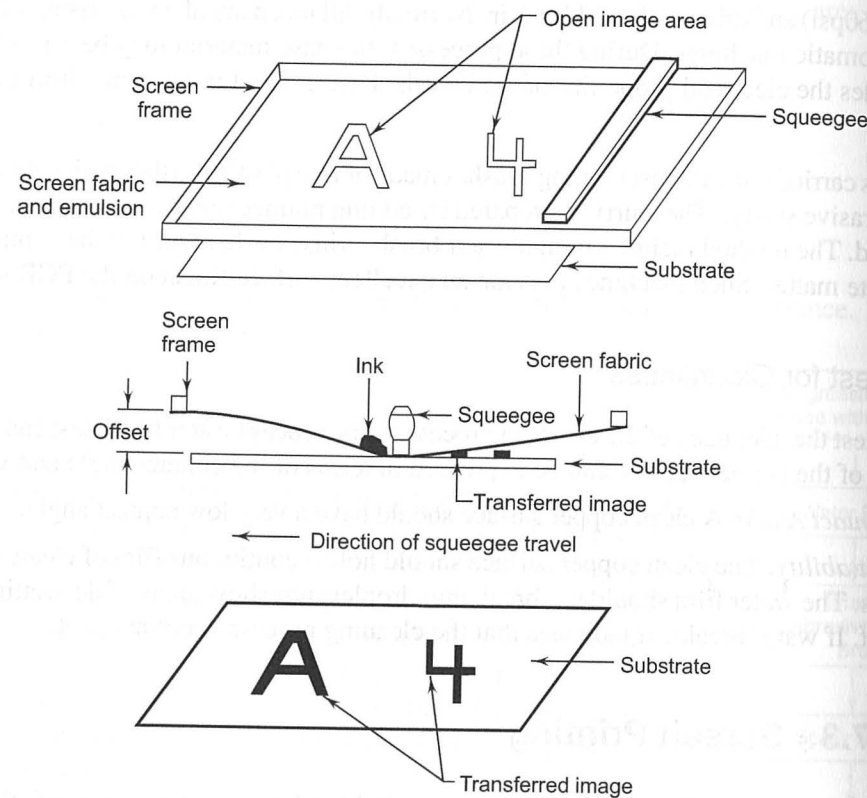


Fig. 7.4 Screen printing process (NTTF Notes)*

7.3.1 Screen Frame

The frame provides a support on which a screen fabric of uniform mesh or opening is stretched. Normally two types of flat screen frames are used: 'rigid type' and 'adjustable type' using floating bar frame. These are shown in Figure 7.5.

The frames are made of strong materials like wood, aluminium alloy, stainless steel or plastics. Aluminium alloys and stainless steel frames are the most widely used frames in the PCB industry. Even though the weight of a wooden frame is comparatively lesser, it is not used due to its absorption of water during the process, leading to a bend tendency. Normally, aluminium alloy is used for rigid type and stainless steel frame for stretchable type (variable tension) frames. The frame size is usually 400 m × 400 m. However, only 10-25 per cent of the screen area should be utilized.

* NTTF: NTTF Electronics Centre, Bangalore, India.

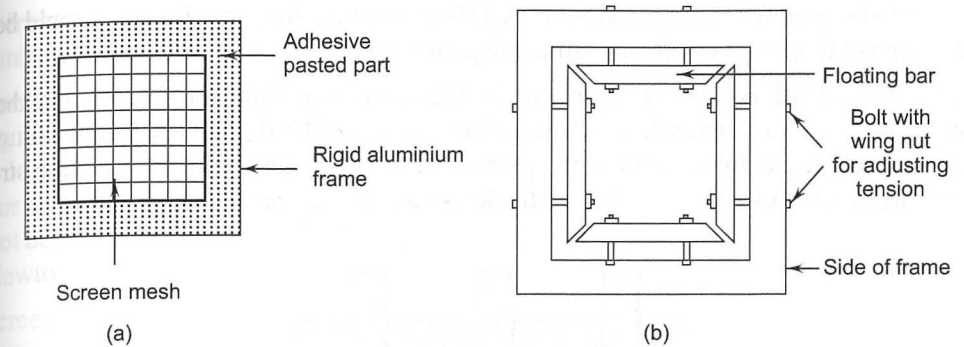


Fig. 7.5 Types frames for screen printing (a) rigid type frame (b) stretched type floating bar frame

7.3.2 Screen Cloth

The woven mesh fabric used in screen printing may consist of stainless steel, silk, polyester, nylon or similar material.

The threads of the above fabrics are classified into the following two types:

- **Mono-filament:** This is woven with single thread fibres. Its examples are stainless steel and phosphor bronze, nylon and polyester.
- **Multi-filament:** A multi-filament cloth is woven with more than one fibre or many separate fibres. Its examples are silk and polyester.

Screen fabrics are classified on mesh counts, mesh opening and the percentage of open area or ink coverage area. The screen fabric mesh is classified according to the number of threads or wires (openings) per linear cm thread or wire diameter. The weave style and mono-multi-filament are important factors to be considered along with the mesh classification. The larger the mesh classification number, the more wires per linear cm and therefore, the smaller the openings, the finer will be the resulting circuit image. However, it may be sometimes be more difficult to push the screen ink through the small screen openings onto the substrate surface.

Multi-filament polyester is not used in circuit printing. It is used basically in ceramic printing. Mono-filaments like polyester and stainless steel are generally used since it gives more passage for inks to penetrate deposits better and to give uniformity of colour or paste. It produces less drag during the squeezing process and so mono-filaments are used for screen printing with direct emulsion and indirect emulsion methods.

Mono-filament polyester fabrics have a mesh number in the range of 100-140 mesh/cm. However, the applications where a maximum precision, such as registration error to be less than 0.1 mm is required, stainless steel fabrics are recommended for best results. With stainless fabrics, the mesh number is chosen within the range of 120-140 mesh/cm. Typically maximum particle size should be no larger than one-third of the mesh opening to prevent jamming. For example, an 80 mesh screen has openings of about 224 μm , so the particle size should not exceed 75 μm . Finer mesh

screens tend to be used for thinner deposits. A 180-mesh and a fine powder paste could be used to produce a deposit thickness of 100-150 μm , using an 80-mesh.

The thickness of ink deposit is controlled by the fabric material. For determining the correct mesh count of the fabric, theoretical colour volume is used. The theoretical colour volume is the amount of ink held in the fabric prior to the passage of the squeegee during the printing stroke, and it is determined by the thread count, thread thickness and mesh opening (Figure 7.6).

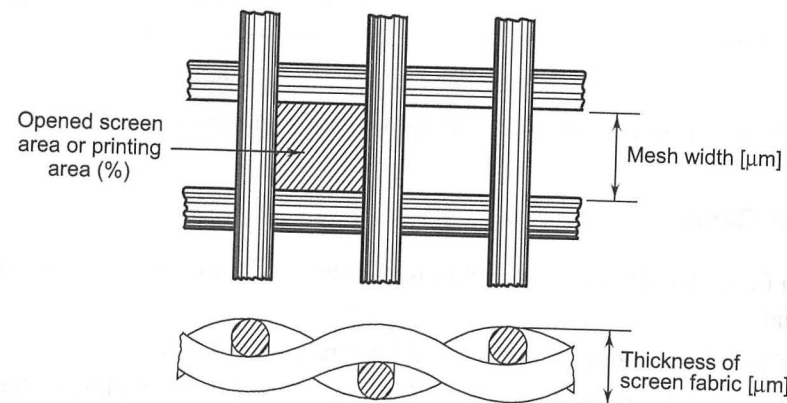


Fig. 7.6 Geometry of a screen fabric

The edge definition of the image produced by the printing screen varies according to the thread thickness, (d) and mesh opening, Mo . The printing fabrics differ according to the quality of the threads (HD, T, S—thread types). The most commonly used thread types for screen printing in the PCB industry are T-type and S-type, as shown in Figure 7.7.

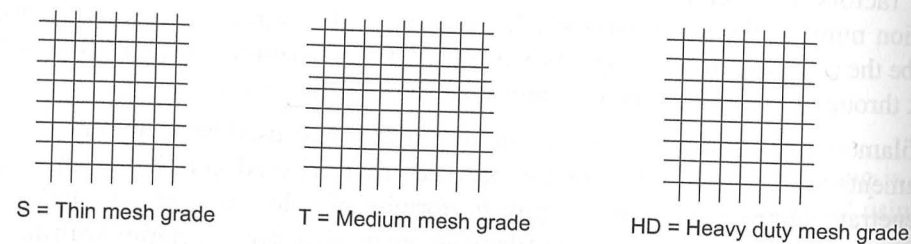


Fig. 7.7 S, T and HD mesh grades

7.3.3 Screen Preparation

The screen is prepared by considering various factors such as the image transfer line and space requirement, panel size, fabrication process, run length, etc. It is followed by properly selecting the fabric type, mesh size, frame stencil and ink. Once this is done, the fabric is cut several cm longer

than the frame moulded into the rails of the frame with staples and adhesives, under maximum tension uniformly applied by hand.

The correct stretching of the fabric is very important in the process of circuit printing. Insufficient stretching of fabric will cause poor registration of the printed detail, colour, pattern, off-contact printing, loss of open mesh and loss of ink passage. During the stretching, the threads should be parallel or at an angle of 45° to the sides of the frame, and the frame should be rigid. The sides should not bend during stretching. The tension of the screen should be maintained between 10 to 15 N/cm (Newtons per centimetre).

The screen is stretched over frame in the following ways:

- The fabric is stretched permanently using special adhesive or nail on the frame and in this case, the tension is not adjustable. For stretching, equipment like a pneumatic system (using air pressure) containing pneumatic clamps can be used.
- The fabric can also be stretched by using a floating bar frame. The screen tension is adjustable in this case.

Once the stretching is complete, the tension of the fabric is measured using the tension gauge. The meter is placed on any part of the screen fabric and the reading becomes available on the gauge dial. The gauge measures tensions from 7 to 50 and the dial is calibrated in Newtons per centimetre (N/cm).

The stretched frame is removed and the fabric is thoroughly cleaned and de-greased with a proper de-greasing agent. The de-greasing agent should not affect the screen fabric. It can be 20 per cent sodium hydroxide or disodium phosphate or any other de-greasing solution recommended by the screen fabric manufacturer.

After the screen is de-greased, by the screen is rinsed with warm or cold water. The screen is then dried by using a fan or a blower which gives hot air. The screen is now ready for printing.

7.3.4 Squeegees

The squeegee used in the screen printing process consists of an elastomeric material such as urethane or a strip of rubber which is mounted on a wooden or an aluminium holder (Figure 7.8a). The material is available in various thicknesses consisting of different compositions of natural and synthetic (polyurethane) rubber. The flexibility of the rubber is measured in terms of durometers (hardness). Usually:

- 40 to 50 durometer rubber is used in textile industries for printing; and
- 70 to 80 durometer rubber is used in the PCB printing process.

The sharpness of the squeegee is very important for printing. Line sharpness, uniform ink coating thickness and the edge definition depend upon the squeegee's sharpness. The sharpness is reduced after continuous operations as the edges become rounded. Hence, it is required that the sharpness be maintained. This is done by sandpaper or any other abrasive material. Over-sharpness also damages the screen. The application of a squeegee is shown in Figure 7.8b.

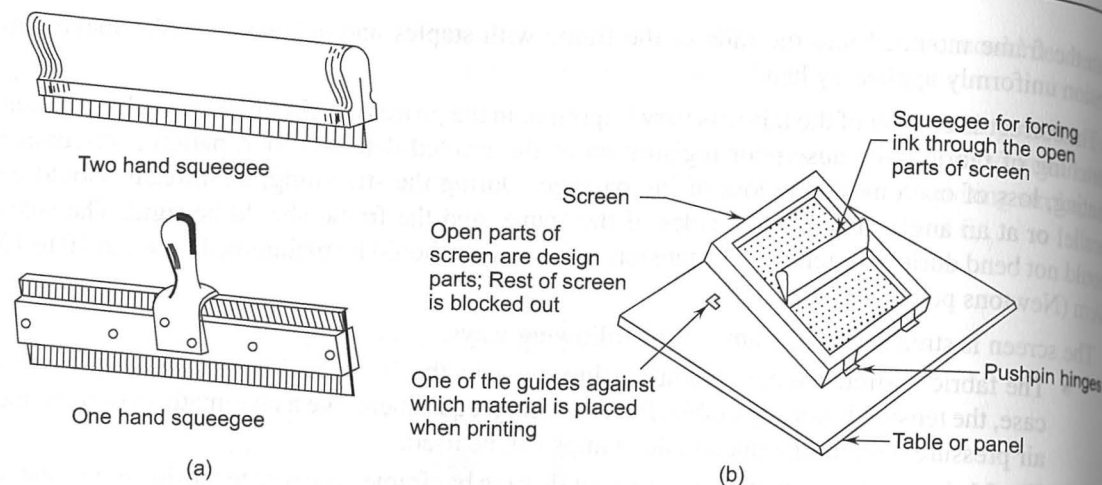


Fig. 7.8 (a) Types of squeegee (b) application of a squeegee

7.4 Pattern Transferring Techniques

7.4.1 Screen Stencil Method

In this method, a photographic emulsion is coated directly on to the screen fabric and is exposed to light, thereby establishing a direct contact with the film master. When the screen is developed, the mesh opens at the pattern areas. The photographic emulsion used is polyvinyl alcohol, polyvinyl acetate or polyvinyl chloride, and they contain some dye to make the pattern visible while processing.

The emulsion is not in its sensitized form. In order to sensitize it, the emulsion is mixed with a sensitizer before use. Some of the sensitizers used are potassium dichromate, ammonium dichromate or sodium dichromate.

The sensitized emulsion-coated screen is exposed to ultraviolet or other fluorescent light sources. The exposure time depends upon the light intensity and the distance between the bulb and the screen. Normally the distance between the bulb and the screen is 1.5 feet.

During the process of exposing, the positive or negative film is placed on the printing side of the emulsion-coated screen. The exposure brings about a polymerization in the exposed areas, which makes them insoluble in the developer. If any air gap is present between the photo-tool and screen, it must be removed by using a vacuum system. The air gaps can lead to incomplete pattern forming and insufficient half-tone work. Vacuum printing facilitates sharp details of line, half-tone work and no pin holes. Vacuum printing machines with light source and all the other regulating facilities are available in the market. The suppliers of the emulsion also specify exposure calculations that ensure the correct exposing time.

Developing is carried out immediately after the exposure, keeping the screen with the emulsion side up. The screen is rinsed by using a strong water supply of hot or cold water till the mesh at the pattern areas opens and the sharp pattern with all details is visible. The screen is then dried well with cold or warm air of 40 °C maximum. The screen is then thoroughly inspected and is then ready for printing.

The direct method provides very durable screen stencils with a high dimensional accuracy. However, the finest details are not reproduced. There may be porosity in the washout stencil, which could be due to thin coating, too short exposure, too old emulsion, too long developing or improper sensitizing.

7.4.2 Indirect Method [Transfer Type Screen Method]

In this method, a separate screen process film supported on the backing sheet is used. The film master is exposed on to this film, followed by film developing which dissolves the unexposed areas of the film and leaves the rest on the backing sheet. The film on its backing sheet is thereafter pressed on to the screen fabric and stuck. Finally the backing sheet is peeled off, opening all those screen meshes which are not covered by the film pattern. Thus the screen is prepared with photographic screen printing films and this is known as the indirect or transfer type screen method. This process is carried out only in low intensity illuminated rooms with yellow, orange or red lights.

Emulsions are available as unsensitized or pre-sensitized products. The unsensitized emulsions are readily mixed with the correct amount of sensitizer and are applied directly from the original container on to the fabric. Pre-sensitized emulsions eliminate possible mixing errors and save working time, and are sold with a prescribed shelf life. In the transfer type screen method, the following commercially available films are used: (a) Chromoline film, (b) Five-star film, and (c) Capillary film.

The emulsion container must be kept under controlled humidity conditions, well closed and kept in low intensity lighted surroundings. However, the shelf life is seriously affected.

The indirect method is more suitable when small quantities of PCBs are to be manufactured. The method is faster but dimensionally less accurate. The screen stencils are less durable, and more sensitive to mechanical damages and interruption of the printing process.

The films should be handled in yellow rooms only. They should not be subjected to bends as this will affect adhesion to the screens. Any contacts of the moisture with the films must be avoided.

7.4.3 Knife-cut or Hand-cut Film Process

For electronic circuit printing, the manually prepared screens are usually of the knife-cut film types. This consists of a transparent or translucent film coating. The emulsions on the knife-cut film are either coated or adhered semi-permanently with an adhesive or by means of the electrostatic process to a transparent or translucent plastic or paper backing sheet. The film may be either of the lacquer or of water soluble type; water soluble film may be softened or dissolved, or adhered with water, water and alcohol, or with an adhering liquid recommended by the manufacturer of the film. The film or coating used for the printing screen must be one that will not get dissolved by the inks or

and through adhesion to copper. It is also difficult to cure completely when the wet film thickness increases.



7.6 Printing Process

The circuit patterns are screen printed on the substrate by two ways: (1) Manual screen printing process, and (2) Automatic or semi-automatic screen printing process.

7.6.1 Manual Screen Printing Process

Manual screen printing is carried out on a sturdy workbench. The patterned screen is fixed in a frame holder held by a high mechanism. A squeegee is selected with width of 2-4 cm more than the pattern width. The squeegee speed and squeegee pressure are controlled manually. The quality of the printing depends upon the operator skill. This type of process is used in the manufacture of low cost non-professional boards.

7.6.2 Automatic or Semi-automatic Screen Printing Process

In the automatic screen printing technique, the machine operates with a hydraulically controlled squeegee for constant speed and a pneumatically controlled squeegee for constant pressure. The squeegee pressure and speed are constant in all strokes, so the quality of printing is more uniform in this case than in the manual printing process. Normally all inner layers of the multi-layer boards are printed by this method.

Advances in the design and operation of screen printing machines have resulted in a progressive reduction in the use of manual machines. Semi-automatic and fully automatic machines are now available with automatic feeders, racks and conveyORIZED curing units to make them very efficient. Standard panel sizes, 0.25 to 0.4 mm line and space and long run lengths, are most suitable for machine printing. However, the highest resolution in screen printing is only obtained by manual means as skilled persons are able to adjust their technique to changes in ink viscosity, humidity, screen tension and conditions of the substrate. But the manual method is generally too slow for most PCB operations.



7.7 Photo Printing

Photo printing of PCBs basically means applying photo-sensitive material having the ability to form a continuous film, which is sensitive to light or other radiation so that the exposed (or unexposed) areas of the film can be further processed without affecting the unexposed (or exposed) areas. This photo-sensitive material is called 'photo-resist'. The essential property of a photo-resist is that an

exposure to proper radiation must produce a change in it to enable a clear distinction in the later operations between the exposed and unexposed areas. Usually, in all the photo-resists, a light-induced change in solution forms the basis of their action.

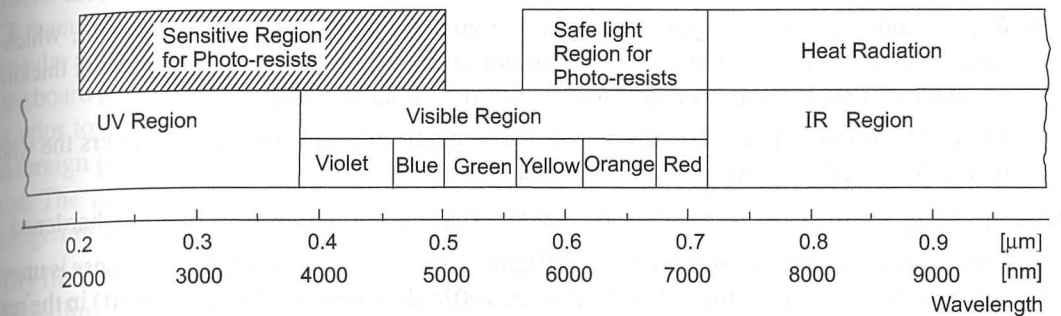


Fig. 7.10 Photo spectrum and sensitive region for photo-resists

The photo-resists are sensitive to ultraviolet light with a wavelength of 200-500 nm. Like the photo-footing film, the photo-resist has a safe visible light area in the yellow to red wavelength (560 to 700 nm). This is shown in Figure 7.10.

Photo-resists can be classified as wet film (liquid film) and dry film (solid film) resists. Irrespective of whether wet or dry resists are used, the following steps are applied for image transfer with the photo printing technique. The coated board is exposed through the appropriate negative or positive film to ultraviolet light. If the board is double-sided, a double-sided exposure is made by ensuring that the patterns on both sides are aligned by matching up, special punch holes on the boards and photographic film. Exposure times can range from a few tens of seconds to several minutes. The resist is then developed leaving those portions of the copper to be retained on the board covered by the resist.

In the case of liquid resists, however, two additional steps are employed. These are: (1) Applying a dye to the resist so that it is visible on the board, and (2) Baking the resist at about 80-100 °C for ten minutes so that the resist is hardened before the etching process.

7.7.1 Liquid Photo-resist (Wet Film Resist)

The wet film resists are organic liquids which, when exposed to light of a proper wavelength, chemically change their solubility to certain solvents (developers). They are available in unsensitized (two-component system) and pre-sensitized (single-component system) forms. The chemical constituents, the mixing ratios, the residual solvents and/or monomer levels are all important in determining the correct chemical and physical performance of the wet film resist. The wet film resist is coated on the laminate by several techniques. These are detailed below.

- **Dip:** The board is dipped into the resist and pulled out slowly. The coating is usually thicker at the bottom of the board than the top.

8.10 Solder Mask

A solder mask (resist) is a coating used to mask or protect the untinned copper track from chemical and abrasive damage which may take place from the action of an etchant, solder or plating. The solder mask also masks off a printed circuit board surface and prevents those areas from solder shorts during wave re-flow soldering (vapour phase or wave soldering process). Such masks are screen printed and are about 0.1 mm thick. In addition, the solder mask provides an environmental protection to the track, acts as an insulating barrier between closely placed cards and prevents the board from getting damaged by dirt, fingerprints, etc. However, the primary function of the solder mask is to restrict the molten solder pick-up or flow to those areas of the printed circuit board, holes, pads and conductor lines that are not covered by the solder resist.

The solder mask in general is expected to perform many more functions than just being a means to restrict the solder pick up (Tennant, 1994). These functions are:

- Reduce solder bridging and electrical shorts;
- Reduce the volume of solder pick-up during wave soldering thereby reducing the weight and cost;
- Provide an environmental protection to the track;
- Act as an insulating barrier between closely placed tracks;
- Prevent the PCB from getting damaged by dirt, fingerprints, etc.; and
- Reduce the solder pot contamination during wave soldering.

8.10.1 Solder Resist Classification

The solder resists are broadly classified into two types:

- Temporary solder resists; and
- Permanent solder resists.

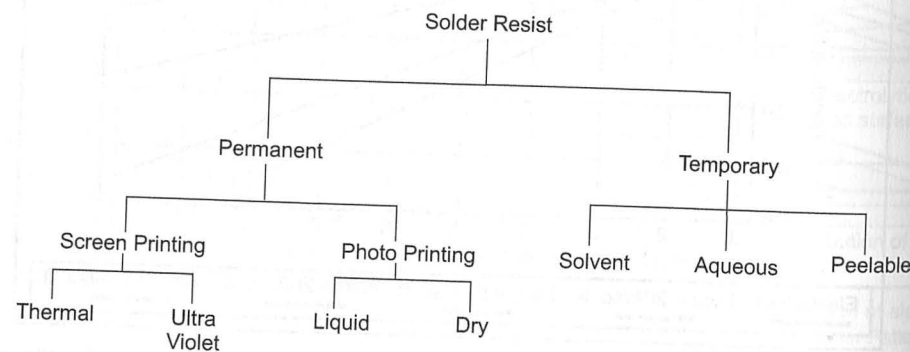


Fig. 8.20 Types of solder resist

Figure 8.20 shows the various types of temporary and permanent solder resists. The permanent resists are applied either by the screen printing method or photo-printing method. The screen printable type can be further classified as thermal curable or ultraviolet curable. The photoprinted method can use either the liquid film solder mask or dry film solder mask. The temporary solder resists are classified on the basis of the chemistry used for developing the resist.

8.10.1.1 Temporary Resists

These resists are used temporarily either to prevent the solder from getting on to certain holes and on to the gold plated edge connectors during wave soldering operations or to prevent certain holes from getting blocked with solder thereby preserving these holes for manual component insertion at a later stage.

8.10.1.2 Permanent Resist

These resists are applied permanently on the PCB surface and form an integral part of the PCB. The chemistry and the demand for this permanent solder masking have undergone a tremendous change with the introduction of surface mounted devices and also due to the fact that the conductor spacing is going down day-by-day. With the increasing complexity of the circuitry, the inspection and the re-work costs are going high. The process of solder masking to a certain extent reduces these costs by generally bringing down the solder bridging and circuit shorts. Also, solder masking increases the environmental protection for the PCB.

8.10.1.3 Solder Mask Application

The solder mask or solder resist are applied either by screen printing or photoprinting method. Figure 8.21 shows the types of solder resist applications.

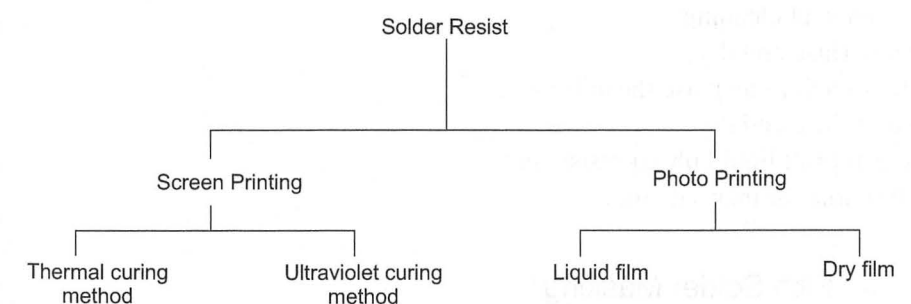


Fig. 8.21 Solder resist application methods

The solder mask for pads or lands used to mount through-hole leaded components is designed to allow a clearance of at least 0.25 mm around the pad, i.e. the edges of the pad are to be covered by the solder mask. This is shown in Figure 8.22.

will also dissolve epoxy coated components and even the epoxy glass PCB itself. Component replacement, therefore, necessitates burning through the epoxy coating with a knife or soldering iron.

Polyurethane Coatings: They are available as either single-component or two-component systems. They provide excellent humidity and chemical resistance as well as outstanding dielectric properties over extended periods. Polyurethane can be burnt through with a soldering iron, thereby simplifying component replacement.

Although cleaning is a vital step prior to the application of any conformal coating, polyurethanes are particularly sensitive to moisture, as it can cause blistering under humid conditions. This can eventually lead to circuit failure. Polyurethane formulations require careful application and close control of coating and curing environments.

Silicone Conformal Coatings: They are particularly useful for high temperature service up to about 200 °C. They provide excellent humidity and corrosion resistance as well as good thermal endurance, making silicone conformal coatings highly desirable for PCBs that contain high heat-dissipating components.

The typical electrical and thermal performance characteristics of conformal coatings are given in Table 8.4. (Waryold and Lawrence, 1991)

Table 8.4 Characteristics of Conformal Coatings

Electrical	Acrylics	Epoxies	Silicones	Polyurethanes
Dielectric strength (short time), 23 °C volts/mil at 0.001"	3,500	2,200	2,000	3,500
Surface resistivity at 23 °C, 50 per cent rh, ohms/cm	10 ¹⁴	10 ¹³	10 ¹³	10 ¹⁴
Dielectric constant at 23 °C, 1 MHz	2.2–3.2	3.3–4.0	2.0–2.7	4.2–5.2
Thermals				
Resistance to heat, continuous, °C	125	125	200	125
Linear coefficient of thermal expansion, $\mu\text{in./}^\circ\text{C}$	50–90	40–80	220–290	100–200
Thermal conductivity 10 ⁴ cal/ sec/cm ² /cm/°C	4–5	4–5	3.5–8	4–5

8.11.2 Methods of Applying Conformal Coatings

There are basically four methods of applying conformal coatings. These are discussed below. (Waryold, et al., 1998):

Dipping: In this method, the masked assembly is immersed in a tank of liquid coating material and withdrawn, ensuring uniform coverage and even deposit. Immersion and withdrawal rates are important factors that need to be controlled to allow the viscous liquid material to fill all the voids in the assembly. The typical immersion speeds are 2 to 12 inches per minute to enable the coating to displace the air surrounding components.

When conformal coatings are applied by dipping, the evaporation of the solvents in the liquid plastic may occur so fast that the viscosity of the bath increases rapidly. For this reason, it is important to continuously monitor and maintain proper viscosity.

Spraying: Spraying is the most popular and the fastest method for applying conformal coatings. With the proper combination of solvent dilution, nozzle pressure and patterns, consistent and repeatable results can be obtained. Spraying can be done by both manual or automated means by using computer-controlled systems integrated into existing wave soldering and cleaning lines. A major drawback of the spraying method is that little or no coating is applied underneath components or on components that are shadowed.

Coatings should be sprayed on assemblies using clean, dry gas at the minimum pressure necessary to provide good atomization. The assembly should be sprayed holding the gun at a 45° angle and the assembly should be rotated 90° after each back and forth pass.

Brushing: It involves manual operation. It is the least effective method of applying conformal coatings because of the difficulty in achieving uniform and repeatable coverage. The method is practical only for small numbers of printed circuit boards.

Each coating method has its strengths and weaknesses. A combination of dipping and spraying is better than the adoption of a single method.

8.11.3 Standards for Coatings

Some coatings are manufactured strictly for commercial use and are not certified by the manufacturers to meet any particular specifications. The military specification for coatings was Mil-I-46058C, but this is obsolete and the new specification is IPC-CC-830. Coatings are available that also meet UL requirements for components.

8.12 Useful Standards

- *IPC-4552: Specification for Electroless Nickel/Immersion Gold (ENIG) Plating for Printed Circuit Boards:* Sets the requirements for the use of electroless nickel-immersion gold as a

The rule of thumb for calculating the minimum distance states that the minimum thickness of the prepregs after bonding must be at least twice the thickness of the copper being embedded. In other words, where you have two adjacent copper layers, each of which is $30\text{ }\mu\text{m}$ thick, a minimum prepreg thickness of $2 (2 \times 30\text{ }\mu\text{m}) = 120\text{ }\mu\text{m}$ is required, which can be achieved by using two prepregs (1080 is the type of fibre-glass fabric).

11.4.1.4 Inner Layer Copper

The most commonly used copper is 1 oz. (one ounce of copper foil per square foot area of surface area). However, for dense boards where board thickness is crucial and which require tight impedance control, 0.5 oz copper is used. Heavier copper, of 2 oz or above is preferred for voltage and ground planes. However, etching heavier copper results in reduced control of the desired pattern with regard to line width and spacing tolerances. Special processing techniques are thus required.

11.4.1.5 Holes

The plated through-hole diameter is generally kept between 0.028" and 0.010" from the nominal component lead diameter or diagonal to ensure sufficient volume for good soldering.

11.4.1.6 Aspect Ratio

The 'aspect ratio' is the thickness of the boards as compared with the diameter of the drilled hole. An aspect ratio of 3:1 is generally considered standard, though higher values like 5:1 are not unusual. The aspect ratio is determined by considerations such as drilling, smear removal or etch-back and plating. Via holes are required to be kept as small as possible, while keeping the aspect ratio within a producible range.

11.4.2 Electrical Design Considerations

A multi-layer board is a high performance, high speed system. At higher frequencies, the signal rise times decrease and consequently, signal reflections and line lengths become critical. The multi-layer board is a critical electronic component of the system with controlled impedance characteristics, designed so as to accommodate the above effect. The factors which determine impedance are the dielectric constant of the laminate and prepreg, conductor line width spacing between one layer of the conductor, dielectric thickness between layers, and thickness of the copper conductors. The layering sequence of conductors in the multi-layer board and the sequence in which the signal nets are connected are also critical in high speed applications.

Dielectric Constant: The dielectric constant of the laminate material plays a major role in the determination of impedance, propagation delay and capacitance. The dielectric constant of the epoxy glass used for the laminate and the prepreg can be controlled by varying the percentage of the resin content.

The epoxy resin has a dielectric constant of 3.45 and glass of 6.2. Depending upon the percentage of these materials, the dielectric constant of epoxy glass can be achieved from 4.2 to 5.3. The thickness of the laminate is a good indicator for determining and controlling the dielectric constant.

Prepreg materials with low relative dielectric constants are suitable for use in radio frequency and microwave engineering. The low dielectric constant gives rise to a low signal delay at radio and microwave frequencies. Electrical losses are minimized by low loss factors in the substrates.

Prepreg ROR 4403 is a new material produced by ROGERS CORPORATION (<http://www.rogers-corp.com/mwu/index/html>). This material is compatible with other substrates (such as RO 4003 or RO 4350, used for microwave boards) used in the construction of standard multi-layers (FR-4 material).



11.5 Fabrication Process for Multi-layer Boards

11.5.1 General Process

Multi-layer boards are produced by bonding together inner layers and outer layers with prepreg. Prepreg, as explained earlier, is fibre-glass fabric impregnated with partially hardened resin. The individual layers are arranged and bonded by placing them in a pressing tool to prevent misalignment of the layers.

After bonding, the bonded layers are further processed as double-sided through-plated circuit boards. Due to smearing of the hole with epoxy that takes place during drilling, through-hole wall cleaning is required before through-hole plating can take place.

In a multi-layer board, the outer layers may consist of either copper foil and prepreg or of single-sided or double-sided copper clad laminates. The inner layers consist of double-sided copper clad, etched (with structured conductor tracks created) and through-plated board material.

The inner layer etching is done by standard printed circuit techniques. Before bonding, it is important to make a very careful layout for each one of the layers in order to prevent masking of the desired holes. Each layer of B-stage and board substrate requires a different hole arrangement. In order to prevent the flow of resin into aligning pins, the tooling and aligning holes on the prepregs must be 1.25 mm larger in diameter than those of the conductor pads. On the other hand, the holes in the laminates must be 1.25 mm smaller than the pads over which they are to be placed.

In order to illustrate the process, let us take the construction of a four-layer board. The process steps are shown in Figure 11.4. They basically consist of two single-sided laminates and one double-sided laminate with two sheets of prepreg. The process starts by making a sandwich of all the required panels, stacked in the following order from bottom to the top:

- Thermal insulation material (a) to control rate of temperature rise;
- Bottom laminate fixture or caul plate (b);
- Sheet of release material (c) such as Teflon glass cloth;
- Bottom circuit panel (d);

changes with the base materials. Inward looking corners in the contour should be rounded; sharp inward corners could initiate tearing of the board.

Areas of small conductor widths and spaces should be minimized as much as possible. Conductor width should transition from fine lines in tight areas to wider widths where geometry allows. Conductors terminating at PTH vias or component mounting holes should have a smooth fillet transition from the trace into the pad as shown in Figure 12.9. As a general rule, any transition from a straight line to such features as corners or different line widths must be done as smoothly as possible. Sharp corners constitute a natural place for stress to accumulate and for conductor defects to occur.

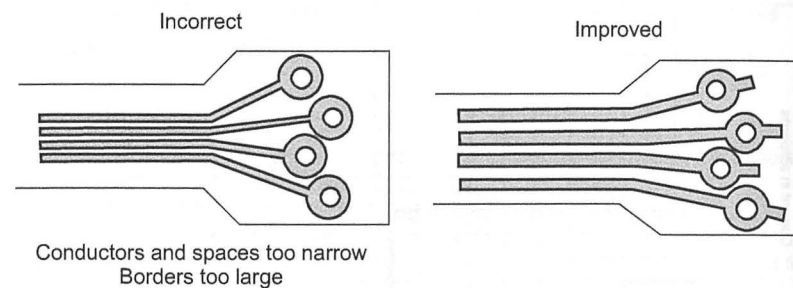


Fig. 12.9 A smooth transition from trace to pad reduces stress and improves reliability. In the top view, the conductors and spaces are too narrow, and the borders too large, the bottom view shows an improved design

Bending: As a general rule, the bending radius should be designed as wide as possible. The possibility of undergoing many cycles is also further improved with thinner laminates (e.g. 50 μm foil instead of 125 μm foil) and larger conductor widths. If subjected to a higher number of bending cycles, single-sided flexible PCBs, in general, show a better performance.

Solder Pads: Around the solder pad, there will be a transition from flexible to rigid material. This zone is highly prone to conductor breakage. Solder pads are therefore, avoided in active bending zones.

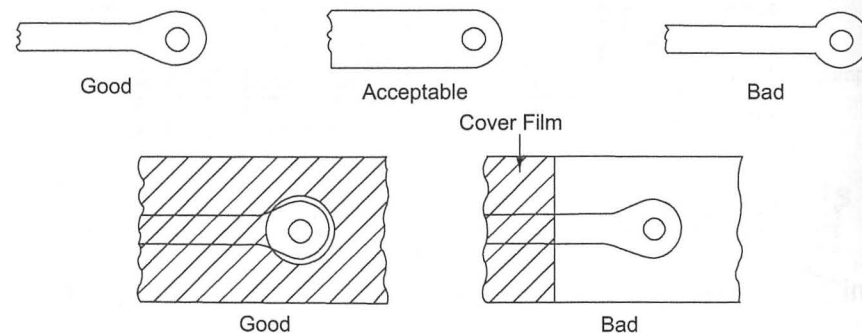


Fig. 12.10 Shape and masking of solder joints (a) shape of solder pads (b) solder joint masking with cover film (after Bosshart, 1983)

The general shape of solder pads should be tear-like (Figure 12.10) and the cover foil must mask the solder joint close by.

Hardboard Stiffeners: The combination of flexible PCBs with adhesively laminated hardboard stiffeners has become extremely popular and cost-effective in the bulk production of small electronic equipment like pocket calculators. The flexible PCB is mounted on one piece of hardboard (e.g. grade G-10) with suitable slots for separating at a later stage. This is illustrated in Figure 12.11. After component assembly and wave soldering, the cutting operation divides the hardboard into different parts, thereby facilitating folding into the planned shape.

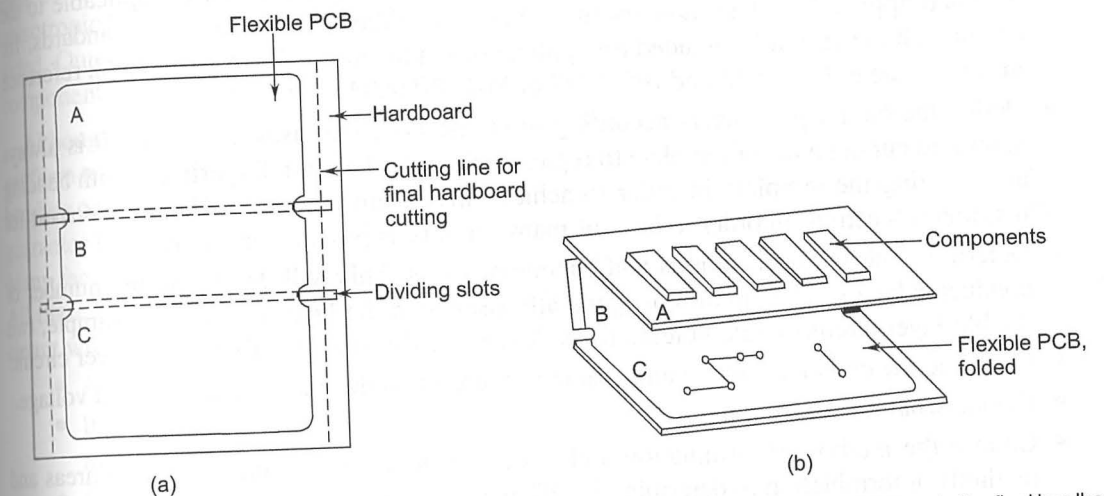


Fig. 12.11 Use of hardboard stiffeners (a) Board as it goes for assembly and wave soldering (b) Board after final hardboard cutting and bending

The above special requirements indicate that designing a flex circuit is only a few steps away from designing a hardboard. However, the important design differences to be kept in mind are:

- Three-dimensionality of a flex circuit is important as creative bending and flexing can save space and layers
- Flex circuits both require and permit looser tolerances than hard-boards
- Because since arms can flex, they are designed slightly longer than required.

The following design tips are useful for minimizing circuit cost:

- Always consider how circuits will be nested on a panel.
- Keep circuits small; consider using a set of smaller circuits instead of one large circuit.
- Follow recommended tolerances whenever possible.
- Design unbonded areas only where they are necessary.
- If circuits have only a few layers, stiffeners can be far less expensive than designing a rigid flex circuit.

soldering should be employed. The following precautions should be observed while soldering flexible circuits:

- Since polyimide absorbs moisture, circuits must be baked (one hour @ 250 °F) before soldering.
- Pads that are located in large conductor areas such as ground planes, voltage planes, or heat sinks, should be provided with relief areas, as shown in Figure 12.16. This limits heat dissipation and therefore makes for easier soldering.
- When hand-soldering pins in dense clusters, try not to solder adjacent pins one after another. Move around to avoid local over-heating.

Information on the design and processing of flexible circuits can be obtained from several sources. However, the best sources are always the producers/suppliers of process materials and chemicals. High quality flexible circuits can be produced, by using the suppliers information in combination with the experience of the process scientist.



Fig. 12.16 Solder relief area around pads

12.5 Rigid Flex Printed Circuit Boards

Rigid flex printed circuit boards constitute a hybrid system in printed circuit board technology. The manufacturing process uses standard multi-layer technology along with standardized manufacturing methods for flexible printed circuit boards. The various combinations of layers are laminated together. After lamination, the through-holes are drilled and then plated so that they connect the various layers together electrically. The rigid-flex assemblies eliminate the need for jumpers and mother-daughter board combinations, thereby reducing wiring errors while increasing packaging density.

The selection of the **material** for rigid flex assemblies is very important as all the component parts must withstand process in steps without damage. The dimensional changes in the X-Y directions as in the flexible circuits and the vertical stresses in the Z-axis arising due to the stacking of the layers need to be carefully considered. This distortion of materials can cause copper hole barrel cracking when the materials are exposed to final lamination and soldering temperatures.

The **processing** of rigid flex circuits involves four difficult areas: laminations, drilling, removal of adhesive and plating of the holes. In drilling, it is necessary to ensure that the centre of all successive pads in all the layers meets correctly. The adhesive smear removal from the hole may cause hole cracking or misregistration of various layers. These problems can only be overcome with careful processing.

After putting together all the component parts of the rigid flex, the next step is to combine the flexible circuits to the rigid circuits. As most of the component parts are made of similar materials, it is a difficult job. Many companies use the standard platen presses but the modern trend rears

towards the use of vacuum **lamination**. Suppliers of flexible and rigid materials usually provide technical assistance on request for carrying out this procedure.

The next step is **drilling** the laminated package. Since many different materials are required to be drilled, the fields and drills should be optimized. After the drilling process, the holes of the stacked materials are cleaned, so that the subsequent copper plating process will yield a hole with no cracks or voids. The **cleaning** can be done by sulphuric acid for epoxy-based system or by plasma etch-back system for acrylic-based materials.

After the holes are drilled and cleaned, they are copper **plated** with electroless copper. Etch-resist placement provides the pattern for the top/bottom services. The final surface and hole electrolytic plating is carried out and the panels are solder plated. The resist is then removed thereby allowing the solder plating to become the new resist. **Etching** then removes the unwanted copper.

Multi-layer rigid flex assemblies often present a difficulty in holding the tolerances. Therefore it is advisable to limit the number of layers as much as possible in rigid flex systems.

12.6 Terminations

There are a variety of possibilities for providing terminations (www.minco.com) for a flex circuit. However, the commonly used terminations are discussed below.

Connectors: Connectors for the flex circuits can be attached (Figure 12.17) by hand soldering, wave soldering or crimping in the case of IDCs (insulation displacement connectors). Connectors can be potted after attachment or conformally coated for protection and insulation with epoxy, polyurethane, etc. Clincher connectors are a good option for many applications.

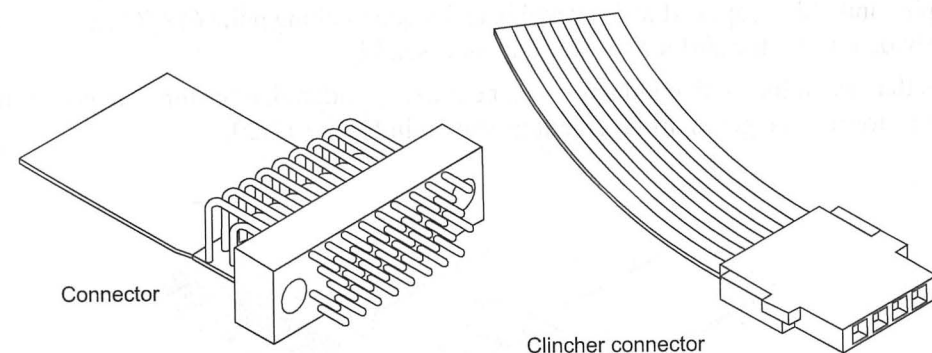
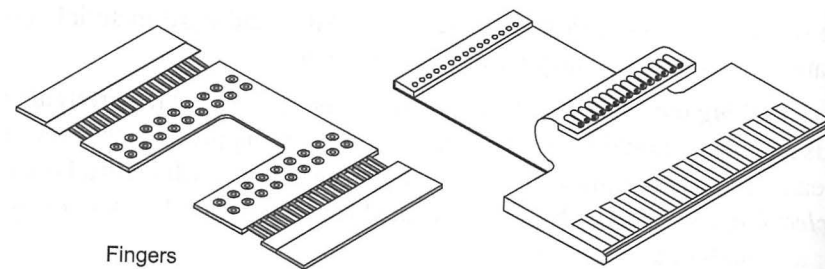


Fig. 12.17 Clincher connectors used with flexible circuits

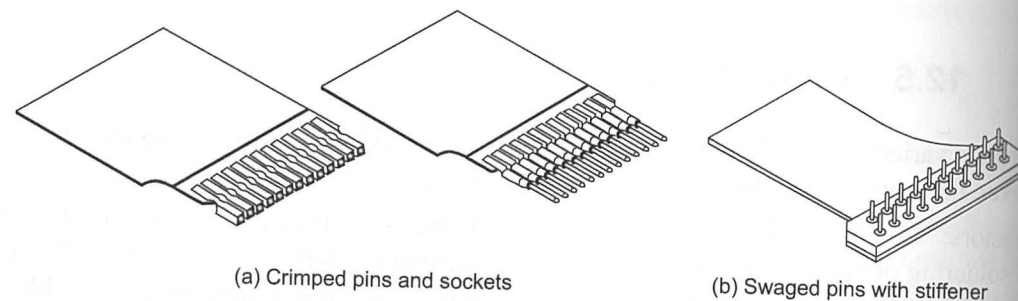
Fingers: Fingers can be supported or unsupported as shown in Figure 12.18.



Fingers

Fig. 12.18 Fingers used with flexible circuits

Pins: Socket pins can be pressed in place and then soldered. Pins can also be swaged to the circuit and soldered after the swaging procedure, or pins can be swaged (Figure 12.19) to a FR-4 stiffener and then soldered.



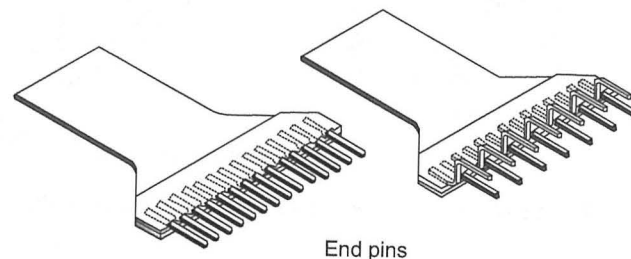
(a) Crimped pins and sockets

(b) Swaged pins with stiffener

Fig. 12.19 (a) Crimped pins and sockets (b) Swaged pins with stiffener

Brazed pins and ribbon, applied with a welding technique (melting point 618 °C), are also available, and can provide a weldable surface for subsequent assembly.

End pins that are in line with conductors can be brazed, soldered, or crimped to conductors. Pins can be sent to form a staggered arrangement as shown in Figure 12.20.



End pins

Fig. 12.20 End pins used with flex circuits

Lap Joints: Flex circuits can interface with hardboards via soldered lap joints, lap joints applied with an anisotropic adhesive (conductive in the Z-axis only), compression dots using a raised metal dot, or 'zebra' strips. Compression methods usually require a stiffener behind the contact area.

12.7 Advantages of Flexible Circuits

Flexible circuits can solve/minimize design/packaging problems in electronic products. The following are the specific advantages on these accounts:

- Replace multiple hardboards and connectors with a single flex circuit or rigid flex circuit.
- Replace hardboard/ribbon cable assemblies.
- Control EMI with solid or patterned shield layers.
- Control impedance with integral ground planes.
- Provide electrical connections through gaps that are too narrow for round wires.
- Design circuits with exposed conductors for use with conductive elastomer key pads.
- Integrate heaters, temperature sensors or wire-wound antenna coils into a flex circuit.
- Use flex circuits as jumpers between hardboards.
- Specify pressure-sensitive conductive adhesive to attach circuit to cabinets or enclosures.

All these advantages have led to the applications of flexible circuits in some the crucial areas like the military and aerospace field, medical field and commercial applications.

12.8 Special Applications of Flexible Circuits

There is a growing market for flexible printed circuits which is driven purely by the special applications that they find in some of the most critical areas. Some of these applications are enumerated below.

- Flexible base materials are light and require less space than the rigid laminate. Therefore, they are preferred for use in space, military and mobile applications.
- With the possibilities for three-dimensional interconnections and construction, there is an increasing number of applications in the telecommunication industries and household appliances, which has even led to substantial cost reductions.
- Flexible printed circuits can be shaped into various forms such as tapes, coils or bent shapes. Therefore, they can be used as machine parts where rigid materials are not suitable, for example, printers, aerials, drawing machines; etc.
- The replacement of rigid cable harness by flexible printed circuit boards has made the field servicing of equipment convenient.
- Flexible printed circuits show a higher current-carrying capacity than the wires because of rapid heat dissipation. Thus, they provide a better volume to surface ratio.

(Holden and Kenyon, 1994). DFM/A essentially focuses on several separate domains such as: optimization of printed circuit design and layout, minimization of printed circuit substrate costs, minimization of assembly costs, use of preferred parts and analysis of test coverage. Boothroyd (1990) explains that considering product assembly at the design stage can lead to improved product reliability. He suggests the idea of Design for Assembly so as to look ahead and anticipate manufacturing problems at the design stage itself.

13.10 Solder Pastes for SMDs

Solder pastes, often called solder creams, are used in "reflow-soldering" of surface mount components, where the application of solder and the heat supply are separate steps in the fabrication process. A solder paste basically consists of solder in powder form and a flux with some additives necessary to produce the desired behaviour of the paste during or after its application.

Solder pastes are applied either by stencil or by screen-printing method. Pick-and-place throughput is an important issue in deciding the type of solder paste dispensing system. Erdmann (1991) brings out the stenciling technique including stencil development, stencil cleaning and printing etc. He points out that the demands of stencil are more rigid than that of ordinary SMT screen printing and stenciling, particularly for maintaining near perfect registration.

Large pick-and-place systems require volumes that only screen printers can provide. Pick-and-place throughput in the range of 1500 to 3000 components per hour, however, is ideal for today's dispensing equipment, which can produce 16000 dots per hour in a typical production environment (Cavallaro and Marchitto, 1991).

A rotary positive displacement programmable pump as shown in Figure 13.23(a) can be effectively used for solder paste dispensing involving high speed application of a large number of very small dots. The pump is driven by a DC motor. An electro-magnetic clutch engages and disengages the Archimedian screw. Mounted above the screw is a bellows coupling that aligns the clutch and lead screw and reduces the impact of the Z-axis sensor by more than 60 per cent. The combination of constant motor speed, low air pressure, the software-controlled clutch, and the precise rotation of the Archimedian screw ensures a repeatability that is far superior to pulsed air or piston dispensing systems (Cavallaro, 1994).

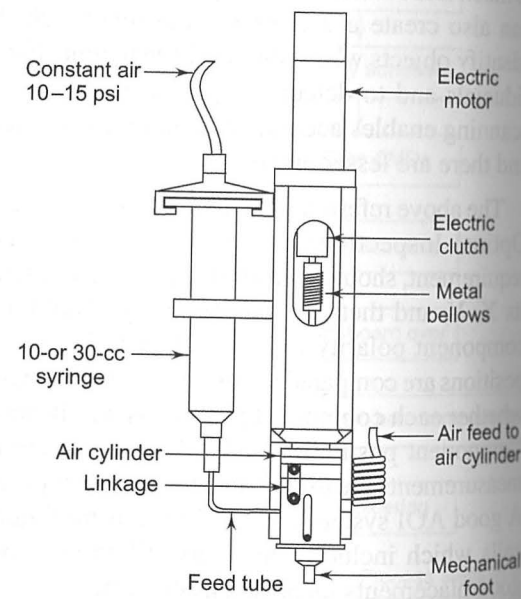


Fig.13.23 (a) The dual-height rotary positive displacement pump, which can perform 25 and 50 mil dispensing within the same program. The pump is appropriate for SMD epoxy or solder paste

The pump can be programmed in one-milliseconds (ms) increments from 10 to 10,000 ms. Most solder paste applications require a shot size of 15 to 20 ms for fine-pitch devices and 50 to 100 ms for 50-mil devices. This short cycle time allows the automated dispensing system to dispense up to 16,000 dots per hour, making it compatible with most pick-and-place machines in the marketplace. The real advantage of dispensing over screen printing is programmability. For example, a 25-mil-pitch application, requires a dot diameter of 0.014 to 0.016 inch (0.35 to 0.40 mm) while a 50-mil-pitch application requires a 0.020 to 0.030 inch (0.50 to 0.75 mm) diameter dot. Dual-height dispensing systems are important to accommodate different dispensing requirements without the need to change either the needle or the programme. Dispensing systems are particularly effective for re-work and mixed technology applications.

13.10.1 Requirements of Solder Pastes

Solder pastes must meet the following requirements:

- The individual powder particles of the solder alloy should have a homogeneous distribution of the metal within the paste as well as a fair equality of shape and surface roughness of the individual powder particles.
- It should develop an adhesive action in order to hold the components in place until the reflow operation has been finished.
- It must not tend to solder balling; if they become detached on the substrate, they may cause short circuits.
- It has to maintain its shape during curing and reflow and must remain on the pads, not leaking to unsolderable parts of the PCB.
- It must have sufficient activity as the solder paste is sometimes in contact with the parts for several hours.

13.10.2 Composition of Solder Pastes

Solder pastes are available with several fluxes and alloy compositions. Their consistency may vary from liquid cream to thick paste. Solder pastes usually contain a certain percentage of silver, most commonly 2 per cent, that gives a solder alloy of 62 % tin (Sn) + 36 % lead (Pb) + 2 % of Silver (Ag). This alloy has a melting temperature of 179 °C. The powder must contain granules of the alloy and not of the individual metals, which make up the solder. This will help to melt the solder at the temperature of the alloy used. Beside homogeneity, an important element in solder powder is the structure and shape of the alloy particles. Therefore, microscopic control of the solder particles is essential. An acceptable solder paste should contain alloy-particles only with the shape of a "sphere", an "ellipsoid" a "tear" or a "dog bone". Chilton and Gaugler (1990) describe the design of solder paste to meet the exacting requirements of fine pitch reflow.

- **Low Grade Material:** This comprises power supply units and television boards having ferrite transformers, large aluminium heat sink assemblies and laminate offcuts.
- **Medium Grade Scrap:** This contains precious metal content, generally from pin and edge connectors used in high reliability equipment.
- **High Grade Material:** This comprises high precious metal content boards, gold containing integrated circuits, discrete components, opto-electronic devices, gold and palladium pin boards, etc.

These grading materials help to determine the economics of applying recovery operations. However, it is possible to regrade the material from low to medium category through selective manual disassembly of high percentage mass ferrous and aluminium components.

Recycling involves the disassembly of scrap PCBs followed by sorting, grading and shredding operations. Iron and aluminium metals are removed from the final ground product by using magnetic and eddy current separation. The output from the recycler is either sent for land-fill or to a smelter. However, only those boards which contain sufficient gold or precious metal content are subject to *smelting*, otherwise all non-precious metal bearing board scrap is consigned to land-fill. About one per cent of the scrap is subject to specialized recycling operations solely for the purpose of precious metal recovery. Figure 15.10 shows a general scheme for PCB scrap disposal/treatment.

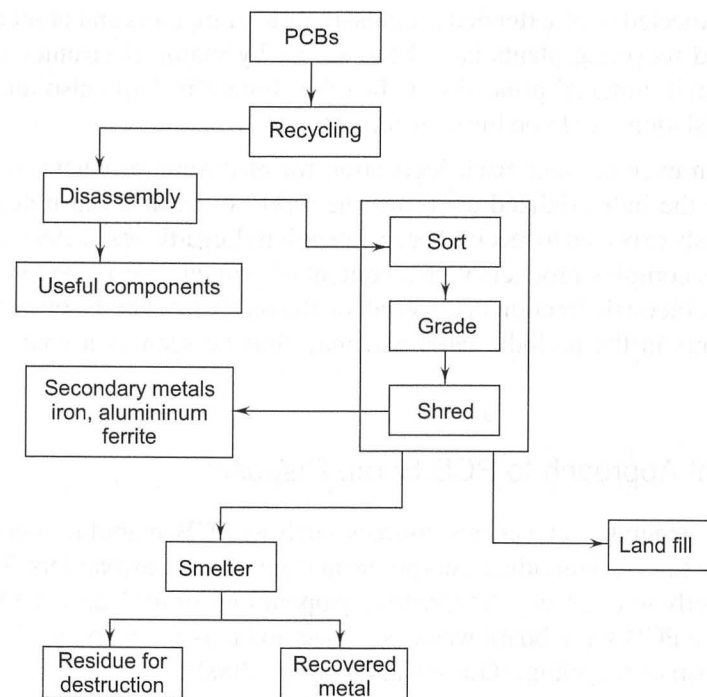


Fig. 15.10 PCB scrap disposal/treatment methodology (redrawn after Goosy and Kellnr, 2002)

15.6.2 Characteristics of PCB Scrap

PCB scrap, particularly populated scrap board, is highly heterogeneous and complex. These boards contain diverse levels of inorganics with relatively low levels of precious metals in conjunction with copper, solders, various alloy compositions, and non-ferrous and ferrous metals (Feldman and Scheller, 1994). The materials and components present in the scrap PCBs have widely differing intrinsic physical and chemical properties. The recycling techniques primarily depend upon various characteristics, which are detailed below.

15.6.2.1 Density Differences

The typical values of the specific gravity of materials contained within scrap PCBs are:

Materials	Specific Gravity range (g/cm ³)
Gold, Platinum group, tungsten	19.3–21.4
Lead, silver, molybdenum	10.2–22.3
Magnesium, aluminium, titanium	1.7–4.5
Copper, nickel, iron, zinc	7.0–9.0
Non-metallic Materials	1.8–2.0

It is evident that various materials can be separated by density-based separation systems, normally employed in the process industry (Barsky, *et al.*, 1991).

15.6.2.2 Magnetic and Electrical Conductivity Differences

The application of low intensity magnetic separators is well developed in the minerals processing industry and can be used to separate ferrous materials from PCB scrap. On the other hand, non-ferrous metals may be separated by means of electrostatic and eddy current separators which are well developed within the recycling industry (Iji and Yokoyama, 1997).

15.6.3 Disassembly of Equipment

In most of the operations for the recycling of printed circuit boards, disassembly is an essential part. The disassembly of components facilitates a selective and profitable recovery of metals and noble metals. In addition, the concentration of valuable materials could reduce the re-processing cost. Another advantage of disassembly is the isolation of hazardous components in order to prevent them from contaminating the shredded waste. Electronic components can be re-used after disassembly for economic and ecological reasons. However, the cost of disassembling, testing and selling the re-usable electronic components has to be seen in relation to the cost of a new product (Keimeier, 1994).

The disassembly of scrap is mostly carried out manually by using simple hand tools, which itself sometimes places limits on all such operations due to the costs involved. Disassembly is considered to be an area of great importance in case of recovery of low cost components and is considered to have a great impact on the overall future recycling strategies. In order to ensure safety during

mounting hole. The hole is used for the attachment and electrical connection of component terminations, including pins and wires, to the printed board.

Component side: (Primary side) The surface layer of a board on which most of the components are placed. Component side is also referred to as the top side (layer one-counting downwards) of the board.

Component: Any of the basic parts used in building electronic equipment, such as a resistor, capacitor, DIP or connector, etc.

Conductive Adhesive: A material to which metal particles (usually silver) are added to increase electrical conductivity.

Conductive Foil: A thin sheet of metal that may cover one or both sides of the base material and is intended to form the conductive pattern.

Conductive Pattern: The configuration or design of the conductive material on the base laminate. Includes conductors, lands, and through connections.

Conductor Base Width: The conductor width at the plane of the surface of the base material. See also: Conductor width.

Conductor Layer: The total conductive pattern formed upon one side of a single layer of base material.

Conductor Pattern: See "Conductive Pattern".

Conductor Side: The side of a single-sided printed board containing the conductive pattern.

Conductor Spacing: The average or minimum (as specified) distance between the adjacent edges of conductors on the same layer of a printed board.

Conductor Thickness: The thickness of the conductor including all metallic coatings.

Conductor-to-hole Spacing: The distance between the edge of a conductor and the edge of a supported or unsupported hole.

Conductor Width: The observable width of the pertinent conductor at any point chosen at random on the printed board, normally viewed from vertically above unless otherwise specified.

Conductor: A single conductive path in conductive pattern. A PCB has at least one layer of conductors. *Synonyms:* path, trace.

Conformal Coating: A thin dielectric cover brushed, dipped or sprayed over parts and circuits of PCBs for environmental and mechanical protection.

Connection: One plug or receptacle which can be easily joined to or separated from its mate. Multiple-contact connectors join two or more conductors with others in one mechanical assembly.

Connector Tongue: A protrusion of the Printed Circuit Board edge that is manufactured to a configuration to mate with a receptacle that provides electrical and/or mechanical junction between the Printed Circuit Board and other circuitry.

Contact Area: The common area between a conductor and a connector through which the flow of electricity takes place.

Contact Resistance: The electrical resistance of the metallic surfaces at their interface in the contact area under specified conditions.

Contact Spacing: The distance between the centre lines of adjacent contact areas.

Contaminate/Contaminant: An impurity or foreign substance whose presence on printed wiring assemblies could electrolytically, chemically, or galvanically corrode the system.

Continuity Testing: A procedure in which voltage is applied to two interconnected lands to ascertain the presence or absence of current flow.

Copper Clad: A material, usually supplied in large sheets, consisting of a base material to one or both sides of which a thin copper foil is bonded. *Synonym:* laminate.

Copper Foil-See Foil: Quality electrolytic copper used to form conductive patterns on printed circuits. It is available in a number of weights (thickness) the traditional weights are 0.5, 1 and 2 ounces per square foot. (18, 35 and 70 μm thick).

Copper-mirror Test: A corrosivity test for fluxes in which the compound's reaction to a copper film vacuum-deposited on a glass plate is noted.

Corner Mark: Marks placed inside the edges of PCB corners to align and register the layers.

Corrosive Fluxes: Materials of inorganic acids and salts for surfaces of difficult solderability. Also called acid fluxes.

Cosmetic Defects: A smear or stain on the board representing flux residue after soldering and a variation from conventional appearance.

Coupon: One of the patterns of the quality conformance test circuitry area. (See: also Test Coupon).

Cover Lay, Cover Layer, Cover Coat: An outer layer(s) of insulating material applied over the conductive pattern on the surface of the printed board.

Cracking: A condition that makes breaks or separations in coatings that extends through to an underlying surface.

Crazing: A condition existing in the base material in the form of connected white spots or "crosses" on or below the surface of the base material, reflecting the separation of fibres in the glass cloth and connecting weave intersections.

Crimp Contact: A type of connector contact whose end is a hollow cylinder that can be crimped onto a wire inserted within it.

Cross-hatching: The breaking of large conductive areas by the use of a pattern of voids in the conductive material.

Cross-linking: The formation of chemical ties between reactive atoms in the molecular chain of a plastic.

Cross-talk: The undesirable interference caused by the coupling of energy between signal paths.

C-stage Material Laminate: The condition of a resin polymer when it is in the solid-state, with high molecular weight, being insoluble and infusible.

Cure: Change of physical properties of a material via chemical reaction or heat and catalysts, usually from a liquid to a solid.

Current-carrying Capacity: The maximum current which can be carried continuously, under specified conditions, by a conductor without causing degradation of electrical or mechanical properties of the printed circuit board.

Cut: Open circuit. An unwanted break in the continuity of an electrical circuit, that prevents current from flowing.

Cycle Rate: A component placement term measuring machine speed from pick-up to board location and return.

Database: A collection of inter-related data items stored together without unnecessary redundancy to serve one or more applications.

Datum: A defined point, line, or plane used to locate the pattern of layer for manufacturing or inspection, or for both purposes.

De-bugging: The process of locating and fixing problems (bugs) in the hardware and software portions of an electronic system.

De-lamination: A separation between any of the layers of a base material or between the laminate and the conductive foil, or both.

Dendrite: Metallic filaments growing between pads and traces resulting from electromigration.

Design Width of Conductor: The width of a conductor as delineated or noted on the master drawing. (See also: Conductor Base Width and Conductor Width).

Device: An individual electrical element, usually in an independent body, which cannot be further reduced without destroying its stated function.

De-wetting: A solder coating that has receded, leaving irregular deposits and indicating that the base metal has not been adequately de-oxidized.

Dezincification: Kind of galvanic corrosion, generally associated with two-phase brass alloys, in which the zinc-rich beta phase is selectively leached out of the brass. It occurs when brazed joints are exposed to salt or seawater.

DFSM: Dry Film Solder Mask.

Dice: Two or more dies.

Die: The uncased and normally leadless form of an electronic component that is either active or passive, discrete or integrated.

Dielectric Constant: The property of a dielectric which determines the electrostatic energy stored per unit volume for a potential gradient.

Dielectric Strength: The maximum voltage that a dielectric material can withstand, under specified conditions, without rupturing (usually expressed as volts/units thickness.)

Dielectric: An insulating medium which occupies the region between the conductors. It is also the distance between bonded inner layer conductors.

Digital Circuit: A circuit comprised mostly of integrated circuits and operates like a switch (i.e., it is either "ON" or "OFF").

Digital Logic Simulator: A tool used to verify the design (or portions thereof) in the digital domain by applying virtual test signals to a virtual model of the design.

Digitizing: Any method of reducing feature locations on a flat plane to digital representation of X-Y co-ordinates.

Dimensional Hole: A hole in a printed board where the means of determining location is by co-ordinate values not necessarily coinciding with the stated grid.

Dimensional Stability: A measure of dimensional change cause by such factors as temperature, humidity, chemical treatment, age, or stress (usually expressed as units/unit).

DIP Soldering: A process whereby printed boards are brought into contact with the surface of a static pool of molten solder for the purpose of soldering the entire exposed conductive pattern in one operation.

Discrete Component: A component which has been fabricated prior to its installation (i.e., resistors, capacitors, diodes, transistors, capacitors, diodes, transistors, etc). A single function component.

Dispersent: A chemical added to the cleaning solution to improve its particulate-removal ability.

Disturbed Solder Connection: A solder connection that is characterized by the appearance that there was motion between the metals being joined when the solder was solidifying (not accepted).

Documentation: Information on an assembly that explains the basic design concept, type and quantity of parts and materials, special manufacturing instructions and up-to-date revisions.

Double-sided Assembly: A PCB with components mounted on both sides.

Double-sided Board: A printed board with a conductive pattern on both sides.

DPF: Disc Plot Format.

Drag Soldering: The making of soldered terminations by dipping the solder side of a supported printed board with through-hole mounted components into the surface of a static pool of molten solder and moving it horizontal in one direction.

Drag-out: A measure of cleaning solution lost via board emergence after immersion.

DRC: Design Rule Check: Checks the integrity of print (PCB) so that problems like short-circuits, unrouted nets, etc. are recognized.

Drills: Solid, carbide cutting tools designed specifically for the fast removal of material in extremely abrasive, glass-epoxy materials.

- Non-polar Compound:** Material having electrical charges distributed over the surface of the molecule, thereby showing an electrical effect in solution.
- Non-wetting:** A condition whereby a surface has contacted molten solder, but has had none of the solder adhere to it.
- NPTH:** Non plated through-hole
- Omegameter:** A test instrument measuring ionic residues on PCBs via the drop of resistivity over a specific time.
- One-sided Board:** See Single-sided Board.
- Open:** An area of a bare PCB which, due to over-etching or fabrication problems, separates two electrically connected points.
- Organic Activated (OA):** A water-soluble flux using organic acids as activators.
- OSP:** Organic Solderable Preservative.
- Outgassing:** The gaseous emission from a laminate printed board or component when the board or the printed board assembly is exposed to heat or reduced air pressure or both.
- Outgrowth:** The increase in conductor width at one side of a conductor, caused by plating build-up, over that delineated on the production master.
- Overhang:** Increase in printed circuit conductor width caused by plating build-up or by undercutting during etching.
- Packaging Density:** Quantity of functions (components, interconnection devices, mechanical devices) per unit volume, usually expressed in qualitative terms, such as high, medium, or low.
- Pad:** A portion of the conductive area of which components, terminals, traces, etc., are mechanically attached. (Also called land).
- Panel:** The base material containing one or more circuit patterns that passes successively through the production sequence and from which printed circuit boards are extracted. See Backplanes/Backpanels.
- Panel Plating:** The plating of the entire surface of a panel (including holes).
- Parylene:** A polymer resin (polyparaxylene) that provides a thin, uniform coating on PCBs and components. It can be applied via vacuum for deposition on sharp edges and complex shapes.
- Pattern Plating:** Selective plating of a conductive pattern.
- Pattern:** The configuration of all conductive and/or non-conductive areas on a PCB. Letters and inscriptions may also be included. Pattern also denotes the circuit configuration on related tools, drawing and masters. *Synonym:* image.
- PCB:** Printed Circuit Board.
- PEC:** Printed Electronic Component.
- Peel Strength:** The force per unit width required to peel the conductor or foil from the base material.

- Permanent Mask:** A resist which is not removed after processing, e.g., plating resist used in the fully-additive process.
- Photomaster:** See Artwork Master.
- Photopolymer:** A polymer that changes characteristics when exposed to light of a specific frequency.
- Photoplotter:** A plotter that writes using light.
- Physical Layer:** A conductive board layer or artwork image representing a complete conductive layer.
- Pick-and-place Machine:** A programmable machine usually with a robot arm for picking components from a feeder. It moves the part for placement and/or insertion to a specific site on the board.
- PIH Assembly:** Pin-in-hole a printed board assembly made up of components with leads which pass through holes in the board and lands. *Synonyms:* traditional assembly, conventional assembly.
- Pilot Hole:** See Locating Hole.
- Pin Density:** The quantity of pins on a printed board per unit area.
- Pin:** A terminal on a component. A component lead that is not readily formable without being damaged.
- Pinhole:** A minute hole through a layer of pattern.
- Pinholes:** Small imperfections which penetrate entirely through the conductor and/or solder.
- Pink Ring:** Chemically-induced fracturing or de-lamination on or below the surface of the base material; it is usually exhibited by a light area around holes, other etched areas or both.
- Pit:** A depression in the conductive layer that does not penetrate entirely through it.
- Pitting:** Small holes or sharp edges on the surface of a solder joint generally caused by flux entrapment, oxidation or over-heating.
- Plate Finish:** Pertaining to Laminating. The finish present on the metallic surface of metal clad base material resulting from direct contact with the laminating press plates without modification by any subsequent finishing process.
- Plated Through-hole:** A hole with the deposition of metal (usually copper) on its sides to provide electrical connections between internal or external conductive patterns.
- Plating Bar:** The temporary conductive path interconnecting areas of a printed board to be electroplated, usually located on the panel outside the borders of such a board.
- Plating Resists:** Materials which, when deposited on conductive areas, prevent the plating of the covered areas. Resists are available both as screened-on materials and as dry-film photopolymer resists.
- Plating Up:** The process consisting of the electrochemical deposition of a conductive material on the base material (surface holes, etc.) after the base material has been made conductive.

- Plating:** A uniform coating of conductive metal upon the base material of the printed circuit board.
- Plotting:** The practice of mechanically converting X-Y positional information into a visual pattern, such as artwork.
- Poise:** A centimeter-gram-second unit of viscosity equal to that of a fluid requiring a shearing force of one dyne to move from a square centimeter area with a velocity of one centimeter per second (cps).
- Polarization:** A technique of eliminating symmetry within a plane so that parts can be engaged in only one way in order to minimize the possibility of electrical and mechanical damage or malfunction.
- Polyester:** (Mylar) low melting point plastic film used for cheap flexible circuits.
- Polyimide Resins:** High temperature thermoplastics used with glass to produce printed circuit laminates from multi-layer and other circuit applications requiring high temperature performance.
- Polyimide:** (Kapton) Higher melting point plastic film used as base for flexible portions of flexi-rigid boards as well as for many flexible circuits.
- Populated PCB:** A printed board on to which all passive and active components have been assembled. *Synonyms:* printed board assembly (PBA), card, assembled board.
- Positive (Noun):** An artwork, artwork master, or production master in which the intended conductive pattern is opaque to light, and the areas intended to be free from conductive material are transparent.
- Positive-acting Resist:** A resist which is decomposed (softened) by light and which, after exposure and development, is removed from those areas which were under the transparent parts of a production master.
- Pre-heat:** The process portion of the re-flow heat curve in which the PCB is heated from ambient at a pre-set rate and prior to full liquidus at the solder joint areas.
- Prepreg:** Sheet material consisting of the base material impregnated with a synthetic resin, such as epoxy or polyimide, partially cured to the B-stage.
- Press-fit Contact:** An electrical pin contact which can be pressed into a hole in a printed board to make immediate contact.
- Primer:** A coating applied before the application of an adhesive to improve the bond.
- Printed Board Assembly:** A printed board with electrical or mechanical components, other printed boards, or a combination of these, attached to it with all manufacturing processes, soldering, coating, etc. completed.
- Printed Board:** The general term for completely processed printed circuit or printed wiring configuration. It includes single, double, and multi-layer boards, both rigid and flexible.
- Printed Circuit Assembly:** A printed circuit board to which discrete components, hardware, and other electronic devices have been attached to form a complete operating unit.

- Printed Circuit Board:** An insulating material onto which an electronic circuit has been printed or etched.
- Printed Circuit:** Circuit where the interconnections between components, terminals, sub-assemblies, etc., are made by conductive strips (traces) that have been printed or etched onto an insulating board.
- Printed Component:** A component part, such as an inductor, resistor, capacitor, or transmission line, which is formed as part of the conductive pattern of the printed board.
- Printed Contact:** A portion of a conductive pattern formed by printing, serving as one part of a contact system.
- Printed Wiring Assembly Drawing:** A document that shows the printed board (rigid or flexible), the separately manufactured components which are to be added to the board, and any other information necessary to describe the joining of these parts to perform a specific function.
- Printed Wiring Board:** See Printed Board.
- Printed Wiring Layout:** A sketch that depicts the printed wiring substrate, the physical size and location of electronic and mechanical components, and the routing of conductors that electrically interconnect components, in sufficient details to allow the preparation of documentation and artwork.
- Printed Wiring:** A conductive pattern within or bonded to the surface of a base material intended for point-to-point connection of separate components and not containing printed components.
- Probing Systems:** Equipment for making electrical contact between the bare PCB, components or assemblies and the continuity tester. Probing devices range from manual units for low volumes to computer-controlled systems.
- Process Indicator:** A detectable anomaly, other than a defect, that is reflective of material, equipment, process and/or workmanship variations.
- Production Master:** A 1 to 1 (1:1) scale pattern which is used to produce one or more printed boards (rigid or flexible) within the accuracy specified on the master drawing.
- PTH:** Plated through-holes. Plating the holes on their internal wall.
- Pull Strength:** See Bond Strength.
- PWB:** Printed Wiring Board; same as PCB.
- Radial Lead:** A lead extending out of the side of a component, rather than from the end.
- Reference Edge:** The edge of cable or conductor from which measurements are made.
- Re-flow Soldering:** Joining components to substrates by placing the parts into solder paste and then melting the paste to achieve re-flow and the interconnection.
- Re-flow Spike:** The portion of the re-flow soldering process during which the temperature of the solder is raised to a value that is sufficient to cause the solder to melt.
- Re-flowing:** The melting of an electro-deposit followed by solidification.

Register Mark: A symbol used as a reference point to maintain registration.

Registration: The alignment of a pad on one side of the printed circuit board or layers of a multi-layer board to its mating pad on the opposite side.

Relative Humidity: The ratio of the quantity of water vapour present in the air to the quantity which would saturate the air at the given temperature.

Reliability: The probability that a component, device or assembly will function properly for a defined period of time under the influence of specific environmental and operational conditions.

Repair: The process of restoring the functional capability of a defective component or circuit.

Repeatability: The ability of a system to return to a specific parameter, said of equipment when evaluating its consistency of processing.

Residue: Any visual or measurable form of process-related contamination.

Resin Smear: Resin transferred from the base material onto the surface or edge of the conductive pattern normally caused by drilling. Sometimes called epoxy smear.

Resin: A high-molecular-weight organic material with no specific melting point. A polymer.

Resist: Coating material used to mask or to protect selected areas of a pattern from the action of an etchant, solder, or plating. Also see Dry Film, Resists, Plating Resists and Solder Resists.

Reverse Image: The film pattern on a printed circuit board enabling the exposure of conductive areas for subsequent plating.

Re-work: A manufacturing step or process that is repeated to bring a non-performing or non-conforming component or circuit to a functional condition.

Re-working: The act of repeating one or more manufacturing operations for the purpose of improving the yield of acceptable part.

Rheology: Science of flow, is the study of the flow and deformation of matter and is particularly important with regard to colloidal systems.

Ribbon Cable: A flat cable with round conductors.

Right-angle Edge Connector: A connector which terminates conductors at the edge of a printed board, while bringing the terminations out at right angles to the plane of the board conductors.

Roadmap: A printed pattern of non-conductive material by which the circuitry and components are delineated on a board to aid in service and repair of the board.

Rosin Flux: The mildest of solder fluxes and generally requiring added organic activating agents.

Rosin: A hard, natural resin (nowadays also synthetic), consisting of abietic and primaric acids and their isomers, some fatty acids and terpene hydrocarbons, that is extracted from pine trees and subsequently refined.

Saponifier: An aqueous organic or inorganic base solution with additives that promote the removal of flux.

SBU: Sequential Build-up.

Schematic Diagram: A drawing that shows, by means of graphic symbols, the electrical connections, components and functions of a specific electronic circuit arrangement.

Screen Printing: A process for transferring an image to a surface by forcing suitable media through a stencil screen with a squeegee. Also called silk screening.

Screen: A network of metal or fabric standards mounted tautly on a frame and upon which the PCB's circuit pattern is superimposed by photographic means.

Semi-additive Process: A process for obtaining conductive patterns by a combination of electroless metal deposition with etching and/or electroplating. A semi-additive process is used in conjunction with a metal clad base material.

Semi-aqueous Cleaning: A technique involving the use of a solvent followed by hot-water rinses and drying.

Sensitizing: See Activating.

Shadowing: A condition occurring during etching in which the dielectric material, in intimate contact with the foil, is incompletely removed though acceptable etching may have been achieved elsewhere. In re-flow soldering, a condition in which component bodies block the infra-red energy from certain areas of the board. In wave soldering, the solder fails to wet some parts leads due to other devices blocking the flow of solder.

Shielding, Electronic: A physical barrier, usually electrically conductive, designed to reduce the interaction of electric or magnetic fields upon devices, circuits or portions of circuits.

Short: Short-circuit. An abnormal connection of relatively low resistance between two points of a circuit or conductors from different nets either touch or come closer than the minimum spacing allowed for the design rules being used.

Signal Conductor: An individual conductor used to transmit an impressed signal.

Signal Plane: A conductor layer intended to carry signals, rather than serve as a ground or other fixed voltage function.

Signal: An electrical impulse of a pre-determined voltage, current, polarity and pulse width.

Silkscreen: Often used for legend print and means the printed reference designators on a printed wiring board.

Simulation: The process of creating a virtual representation (a computer model) of an electronic component, circuit board, or system and applying virtual test signals to the model to verify its functionality and possibly its timing.

Single-sided Board: A printed board with a conductive pattern on one side only.

Single-image Production Master: A production master used in the process of making a single printed board.

Surface Leakage: The passage of current over the boundary surface of an insulator as distinguished from passage through its volume.

Surface Mounting: The electrical connection of components to the surface of a conductive pattern that does not utilize component holes.

Surface Tension: The natural, inward, molecular attraction force that inhibits the spread of a liquid at its interface with a solid material.

Surfactant: A chemical added to the cleaning solution to lower surface tension and to promote wetting.

Swaged Lead: A component lead wire that extends through a hole in a printed board and its lead extension is flattened (swaged) to secure the component to the board during manufacturing operations.

Tape-and-reel: A packaging method of housing surface-mount parts in their own tape cavities in a long continuous strip. The cavities are covered so that the tape can be wound around a reel for convenient handling and machine set-up.

Taped Components: Components attached to continuous tape for automatic assembly.

Teflon: Du Pont trade name as an inventor for PTFE (Polytetrafluoroethylene).

Temperature Profile: The depiction of the temperature that a selected point traverses as it passes through the re-flow process.

Tented Via: A via with solder mask completely covering both its pad and its plated through-hole. This completely insulates the via from foreign objects, thus protecting against accidental shorts, but it also renders the via unusable as a test point.

Tenting: A printed board fabrication method of covering over plated through-holes and the surrounding conductive pattern with a resist, usually dry film.

Terminal Area: A portion of a conductive pattern usually, but not exclusively, used for the connection and/or attachment of components.

Termination: The part of a component that makes contact with a pad on a substrate.

Terpenes: (Turpentine) A solvent used in cleaning electrical assemblies.

Test Board: A printed board suitable for determining acceptability of the board or of a batch of boards produced with the same process so as to be representative of the production board.

Test Coupon: A pattern as an integral part of the PCB on which electrical tests may be made to non-destructively evaluate process control. A portion of a circuit used exclusively to functionally test the circuit as a whole.

Test Fixture: A device that adapts a specific assembly under test to a test system via inter-connection.

Test Pattern: A pattern used for inspection or testing purposes.

Test Point: Special points of access to an electrical circuit, used for testing purposes.

Thermocouple: A device made of two dissimilar metals which, when heated, generate a voltage that is used to measure temperatures.

Thermoplastic: A plastic set into final shape by forcing the melted polymer into a cooled mould. The hardened form can be re-melted several times.

Thermoset: A plastic cured or hardened by heating into a permanent shape. Thermosets cannot be re-melted.

Through Connection: An electrical connection between conductive patterns on opposite sides of an insulating base, e.g. plated through-hole or clinched jumper wire.

Tinning: The application of molten solder to a basis metal in order to increase its solderability.

Tolling Hole: Also called fabrication hole, pilot hole, or manufacturing hole. These are used as PCB reference points upon which other dimensions are based.

Tombstoning: A soldering defect in which a chip component moves into a vertical position during solidification of the solder so that only one terminal is connected. It is caused by defective re-flow processing.

Trace: A single conductive path in a conductive pattern.

Traces: The metallic conductive strips that provide connections between components, terminals, etc., on printed circuits.

Transmission Cable: Two or more transmission lines.

Transmission Line: A signal-carrying circuit composed of conductors and dielectric material with controlled electrical characteristics used for the transmission of high-frequency or narrow-pulse type signals.

Triazine: Dielectric material with higher glass transition temperature and better thermal stability than epoxy resin, however, more expensive and not in common use.

Trim Lines: Lines which define the borders of a printed board.

Twist: The deformation of a rectangular sheet, panel or printed board, that occurs parallel to a diagonal across its surface in such a way that one of the corners of the sheet is not in the plane formed by the other three corners.

UL: Underwriter's Laboratories, Inc., a corporation for the purpose of establishing safety standards on types of equipment or components in USA and Canada.

Ultrasonic Soldering: Fluxless soldering wherein molten solder is vibrated at ultrasonic frequencies while making the joint.

Undercut: A groove or excavation at one edge of a conductor caused by etching.

Underwriters' Symbol: A logotype authorized for placement on a product which has been recognized (accepted) by Underwriters Laboratories, Inc. (UL).

Unsupported Hold: A hole containing no conductive material nor any other type of reinforcement.