

Part One

Introduction

1

Printed Circuits

1.1

Technology Presentation

Printed circuit boards (PCBs) and printed circuit board assembly (PCBA) case studies serve to illustrate the new design of experiment (DOE) methodology discussed in this book. The PCB is an essential part of the electronic circuit packaging system that interconnects the electronic components for specific tasks. The PCB provides the mechanical support and the necessary connections between the components attached.

The modern PCBs should be smaller, highly integrated, and should have faster operating speed, higher power ranges, and higher reliability.

PCBs are categorized in several ways according to

- Layer count
- Substrate
- Additive or subtractive technology
- Rigidity or flexibility.

According to layer count, the PCBs are classified into three main categories:

- Multilayer PCB
- Double-sided PCB
- Single-sided PCB.

PCBs are also categorized by substrates or base materials into three classes:

- Rigid PCB
- Flexible
- Rigid–flex.

Rigid PCBs are the most common type of PCBs especially when used to interconnect components. Flexible circuits are manufactured on polyimide and polyester substrates that remain flexible at finished thickness. They allow 3D movements. Rigid–flex boards are assembly of rigid and flexible boards laminated together during the manufacturing process.

The most common manufacturing method is based on subtractive processing in which the metal is selectively removed from a PCB, and what remains forms the conductive circuit.

Additive processing refers to a process whereby the circuit is formed by selectively plating metal on a substrate to create a circuit layer. Hybrid methods referred to as partially additive and semiadditive are essentially subtractive methods.

The main constituents of a standard PCB are the copper foil, electroplated on titanium or stainless steel, and the dielectric prepreg, consisting of resin, which may or may not be reinforced with glass fibers, woven or nonwoven, or filaments, or other inert fillers. The prepreg is manufactured by permeating woven glass fabric with a solution of epoxy resin and then passing it through a heat treatment that removes the solvent and partially cures the resin, taking it from the nonreacted stage to the “B-stage,” partially cured. Prepreg, that is, “B-stage” with different fabric weaves, different resin systems, and different resin/glass ratios, is accessible. Despite their variety, all the PCBs are basically composed of conductors, dielectrics, and vias. This generic structure determines the few basic steps that are common to most PCB fabrications: materials preparation, inner-layer processing, laminate preparation and lamination, drilling, making conductive holes, imaging, developing, electroplating, etching, solder mask (SM) application, surface finish application, and routing and testing (Coombs, 1996).

1.2 Inner-Layer Processing

In inner-layer processing stage, each layer is processed in a printed circuit structure by resist or film application, imaging, and developing, followed by copper etching and resist stripping. To ensure adhesion between the layers and the additional prepreg layers, all layers are chemically treated by oxidation, to black or red oxide, by application of thin coats of metallic base with bonding properties, such as tin complex compounds or even by the creation of specific topography by inhibited acid treatment.

1.3 Materials Preparation

The core material is sheared to panel size and then cleaned mechanically, chemically, or by combination of both.

Mechanical scrubbing methods include abrasive brush scrubbing and aluminum oxide scrubbing. Brush scrubbing removes a thin layer of surface copper but can produce a surface noncompatible with fine-line circuit design. Aluminum oxide produces a favorable surface for photoresist application.

Chemical cleaning is accomplished in spray chambers with agents such as potassium persulfates. Supplementary steps may include mild oxidizers.

1.4

Lamination

The lamination process involves two distinct yet linked operations:

- Layup
- Lamination.

The layup is referred to as “building up the book.” The material is chosen and sized taking into account the expected lifetime of the assembled board.

During the lamination process, the thin-core inner layers are subjected to heat and pressure and compressed into a laminated panel. Prepreg or B-stage sheets are slipped between the layers to bind the layers together.

In this stage, the objective is to form the layup consisting of sheets of copper foil separated by two or more plies of prepreg. This brings the resin to a new stage, sometimes designated as the “C-stage,” which corresponds to a more complete cure. The lamination is done under vacuum to remove volatiles as the B-stage cures. The resulting raw board is cleaned and sized.

Sequential lamination is a technology that takes several multilayered circuits and laminates them together to produce one or more multilayered boards. In sequential lamination technology, the panels are drilled and another lamination step takes place for the outer layers. When a design includes different types of vias, it typically requires a set of sequential lamination and electroplating cycles.

1.5

Drilling

The purpose of through-hole drilling PCB is twofold:

- To produce an opening through the board that will permit a subsequent process to form an electrical connection between top, bottom, and internal conductor pathways.
- To permit through the board component mounting with structural integrity and precision of location.

There are critical points to take into account for drilling operation: the alignment between the inner layers and between the inner and outer layers, the drill geometry correlation, speed, and material with smear formation control, the sidewall’s integrity, and the removal of residue. The multilayered board may include different types of vias, for instance,

- Through holes
- Buried via
- Blind via.

Buried via is drilled through inner layers and does not exit to either outer layer. Blind via starts at one surface layer but terminates before it penetrates all layers. The

blind via is a through hole connecting the surface to one of the internal layers. For some electrical connections, the mechanical drilling was replaced by laser ablation, allowing different sizes of blind via holes in the external layers of the board. Adequate cleaning methods such as mechanical, chemical, or plasma etching follow the drilling. A nonconventional method to make well-defined blind vias in outer layers is to apply the image on the external monomer layer, the photodefinable dielectric that is polymerized by exposure and by blind via emptied as a result of developing. Such materials can be made conductive using specific methods.

The drilling or ablation causes smearing or residues of epoxy resin on the inside of the holes. Aggressive chemicals remove these imperfections.

Hole cleaning refers to a process called desmear and to the closely related process of etchback. Desmear removes the melted resin smears that result from the friction of the drill cutting through the material. At present, the most widely used chemistry is sodium or potassium permanganate.

During etchback, glass fibers are etched in addition to removing the resin smear. Glass etchings include hydrochloric acid. Etchback with plasma can be achieved by varying the type and the amount of reactive gases.

1.6

Making the Hole Conductive

To provide the intended interconnection between layers, the holes must be coated or plated with a conductor substance. Therefore, the next stage is to render all drilled surfaces conductive. The PCB substrate is not conductive, so a nonelectrolytic deposition method is required.

The hole is made conductive using copper electroless or direct plate. The electroless technology consists in plating a thin copper layer with a controlled structure thickness and adhesion on each material of composite complex.

The main steps of electroless process are

- Cleaning the hole and surface to ensure that the copper is not deposited on the base material without a passive interface.
- Activation and acceleration by depositing a catalyst on the hole surface that allows subsequent copper deposition.
- Copper deposition, that is, reduction of the copper ion in solution causing copper metal to deposit on the hole surface.

The controlled structure and thickness will ensure the film's continuity after surface preparation and before electroplating. The direct plate allows avoiding the electroless or electroplating voids or cavities. The good adhesion will ensure the removal of one of the major defects affecting the product quality and reliability, the inner-layer separation.

In addition to the process of electroless copper deposition, a process called "fully additive" can be used to build up the conductors without electroplating processes.

1.7 Imaging

Typically, the imaging process includes three steps:

- Photoresist application
- Exposing or printing
- Developing.

Photoresists are available in the form of dry films and liquid resists.

Film photoresist is applied with heat and pressure to the surface of the panel. This is done in a laminator.

The printing of the image is usually accomplished by placing a film or glass phototool between the panel to image and a light source. Advances in resist chemistry and laser tools have made possible “direct imaging” that expose images on the photoresist directly from computer database design information and does not need a phototool, an important option for special applications.

The selective copper plating allowing the buildup of the circuit is possible only by exposing the conductive surfaces according to predetermined design. The rest of the base copper foil is used only temporarily to conduct the electrical current for electroplating and is for that reason protected. If there are several options for inner layers, the image for outer layers is defined by means of a dry film.

For good adhesion, the copper base, electroless copper, or electrolytic copper for panel plate technique should be textured to an appropriate roughness by using a mechanical wet abrasion product, such as aluminum oxide, or pumice, together with an acid cleaning. The film lamination depends on the temperature and on the pressure of rollers, and advances according to both chemical composition of the monomer film and its thickness. The exposure step, the polymerization of the film on nonuseful base copper surfaces, is a difficult operation due to the degree of precision imposed. Misalignment may have a significant impact on the PCB quality. The cleanliness of the environment and a perfect contact between the board and the pattern image, ensuring the perfect vacuum, are critical. The accuracy of the energy application defines the resist wall quality.

A distinctive imaging method for ultrafine lines is based on the electrodeposited photoresist. Conductive surfaces are completely covered with resist by submerging them in an aqueous, micellar dispersion of the resist. The panel is charged to either a negative or a positive potential and attracts the polymer micelles in the resist bath. Lines and spacing at 1 mil (25 μm) have been successfully produced with this technology.

To remove nonreacting monomers and display the base copper, that is, the raw circuit, the exposed panel is passed through a developer. A critical point here is the removal of developer traces from the copper. These are nonconductive and may promote missing plating, step plating, plating voids, separations, and other defects.

1.8

Electroplating

After surface preparation for residual dirty, dry film chips and oxidation removal, the boards are electroplated with copper. The electroplating process produces the plated through holes (PTHs). This step is significant for reliability. The average copper thickness is correlated to specific field applications.

Two approaches can be used to create the final circuit pattern:

- *Panel plating:* In this case, the entire panel is plated to its full thickness before a plating resist is deposited and the etch-resist metal is plated on the board. The subsequent step is to etch away all unwanted copper.
- *Pattern plating:* In this approach, the plating resist is applied before the copper plating and only the final conductor pattern is plated. The subsequent step is to etch away the remaining copper.

There are different copper plating additive systems, all with the possibility to use a direct current or a reverse-pulse current (Dini, 1993). The chemistry of the brightener, of the basic electrolyte recipe, and of the applied parameters allows the control of the copper microstructure, thickness, uniformity, adhesion, and physical properties and of the worst of the defects having this source. Examples of defects are pitting, thickness uniformity, dendrite or macrocrystalline structure, plating cracks, structure modification in roughness form, and so on.

The copper plating may be used alone or as an underlayer for further nickel plating or nickel–gold plating. The nickel–hard gold plating, on a flash of soft gold, is used in applications such as

- Electrical mobile contacts
- Corrosion resistance
- Oxidation resistance in hot and humid environment.

Usual plating is “full gold” or “full-body gold” with thin soft gold, 3–12 μm . (1 μm = 40 $\mu\text{in.}$), continued with selective hard gold plating, up to 30–50 μm . To plate selectively, a gold mask is applied. This is a dry film similar to the film used for plate etching. Otherwise, it is possible to use a thin layer approximately 100 μm . (2.5 μm) of tin or tin–lead on top of the copper to function as etch resist. Significant in this case is to have a continuous layer of a crystalline structure capable of protecting the circuit during the base copper etching.

To protect the conductive circuit pattern during the process that removes all unwanted copper, in the subtractive type of PCB fabrication, a metal that will not react with the etching agent is plated to it. This, typically, is one of the following:

- Tin–lead, usually a fluoborate solution
- Tin, usually fluoborate or organic sulfonic acid solution
- Nickel–gold using a nickel predip, a nickel bath, gold predip, and a gold plating bath
- Nickel.

Of these, tin–lead is the most popular. However, there is an ecologically rooted movement to eliminate lead from electronic equipment and this has an impact on the development of alternatives to tin–lead solder.

1.9 Copper Etching

The removal of nonuseful base copper starts with film stripping.

Panels entering the etch process have been coated with etch resist, usually a dry film resist. The resist layer selectively protects the circuit area from etch, whereas the remaining copper foil is etched away. The etching agents sprayed onto the surface of the panel remove the exposed copper but cannot dissolve the copper residing under resist. Acid cupric chloride and alkaline ammoniac are the most common etching agents.

The board is passed through a controlled etch rate solution. The chemistry and parameters of this solution are significant to avoid extreme situations, such as underetching, shorts or overetching, reduced lines, and opens. After that, the circuit is cut out, and it is possible to remove the temporary tin protection through a chemical dissolution process, the tin strip in an inhibited acid solution.

1.10 Solder Masking

The solder mask is applied to protect the PCB before hot air solder leveling (HASL). Only a part of the lines and pads have to be soldered during the assembly process. The part of the circuit not utilized for assembly is covered with solder mask. The choice of solder mask, epoxy and half-epoxy or fully aqueous, and its mode of application, screening and spray, depend on the use of the product and also on the possibility of further chemical operation application, that is, electroless nickel immersion gold (ENIG) nickel–hard gold, or other final finishes, where thin layers may be required. The types of circuits, fine lines, flip-chip pads, small dams, and so forth will play an important role for solder mask choice through the imposed undercut and mask definition. Surface preparation before solder mask is important to ensure the good adhesion of the mask by creating an adequate topography. The usual way to do this is by scrubber running a wet abrasive powder or by chemically controlled etching. In all cases, solder mask processing is similar to film processing. This means that there are the same steps of application, exposure, and developing. Due to different chemical composition, the chemistry for developing varies. Basically, solvents or diluted carbonates are used. The exposure of solder mask is critical, and as for all photoimage processes, the alignment is determinant. Final curing of the mask is a key step. In this case, it is necessary to take into account other thermal excursions that will additionally bake the mask, such as nomenclature, carbon ink, or compatibility with further chemical processes. For ENIG, the mask should be undercured for better chemical resistance. Overcured masks are brittle.

1.11

Surface Finishing

Most frequently, the exposed circuit will be soldered. To build or to preserve solderability, the boards will pass through a final finishing step. The HASL, or oil-fused plated solder, has been used for a long time to ensure solderability. As the board design becomes increasingly sophisticated, the solder was found insufficiently efficient as a finish. Other surface finishes appeared, for example, organic solderability protection (OSP), ENIG, immersion tin, immersion silver, electroless or immersion palladium, immersion bismuth, and so on to replace HASL.

Electrolytic plating substitutes for HASL include rhodium, palladium, palladium–nickel alloys, and ruthenium.

To ensure finish application and the reliability of the product, surface cleanliness after the final finish is required.

1.12

Routing

To facilitate the use of boards in assembly, they are routed alone or in groups that are in panel forms, depending on their size. Burr or smear is unacceptable. The boards resulting from this process are of their final size. A postcleaning operation usually with pressurized warm water is used to remove all dust or debris resulting from routing.

There are many other specific steps depending on user demand. Examples are the nomenclature, back print, plug via hole with solder mask, temporary masking, carbon conductive ink applied by screening followed by baking, and so on.

1.13

Testing and Inspection

The final step in board fabrication is to verify the integrity and the functionality of the board. After routing, the PCB is submitted to final control and reliability and quality tests. Control and tests are also interoperational.

The tools for parameter PCB product analysis are the test coupons. These are part of the quality assessment processes that cover reliability evaluations, end-product evaluations, work-in-process evaluations, and process parameter evaluations. There are coupons to test every factor, but the challenge is to test factor interactions in real time.

The main classes of tests are as follows:

- Optical (visual, automatic optical inspection, microscopy under cross-sectioning, scanning electronic microscopy (SEM))
- Electrical (continuity, dielectric withstanding, impedance, and arc resistance)

- Mechanical (tensile, ductility, peel strength, adhesion, bond strength, flexural and impact strength, and dimensional stability)
- Solderability tests (wetting balance, sequential electrochemical reduction analysis (SERA), and dip and look)
- Thermomechanical (glass transition temperature T_g , coefficients of thermal expansion (CTE), Z-expansion, and delamination time)
- Chemical (resistance to chemical, flammability)
- Reliability (thermal shocks, fatigue test, and accelerated stress tests)
- Long-term reliability (surface insulation resistance tests, aging tests, corrosion tests, radiation tests, water absorption tests, and biological tests).

The electrical tests are important since PCBs are electric or electronic devices.

The main types of electrical tests and faults are listed in Table 1.1.

The possible mixture of elements to be tested is combinatorial, and so a clear understanding of exactly what is tested and how much is critical to avoid quality and reliability problems during assembly or operation.

1.14

Assembling

The PCB is part of a total electronic circuit packaging system, PCBA. Various types of configurations of components are attached to PCB in assembly. The PCB manufacturer should deliver the product that encounters the assembly requirements.

Performance in automatic assembly is a result of a blend of many ingredients:

- Methodology, that is, board, material, size, and requirements
- Designing the product for automation
- Standards.

Figure 1.1 shows an example of PCB manufacturing process. It contains the main steps in PCB manufacturing in an achievable technology. There are several products resulting from the same flow chart and also several flow charts associated with the same final product.

In principle, the PCB fabrication is analogous to integrated circuit fabrication, to thin layer, nanocircuit, or molecular device fabrication. All these fabrications of

Table 1.1 Electrical tests and fault types.

Test type	Failure type
Continuity	Open
Isolation	Short/leakage
TDR	RF impedance fault
Hi-Pot	Voltage breakdown

TDR, time domain reflectometry; RF, radio frequency.

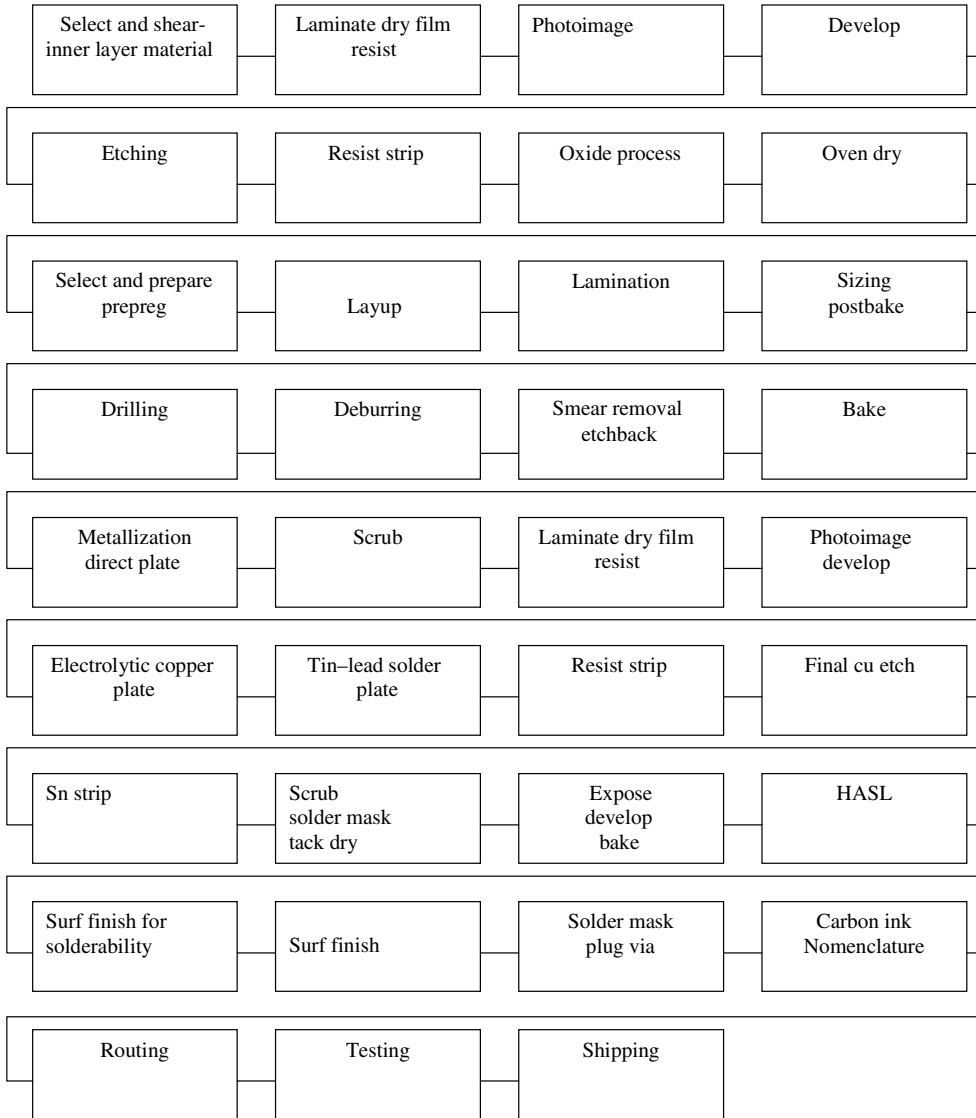


Figure 1.1 PCB manufacturing.

circuits are successions of reversible or irreversible operations of additions and subtractions of new materials joined by masking steps. These operations allow obtaining the designed circuitry.

To outline the similarity in PCB and other circuit fabrication, let us consider in more detail the sequence of manufacturing steps for complementary metal oxide semiconductor (CMOS) process. The starting layup is silicon wafer covered by a layer of oxide and a layer of silicon nitride. On the silicon nitride surface, the photoresist is

applied. After masking, the layout is exposed to UV light and to developing. Then the silicon nitride is etched on the nonmasked areas and the photoresist is stripped. The circuitry made by silicon nitride is now visible. For specific areas, a new photoresist is applied. The boron or other atoms are embedded on the nonmasked areas. The photoresist is stripped again. The oxide growing stage is finally performed.

The PCB manufacture starts from a basic layout containing conductive and dielectric layers. This is the result of the former additive process, lamination.

Additive processes include lamination of layers, electroplating, chemical plating, sputtering, screening, implanting, immersion, deposition, adhesion, and so forth. Etching, stripping, dissolution, ablation by laser or plasma, laser drilling, routing, and so on accomplish subtraction processes of existing material. To restrict or enforce the processes of addition or subtraction, to make them selective, intermediate masking steps are required. The available masks are broadly divided into two main categories, permanent and temporary. We should also start considering masks with specified lifetimes. Typical masking in PCB industry is based on resist application followed by exposure and development. Optical light, UV, ion beams, and laser could do the imaging. Tin or tin-lead has the role of masking the copper. Plugs via hole with solder mask and peelable masks accomplish temporary masking. Tapes and floating shields are used in plating processes to mask, completely or partially, specific areas. It is remarkable how many PCB or other circuit constructions are based on a dozen of elementary processes of subtractive or additive type, coupled by well-chosen masking steps.

