34 Electronics Assembly and Packaging

Chapter Contents

Integrated circuits constitute the brains of an electronic system, but the complete system consists of much more than packaged ICs. The ICs and other components are mounted and interconnected on printed circuit boards, which in turn are interconnected and contained in a chassis or cabinet. Chip packaging (Section 33.6) is only part of the total electronic package. This chapter considers the remaining levels of packaging and how the packages are manufactured and assembled.

34.1 Electronics Packaging

The electronics package is the physical means by which the components in a system are electrically interconnected and interfaced to external devices; it includes the mechanical structure that holds and protects the circuitry. A well-designed electronics package serves the following functions: (1) power distribution and signal interconnection, (2) structural support, (3) circuit protection from physical and chemical hazards in the environment, (4) dissipation of heat generated by the circuits, and (5) minimum delays in signal transmission within the system.

For complex systems containing many components and interconnections, the electronics package is organized into levels that comprise a ***packaging hierarchy***, illustrated in Figure 34.1 and summarized in Table 34.1. The lowest level is the ***zero level***, which refers to the intraconnections on the semiconductor chip. The packaged chip (chip carrier), consisting of the IC in a plastic or ceramic enclosure and connected to the package leads, constitutes the ***first level of packaging***.

Packaged chips and other components are assembled to a printed circuit board (PCB) using two technologies (Section 33.6.1): (1) ***surface-mount technology*** (SMT)

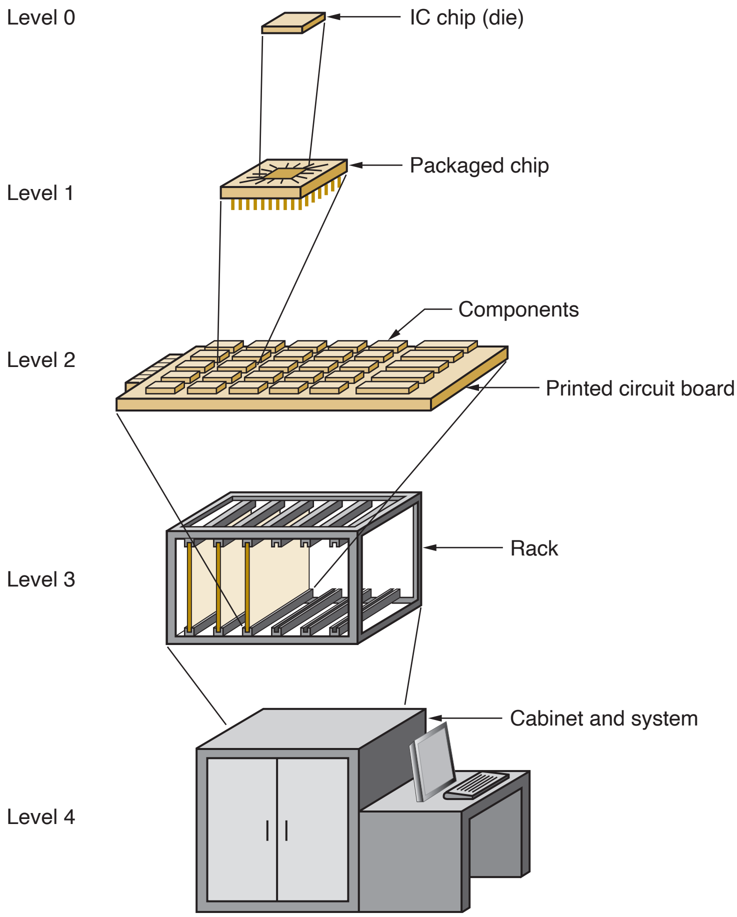

FIGURE 34.1
Packaging hierarchy in a large electronic system.

and (2) ***through-hole technology***, also known as ***pin-in-hole*** (PIH) technology. Today, SMT is much more widely used in industry, especially for mass-produced products. The chip package styles and assembly techniques are different for SMT and PIH. In many cases, both assembly technologies are employed in the same board. Printed circuit board assembly represents the ***second level of packaging***. Figure 34.2 shows two PCB assemblies, illustrating both PIH and SMT types. Note the smaller component sizes of the SMT board.

TABLE • 34.1 Packaging hierarchy.

Level	Description of Interconnection
0	Intraconnections on the chip
1	Chip-to-package interconnections to form IC package
2	IC package to circuit board interconnections
3	Circuit board to rack; card-on-board packaging
4	Wiring and cabling connections in cabinet

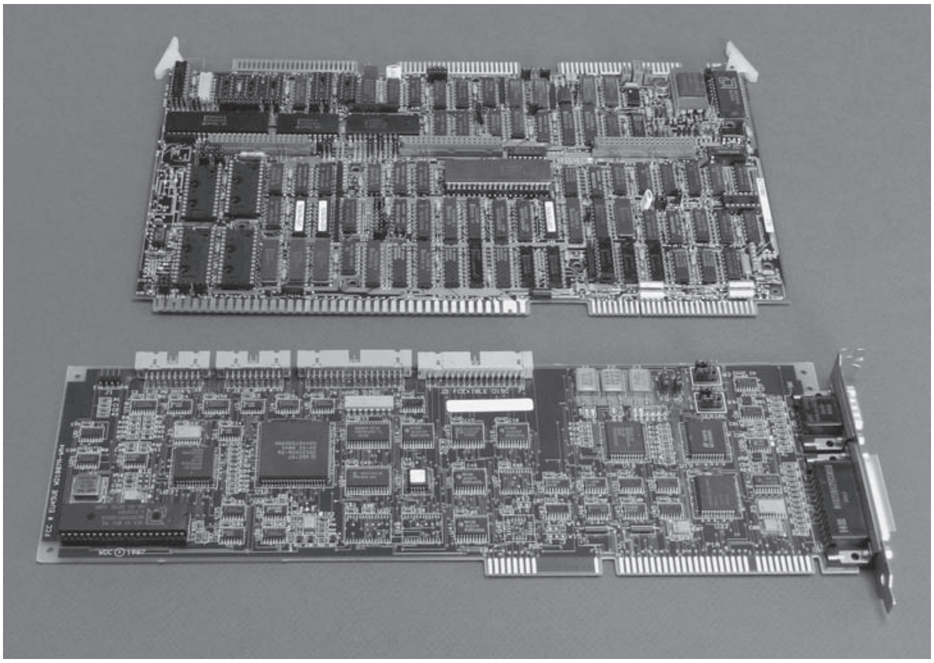

FIGURE 34.2 Two printed circuit board assemblies. Upper PCB assembly is through-hole technology (pin-in-hole) whereas lower assembly is mostly surface-mount technology. (Courtesy of George E. Kane Manufacturing Technology Laboratory, Lehigh University.)

The assembled PCBs are, in turn, connected to a chassis or other framework; this is the ***third level of packaging***. This third level may consist of a ***rack*** that holds the boards, using wire cables to make the interconnections. In major electronic systems, such as large computers, the PCBs are typically mounted onto a larger printed circuit board called a ***back plane***, which has conduction paths to permit interconnection between the smaller boards attached to it. This latter configuration is known as ***card-on-board*** (COB) packaging, in which the smaller PCBs are called cards and the back plane is the board.

The ***fourth level of packaging*** consists of wiring and cabling inside the cabinet that contains the electronic system. For systems of relatively low complexity, the packaging may not include all of the possible levels in the hierarchy.

34.2 Printed Circuit Boards

A printed circuit board consists of one or more thin sheets of insulating material, with thin copper lines on one or both surfaces that interconnect the components attached to the board. In boards consisting of more than one layer, copper conducting paths are interleaved between the layers. PCBs are used in packaged electronic systems to hold components, provide electrical interconnections among them, and make connections to external circuits. They have become standard building blocks in virtually all electronic systems that contain packaged ICs and other components (Historical Note 34.1). PCBs are so important and widely used because (1) they provide a convenient structural platform for the components; (2) a board with correctly routed interconnections can be mass produced consistently, without the variability usually associated with hand wiring; (3) nearly all of the soldering connections between components and the PCB can be accomplished in a one-step mechanized

Historical Note 34.1 *Printed circuit boards*

Before printed circuit boards, electrical and electronic components were manually fastened to a sheet-metal chassis (usually aluminum) and then hand wired and soldered to form the desired circuit. This was called point-to-point construction. In the late 1950s, various plastic boards became commercially available. These boards, which provided electrical insulation, gradually replaced the aluminum chassis. The first plastics were phenolics, followed by glass-fiber-reinforced epoxies. The boards came with predrilled holes spaced at standard intervals in both directions. This motivated the use of electronic components that matched these hole spacings. The dual-in-line package evolved during this period.

The components in these circuit boards were hand-wired, which proved increasingly difficult and prone to human error as component densities increased and circuits became more complex. The printed circuit board, with etched copper foil on its surface to form the wiring interconnections, was developed in the 1950s to solve these problems with manual wiring. The printed circuit boards during that period used through-hole technology, and PCBs had to be drilled to accommodate the component leads. The original technique for "printing" the circuit pattern onto the copper-clad board was screen printing. In the late 1980s, surface mount technology began to replace through-hole construction. SMT did not require lead holes, and the components were significantly smaller, allowing much higher circuit densities to be achieved on the printed circuit boards. As track widths became finer and finer, photolithography was substituted for screen printing.

Initial techniques to design the circuit masks involved a manual inking procedure, in which the designer attempted to route the conducting tracks to provide the required connections and avoid short circuits on a large sheet of paper or vellum. This became more difficult as the number of components on the board increased and the conducting lines interconnecting the components became finer. Computer programs were developed to aid the designer in solving the routing problem. However, in many cases, it was impossible to find a solution with no intersecting tracks (short circuits). To solve the problem, jumper wires were hand-soldered to the board to make these connections. As the number of jumper wires increased, the problem of human error again appeared. Multilayer printed circuit boards were introduced to deal with this routing issue.

operation, (4) an assembled PCB gives reliable performance; and (5) in complex electronic systems, each assembled PCB can be detached from the system for service, repair, or replacement.

34.2.1 STRUCTURES, TYPES, AND MATERIALS FOR PCBs

A ***printed circuit board*** (PCB), also called a ***printed wiring board*** (PWB), is a laminated flat panel of insulating material designed to provide electrical interconnections between electronic components attached to it. Interconnections are made by thin conducting paths on the surface of the board or in alternating layers sandwiched between layers of insulating material. The conducting paths are made of copper and are called ***tracks***. Other copper areas, called ***lands***, are also available on the board surface for attaching and electrically connecting components.

Insulation materials in PCBs are usually polymer composites reinforced with glass fabrics or paper. Polymers include epoxy (most widely used), phenolic, and polyimide. E-glass is the usual fiber in glass-reinforcing fabrics, especially in epoxy PCBs; cotton paper is a common reinforcing layer for phenolic boards. The usual thickness of the substrate layer is 0.8 to 3.2 mm (0.031–0.125 in), and copper foil thickness is around 0.04 mm (0.0015 in). The materials forming the PCB structure must be electrically insulating, strong and rigid, resistant to warpage, dimensionally

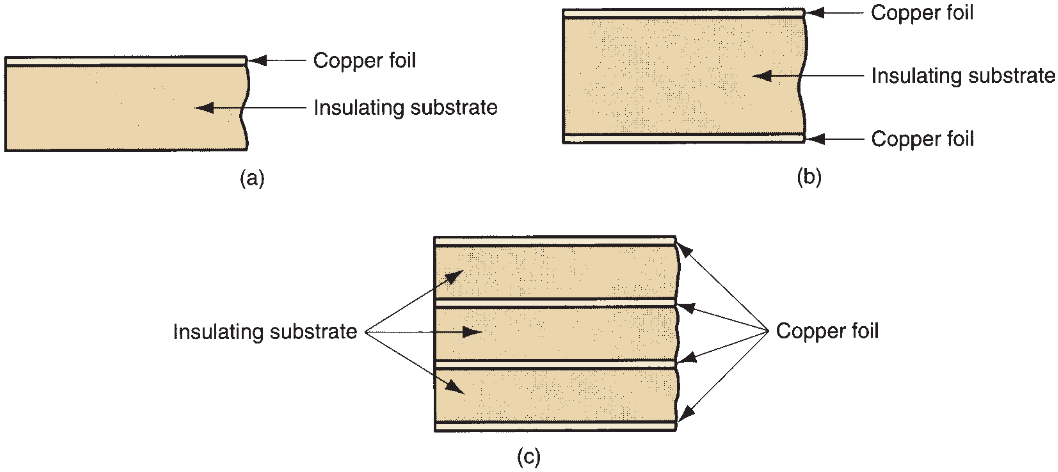

FIGURE 34.3 Three types of printed circuit board structure: (a) single-sided, (b) double-sided, and (c) multilayer.

stable, heat resistant, and flame retardant. Chemicals are often added to the polymer composite to obtain the last two characteristics.

There are three principal types of printed circuit board, shown in Figure 34.3: (a) ***single-sided*** board, in which copper foil is only on one side of the insulation substrate; (b) ***double-sided*** board, in which the copper foil is on both sides of the substrate; and (c) ***multilayer*** board, consisting of alternating layers of conducting foil and insulation. In all three structures, the insulation layers are constructed of multiple laminates of epoxy-glass sheets (or other composite) bonded together to form a strong and rigid structure. Multilayer boards are used for complex circuit assemblies in which a large number of components must be interconnected with many track routings, thus requiring more conducting paths than can be accommodated in one or two copper layers. Four layers is the most common multilayer configuration, but boards with up to 24 conducting layers are produced.

34.2.2 PRODUCTION OF THE STARTING BOARDS

Single- and double-sided boards can be purchased from suppliers that specialize in mass producing them in standard sizes. The boards are then custom-processed by a circuit fabricator to create the specified circuit pattern and board size for a given application. Multilayer boards are fabricated from standard single- and double-sided boards. The circuit fabricator processes the boards separately to form the required circuit pattern for each layer in the final structure, and then the individual boards are bonded together with additional layers of epoxy-fabric. Processing of multilayer boards is more involved and more expensive than the other types; the reason for using them is that they provide better performance for large systems than using a much greater number of lower-density boards of simpler construction.

The copper foil used to clad the starting boards is produced by a continuous electroforming process (Section 27.3.2), in which a rotating smooth metal drum is partially submersed in an electrolytic bath containing copper ions. The drum is the cathode in the circuit, causing the copper to plate onto its surface. As the drum rotates out of the bath, the thin copper foil is peeled from its surface. The process is ideal for producing the very thin copper foil needed for PCBs.

Production of the starting boards consists of pressing multiple sheets of woven glass fiber that have been impregnated with partially cured epoxy (or other thermosetting polymer). The number of sheets used in the starting sandwich determines the thickness of the final board. Copper foil is placed on one or both sides of the epoxy-glass laminated stack, depending on whether single- or double-sided boards are to be made. For single-sided boards, a thin release film is used on one side in place of the copper foil to prevent sticking of the epoxy in the press. Pressing is accomplished between two steam-heated platens of a hydraulic press. The combination of heat and pressure compacts and cures the epoxy-glass layers to bond and harden the laminates into a single-piece board. The board is then cooled and trimmed to remove excess epoxy that has been squeezed out around the edges.

The completed board consists of a glass-fabric-reinforced epoxy panel, clad with copper over its surface area on one or both sides. It is now ready for the circuit fabricator. Panels are usually produced in large standard widths designed to match the board handling systems on wave-soldering equipment, automatic insertion machines, and other PCB processing and assembly facilities. If the electronic design calls for a smaller size, several units can be processed together on the same larger board and then separated later.

34.2.3 PROCESSES USED IN PCB FABRICATION

The circuit fabricator employs a variety of processing operations to produce a finished PCB, ready for assembly of components. The operations include cleaning, shearing, hole drilling or punching, pattern imaging, etching, and electroless and electrolytic plating. Most of these processes have been discussed in earlier chapters. This section emphasizes the details that are relevant to PCB fabrication. The discussion proceeds approximately in the order in which the processes are performed on a board. However, there are differences in processing sequence between different board types, and these differences are examined in Section 34.2.4. Some of the operations in PCB fabrication must be performed under clean room conditions to avoid defects in the printed circuits, especially for boards with fine tracks and details.

Board Preparation Initial preparation of the board consists of shearing, hole-making, and other shaping operations to create tabs, slots, and similar features in the board. If necessary, the starting panel may have to be sheared to size for compatibility with the circuit fabricator's equipment. The holes, called tooling holes, are made by drilling or punching and are used for positioning the board during subsequent processing. The sequence of fabrication steps requires close alignment from one process to the next, and these holes are used with locating pins at each operation to achieve accurate registration. Three tooling holes per board are usually sufficient for this purpose; hole size is about 3.2 mm (0.125 in), larger than the circuit holes to be drilled later.

The board is typically bar coded for identification purposes in this preparation phase. Finally, a cleaning process is used to remove dirt and grease from the board surface. Although cleanliness requirements are not as stringent as in IC fabrication, small particles of dirt and dust can cause defects in the circuit pattern of a printed circuit board; and surface films of grease can inhibit etching and other chemical processes. Cleanliness is essential for producing reliable PCBs.

Hole Drilling In addition to tooling holes, functional circuit holes are required in PCBs as (1) ***insertion holes*** for inserting leads of components that are based on through-hole technology, (2) ***via holes***, which are later copper-plated and used as conducting paths from one side of the board to the other, and (3) holes to fasten certain components such as heat sinks and connectors to the board. These holes are either drilled or punched, using the tooling holes for location. Cleaner holes can be produced by drilling, so most holes in PCB fabrication are drilled. A stack of three or four panels may be drilled in the same operation, using a computer numerically controlled (CNC) drill press that receives its programming instructions from the design database. For high-production jobs, multiple-spindle drills are sometimes used, permitting all of the holes in the board to be drilled in one feed motion.

Standard twist drills (Section 22.3.2) are used to drill the holes, but the application makes a number of unusual demands on the drill and drilling equipment. Perhaps the biggest single problem is the small hole size in printed circuit boards; drill diameter is generally less than 1.27 mm (0.050 in), but some high-density boards require hole sizes of 0.15 mm (0.006 in) or even less [8]. Such small drill bits lack strength, and their capacity to dissipate heat is low.

Another difficulty is the unique work material. The drill bit must first pass through a thin copper foil and then proceed through an abrasive epoxy-glass composite. Different drills would normally be specified for these materials, but a single drill must suffice in PCB drilling. Cemented carbide or coated carbide are favored over high speed steel as the tool material.

The small hole size, combined with the stacking of several boards or multilayer boards, result in a high depth-to-diameter ratio, aggravating the problem of chip extraction from the hole. Other requirements placed on the operation include high accuracy in hole location, smooth hole walls, and absence of burrs on the holes. Burrs are usually formed when the drill enters or exits a hole; thin sheets of material are often placed on top of and beneath the stack of boards to inhibit burr formation on the boards themselves.

Finally, any cutting tool must be used at a certain cutting speed to operate at best efficiency. For a drill bit, cutting speed is measured at the diameter. For very small drill sizes, this means extremely high rotational speeds—up to 100,000 rev/min in some cases. Special spindle bearings and motors are required to achieve these speeds.

Circuit Pattern Imaging and Etching There are two basic methods by which the circuit pattern is transferred to the copper surface of the board: screen printing and photolithography. Both methods involve the use of a resist coating on the board surface that determines where etching of the copper will occur to create the tracks and lands of the circuit.

Screen printing was the first method used for PCBs. It is indeed a printing technique, and the term *printed circuit board* can be traced to this method. In ***screen printing***, a stencil screen containing the circuit pattern is placed on the board, and liquid resist is squeezed through the mesh to the surface beneath. This method is simple and inexpensive, but its resolution is limited. It is normally used only for applications in which track widths are greater than about 0.25 mm (0.010 in).

The second method of transferring the circuit pattern is ***photolithography***, in which a light-sensitive resist material is exposed through a mask to transfer the circuit pattern. The procedure is similar to the corresponding process in IC fabrication (Section 33.3.1); some of the details in PCB processing will be described here.

Photoresists used by circuit fabricators are available in two forms: liquid or dry film. Liquid resists can be applied by roller or spraying. Dry film resists are more commonly used in PCB fabrication. They consist of three layers: a film of photosensitive polymer sandwiched between a polyester support sheet on one side and a removable plastic cover sheet on the other side. The cover sheet prevents the photosensitive material from sticking during storage and handling. Although more expensive than liquid resists, dry film resists form coatings of uniform thickness, and their processing in photolithography is simpler. To apply, the cover sheet is removed, and the resist film is placed on the copper surface to which it readily adheres. Hot rollers are used to press and smooth the resist onto the surface.

Alignment of the masks relative to the board relies on the use of registration holes that are aligned with the tooling holes on the board. Contact printing is used to expose the resist beneath the mask. The resist is then developed, which involves removal of the unexposed regions of the negative resist from the surface.

After resist development, certain areas of the copper surface remain covered by resist while other areas are now unprotected. The covered areas correspond to circuit tracks and lands, while uncovered areas correspond to open regions between. ***Etching*** removes the copper cladding in the unprotected regions from the board surface, usually by means of a chemical etchant. Etching is the step that transforms the solid copper film into the interconnections for an electrical circuit.

Etching is done in an etching chamber in which the etchant is sprayed onto the surface of the board that is now partially coated with resist. Various etchants are used to remove copper, including ammonium persulfate ($(NH_4)_2S_2O_4$), ammonium hydroxide (NH_4OH), cupric chloride ($CuCl_2$), and ferric chloride ($FeCl_3$). Each has its relative advantages and disadvantages. Process parameters (e.g., temperature, etchant concentration, and duration) must be closely controlled to avoid over- or under-etching, as in IC fabrication. After etching, the board must be rinsed and the remaining resist chemically stripped from the surface.

Plating In printed circuit boards, plating is needed on the hole surfaces to provide conductive paths from one side to the other in double-sided boards, or between layers in multilayer boards. Two types of plating process are used in PCB fabrication: electroplating and electroless plating (Section 27.3). Electroplating has a higher deposition rate than electroless plating but requires that the coated surface be metallic (conductive). Electroless plating is slower but does not require a conductive surface.

After drilling of the via holes and insertion holes, the walls of the holes consist of epoxy-glass insulation material, which is nonconductive. Accordingly, electroless plating must be used initially to provide a thin coating of copper on the hole walls. Once the thin film of copper has been applied, electrolytic plating is then used to increase coating thickness on the hole surfaces to between 0.025 and 0.05 mm (0.001 and 0.002 in).

Gold is another metal sometimes plated onto printed circuit boards. It is used as a very thin coating on PCB edge connectors to provide superior electrical contact. Coating thickness is only about 2.5 μm (0.0001 in).

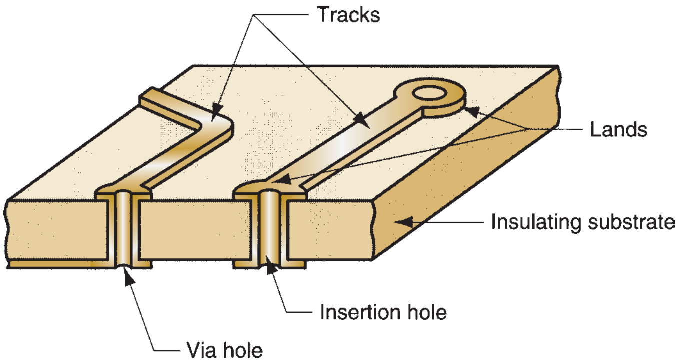

FIGURE 34.4 A section of a double-sided PCB, showing various features accomplished during fabrication: tracks and lands, and copper-plated insertion and via holes.

34.2.4 PCB FABRICATION SEQUENCE

This section describes the processing sequence for various board types. The sequence is concerned with transforming a copper-clad board of reinforced polymer into a printed circuit board, a procedure called ***circuitization***. The desired result is illustrated in Figure 34.4 for a double-sided board.

Circuitization Three methods of circuitization can be used to determine which regions of the board will be coated with copper [13]: (1) subtractive, (2) additive, and (3) semiadditive.

In the ***subtractive method***, open portions of the copper cladding on the starting board are etched away from the surface, so that the tracks and lands of the desired circuit remain. The process is called "subtractive" because copper is removed from the board surface. The steps in the subtractive method are described in Figure 34.5.

The ***additive method*** starts with a board surface that is not copper clad, such as the uncoated surface of a single-sided board. However, the uncoated surface is treated with a chemical, called a ***buttercoat***, which acts as the catalyst for electroless plating. The steps in the method are outlined in Figure 34.6.

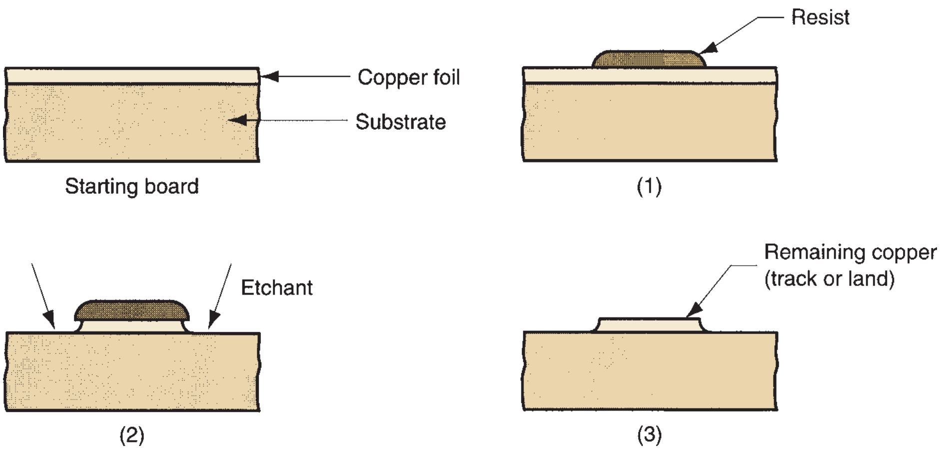

FIGURE 34.5 The subtractive method of circuitization in PCB fabrication: (1) apply resist to areas not to be etched, using photolithography to expose the areas that are to be etched, (2) etch, and (3) strip resist.

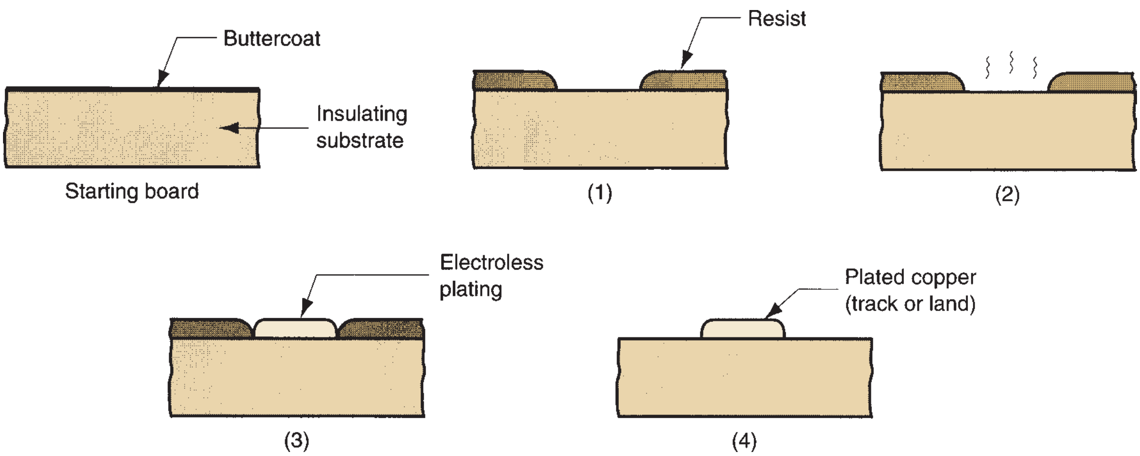

FIGURE 34.6 The additive method of circuitization in PCB fabrication: (1) a resist film is applied to the surface using photolithography to expose the areas to be copper plated; (2) the exposed surface is chemically activated to serve as a catalyst for electroless plating; (3) copper is plated on exposed areas; and (4) resist is stripped.

The ***semiadditive method*** uses a combination of additive and subtractive steps. The starting board has a very thin copper film on its surface—5 μm (0.0002 in) or less. The method proceeds as described in Figure 34.7.

Processing of Different Board Types Processing methods differ for the three PCB types: single-sided, double-sided, and multilayer. A ***single-sided board*** begins

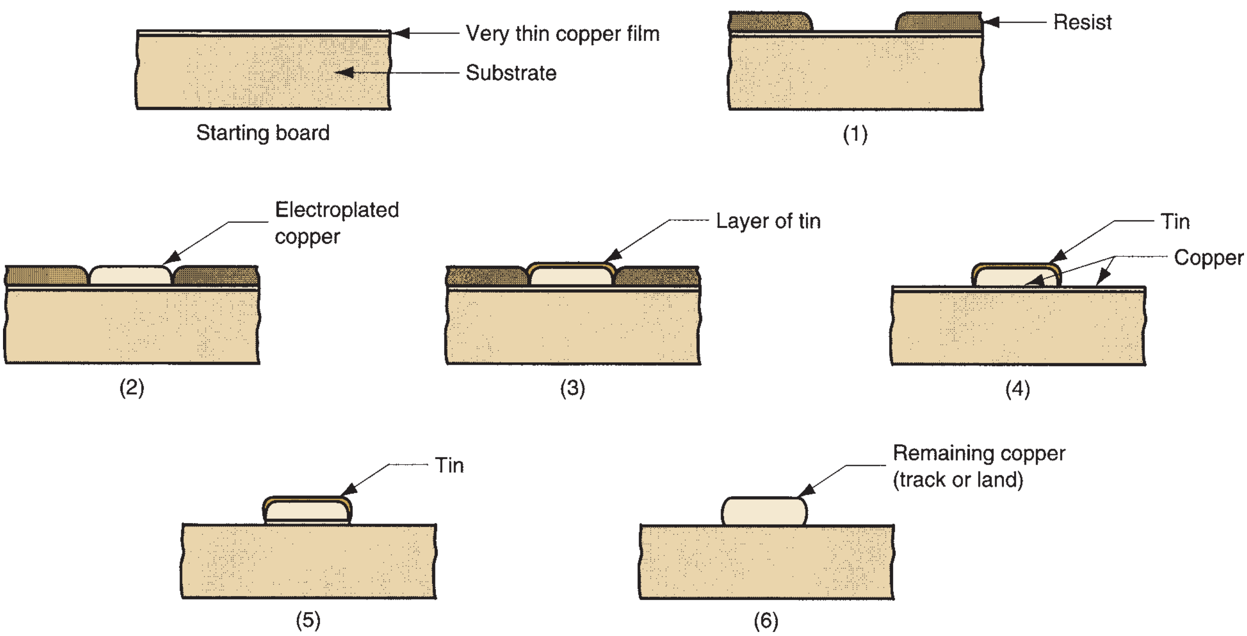

FIGURE 34.7 The semiadditive method of circuitization in PCB fabrication: (1) apply resist to areas that will not be plated; (2) electroplate copper, using the thin copper film for conduction; (3) apply tin on top of plated copper; (4) strip resist; (5) etch remaining thin film of copper on the surface, while the tin serves as a resist for the electroplated copper; and (6) strip tin from copper.

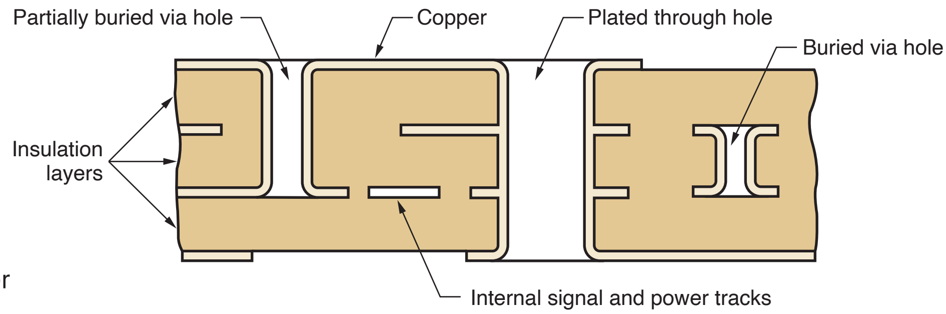

FIGURE 34.8 Typical cross section of a multilayer printed circuit board.

fabrication as a flat sheet of insulating material clad on one side with copper film. The subtractive method is used to produce the circuit pattern in the copper cladding.

A ***double-sided board*** involves a somewhat more complex processing sequence because it has circuit tracks on both sides that must be electrically connected. The interconnection is accomplished by means of copper-plated via holes that run from lands on one surface of the board to lands on the opposite surface, as shown in Figure 34.4. A typical fabrication sequence for a double-sided board (copper-clad on both sides) uses the semiadditive method. After hole drilling, electroless plating is used to initially plate the holes, followed by electroplating to increase plating thickness.

A ***multilayer board*** is structurally the most complex of the three types, and this complexity is reflected in its manufacturing sequence. The laminated construction can be seen in Figure 34.8, which highlights some of the features of a multilayer PCB. The fabrication steps for the individual layers are basically the same as those used for single- and double-sided boards. What makes multilayer board fabrication more complicated is that (1) all of the layers, each with its own circuit design, must first be processed; then (2) the layers must be joined together to form one integral board; and finally (3) the assembled board must itself be put through its own processing sequence.

A multilayer board consists of ***logic layers***, which carry electrical signals between components on the board, and ***voltage layers***, which are used to distribute power. Logic layers are generally fabricated from double-sided boards, whereas voltage layers are usually made from single-sided boards. Thinner insulating substrates are used for multilayer boards than for their standalone single- and double-sided counterparts, so that a suitable thickness of the final board can be achieved.

In the second stage, the individual layers are assembled together. The procedure starts with copper foil on the bottom outside, and then adds the individual layers, separating one from the next by one or more sheets of glass fabric impregnated with partially cured epoxy. After all layers have been sandwiched together, a final copper foil is placed on the stack to form the top outer layer. Layers are then bonded into a single board by heating the assembly under pressure to cure the epoxy. After curing, any excess resin squeezed out of the sandwich around the edges is trimmed away.

At the start of the third stage of fabrication, the board consists of multiple layers bonded together, with copper foil cladded on its outer surfaces. Its construction can therefore be likened to that of a double-sided board; and its processing is likewise similar. The sequence consists of drilling additional through-holes, plating the holes to establish conduction paths between the two exterior copper films as well as certain internal copper layers, and the use of photolithography and etching to form the circuit pattern on the outer copper surfaces.

Testing and Finishing Operations After a circuit has been fabricated on the board surface, it must be inspected and tested to ensure that it functions according to design specifications and contains no quality defects. Two procedures are common: (1) visual inspection and (2) continuity testing. In ***visual inspection***, the board is examined visually to detect open and short circuits, errors in drilled hole locations, and other faults that can be observed without applying electrical power to the board. Visual inspections, performed not only after fabrication but also at various critical stages during production, are accomplished by human eye or machine vision (Section 40.6.3).

Continuity testing involves the use of contact probes brought simultaneously into contact with track and land areas on the board surface. The setup consists of an array of probes that are forced under light pressure to make contact with specified points on the board surface. Electrical connections between contact points can be quickly checked in this procedure.

Several additional processing steps must be performed on the bare board to prepare it for assembly. The first of these finishing operations is the application of a thin solder layer on the track and land surfaces. This layer serves to protect the copper from oxidation and contamination. It is carried out either by electroplating or by bringing the copper side into contact with rotating rollers that are partially submersed in molten solder.

A second operation involves application of a coating of solder resist to all areas of the board surface except the lands that are to be subsequently soldered in assembly. The solder resist coating is chemically formulated to resist adhesion of solder; thus, in the subsequent soldering processes, solder adheres only to land areas. Solder resist is usually applied by screen printing.

Finally, an identification legend is printed onto the surface, again by screen printing. The legend indicates where the different components are to be placed on the board in final assembly. In modern industrial practice, a bar code is also printed on the board for production control purposes.

34.3 Printed Circuit Board Assembly

A printed circuit board assembly consists of electronic components (e.g., IC packages, resistors, capacitors) as well as mechanical components (e.g., fasteners, heat sinks) mounted on a printed circuit board. This is level 2 in electronic packaging (Table 34.1). As indicated, PCB assembly is based on either surface-mount technology (SMT) or through-hole technology (see Figure 34.2). Some PCB assemblies include both SMT and through-hole components. The discussion in this section covers both categories as well as combinations of the two. The scope of electronic assembly also includes higher packaging levels such as assemblies of multiple PCBs electrically connected and mechanically contained in a chassis or cabinet. Section 34.4 explores the technologies by which electrical connections are made at these higher levels.

34.3.1 SURFACE MOUNT TECHNOLOGY

Since the late 1980s, the use of surface mount technology has grown to the point where it is now the dominant process in printed circuit board assembly. The earlier through-hole technology has the following inherent limitations in terms of packing density: (1) components can be mounted on only one side of the board, and (2) center-to-center distance between lead pins in leaded components must be a minimum of 1.0 mm (0.04 in) and is usually 2.5 mm (0.10 in).

Surface-mount technology uses an assembly method in which component leads are soldered to lands on the surface of the board rather than into holes running through the board (Historical Note 34.2). By eliminating the need for leads inserted into through holes in the board, several advantages are gained: (1) smaller components are available, with leads closer together; (2) circuit densities can be increased; (3) components can be mounted on both sides of the board; (4) smaller PCBs can be used for the same electronic system; and (5) drilling of the many through holes during board fabrication is eliminated although via holes to interconnect layers are still required [6]. Typical areas on the board surface taken by SMT components range between 20% and 60% compared to through-hole components.

Historical Note 34.2 *Surface-mount technology*

Printed circuit board assemblies based on through-hole technology were the predominant electronics packaging method from the 1950s through most of the 1980s, when surface-mount technology (SMT) started to be widely used.

SMT traces its origins to the electronic systems in the aerospace and military industries of the 1960s, with the IBM Corporation responsible for much of the original development work. The first components were small, flat ceramic packages with gull-wing leads. The initial reason why these packages were attractive, compared with through-hole technology, was the fact that they could be placed on both sides of a printed circuit board—in effect, doubling the component density. In addition, the SMT package could be made smaller than a comparable through-hole package, further increasing component densities on the printed circuit board.

In the early 1970s, further advances in SMT were made in the form of leadless components—components with ceramic packages that had no discrete leads. This permitted even greater circuit densities in military and aerospace electronics. In the late 1970s, plastic SMT packages became available, motivating the widespread use of surface-mount technology. The computer and automotive industries have become important users of SMT, and their demand for SMT components has contributed to the significant growth in this technology.

Despite its advantages, SMT has not been fully adopted by the electronics industry to the exclusion of through-hole technology. There are several reasons for this: (1) Owing to their smaller size, surface mount components are more difficult to handle and assemble by humans; (2) inspection, testing, and rework of the circuit assemblies are generally more difficult in SMT because of the smaller scale involved; and (3) certain types of components are not available in surface-mount form. This final limitation results in some electronic assemblies that contain both surface-mount and leaded components.

The sequence of steps to produce assembled PCBs is basically the same in SMT and through-hole technology: (1) place the components on the PCB, (2) solder the components to the PCB lands, (3) clean the assembly, (4) inspect, (5) test, and (6) rework. The details of the first two steps differ for SMT and through-hole technology. The final four steps are the same, except that the smaller scale of SMT makes inspection, testing, and rework more challenging.

Component placement in SMT means correctly locating the component on the PCB and affixing it sufficiently to the surface until soldering provides permanent mechanical and electrical connections. Two alternative placement and soldering

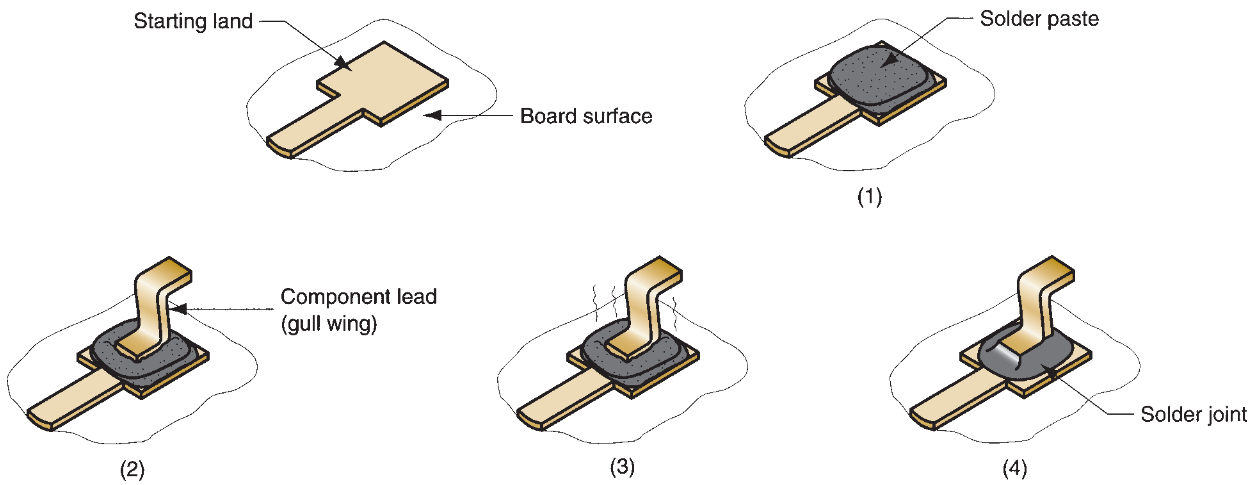

FIGURE 34.9 Solder paste and reflow method: (1) apply solder paste to desired land areas, (2) place components onto board, (3) bake paste, and (4) solder reflow.

methods are available: (1) solder paste and reflow soldering, and (2) adhesive bonding of components and wave soldering. It turns out that certain types of SMT components are more suited to one method whereas other types are more suited to the other method.

Solder Paste and Reflow Soldering The steps in this method are described in Figure 34.9. A solder paste is used to affix components to the surface of the circuit board. A ***solder paste*** is a suspension of solder powders in a flux binder. It has three functions: (1) it is the solder—typically 80% to 90% of total paste volume, (2) it is the flux, and (3) it is the adhesive that temporarily secures the components to the surface of the board. Methods of applying the solder paste to the board surface include screen printing and syringe dispensing. Properties of the paste must be compatible with these application methods; the paste must flow yet not be so liquid that it spreads beyond the localized area where it is applied.

After solder paste application, the components are placed onto the board surface by automated placement machines or semiautomatic placement machines. The automatic machines operate under computer numerical control (Section 37.3). Components (e.g., chip carriers, capacitors, resistors) to be placed are usually supplied in magazines or attached to tape in reels that can be loaded into feeders on the automated machines. The starting blank PCB is moved and positioned under the pick-and-place work head that places the components at specified locations on the board using a high-speed *x-y* positioning system. The work head uses suction nozzles to retrieve and handle components. Automated machines operate at cycle rates of up to 30,000 components per hour [20]. For mass production, they are frequently integrated into production lines, so that most or all of the processes performed on the board during assembly are completed in one automated sequence.

The semiautomatic machines assist a worker in the placement operation using a high-resolution vision system. Components to be placed are presented to the operator in the correct placement sequence by a turret mechanism, and then the operator picks the component and is guided by the vision monitor to place it at the target

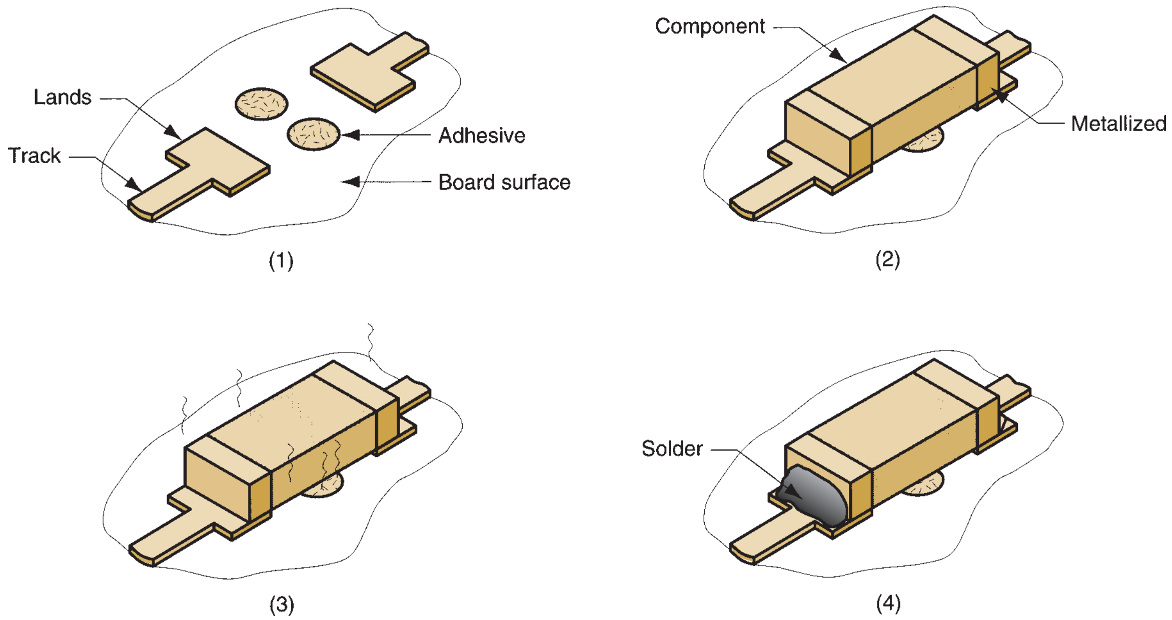

FIGURE 34.10 Adhesive bonding and wave soldering, shown here for a discrete capacitor or resistor component: (1) adhesive is applied to areas on the board where components are to be placed; (2) components are placed onto adhesive-coated areas; (3) adhesive is cured; and (4) solder joints are made by wave soldering.

location on the board. Cycle rates of 1,000 components per hour are claimed for some semiautomatic machines [15].

After all components are placed on the board, a low-temperature baking operation is performed to dry the flux binder; this reduces gas escape during soldering. Finally, the solder reflow process (Section 30.2.3) heats the solder paste sufficiently that the solder particles melt to form a high-quality mechanical and electrical joint between the component leads and the circuit lands on the board.

Adhesive Bonding and Wave Soldering The sequence of steps is depicted in Figure 34.10. Various adhesives (Section 30.3) are used for affixing components to the board surface. Most common are epoxies and acrylics. The adhesive is applied by one of three methods: (1) brushing liquid adhesive through a screen stencil; (2) using an automatic dispensing machine with a programmable *x-y* positioning system; or (3) using a pin transfer method, in which a fixture consisting of pins arranged according to where adhesive must be applied is dipped into the liquid adhesive and then positioned onto the board surface to deposit adhesive in the required spots.

Components are placed on the board by the same type of placement machines used with the solder paste assembly method. After component placement, the adhesive is cured. Depending on adhesive type, curing is by heat, ultraviolet (UV) light, or a combination of UV and infrared (IR) radiation. With the surface-mount components now bonded to the PCB surface, the board is put through wave soldering. The components themselves pass through the molten solder wave. Technical problems sometimes encountered in SMT wave soldering include components uprooted from the board, components shifting position, and larger components creating shadows that inhibit proper soldering of neighboring components.

34.3.2 THROUGH-HOLE TECHNOLOGY

In printed circuit assemblies using through-hole technology, the lead pins must be inserted into through-holes in the circuit board. Once inserted, the leads are soldered into place in the holes in the board. In double-sided and multilayer boards, the hole surfaces into which the leads are inserted are generally copper plated, giving rise to the name ***plated through-hole*** for these cases. After soldering, the boards are cleaned and tested, and those boards not passing the test are reworked if possible.

Component Insertion In component insertion, the leads of components are inserted into their proper through-holes in the PCB. In PCB assemblies based exclusively on through-hole technology (unusual today), a single board may be populated with hundreds of separate components (DIPs, resistors, etc.), all of which need to be inserted into the board. In modern electronic assembly plants, most component insertions are accomplished by automatic insertion machines. Components are loaded into these machines in the form of reels, magazines, or other carriers that maintain proper orientation of the components until insertion. A small proportion is done by hand for nonstandard components that cannot be accommodated on automatic machines. These cases include switches and connectors as well as resistors, capacitors, and certain other components. Although the proportion of component insertions accomplished manually is low, their cost is high because of much lower cycle rates than automatic insertions. Industrial robots (Section 37.4) are sometimes used to substitute for human labor in these component insertion tasks.

The insertion operation involves (1) preforming the leads, (2) insertion of leads into the board holes, and then (3) cropping and clinching the leads on the other side of the board. Preforming is needed only for some component types and involves bending of leads that are initially straight into a U-shape for insertion. Many components come with properly shaped leads and require little or no preforming.

Insertion is accomplished by a work head designed for the component type. Components inserted by automatic machines are grouped into three basic categories: (a) axial lead, (b) radial lead, and (c) chip carrier (e.g., dual-in-line package, Section 33.6.1). Typical axial and radial lead components are pictured in Figure 34.11. Axial components are shaped like a cylinder, with leads projecting from each end. Typical components of this type include resistors, capacitors, and diodes. Their leads must be bent, as suggested in the figure, to be inserted. Radial components have parallel leads and have various body shapes, one of which is shown in Figure 34.11(b). This type of component includes light-emitting diodes, potentiometers, resistor networks, and fuse holders. These configurations are sufficiently different that separate insertion machines with the appropriate

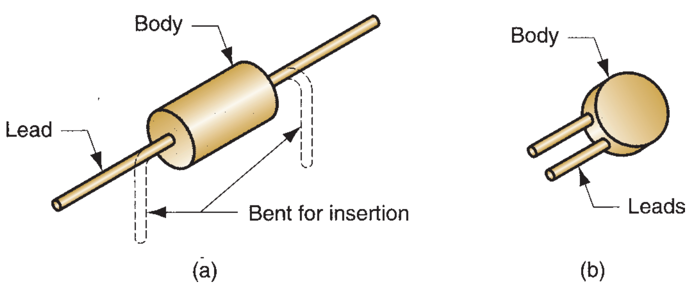

FIGURE 34.11 Two of the three basic component types used with automatic insertion machines: (a) axial lead and (b) radial lead. The third type, dual-in-line package (DIP), is illustrated in Figure 33.20.

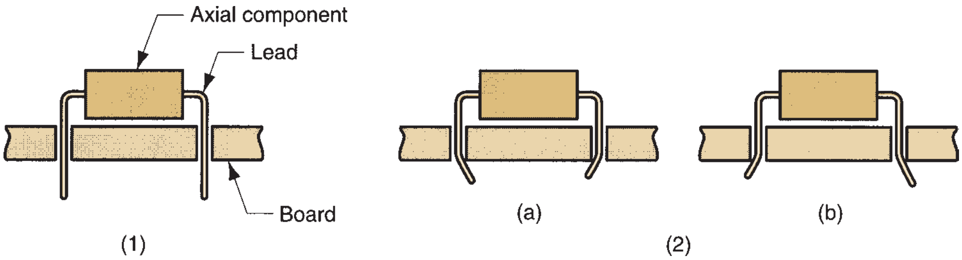

FIGURE 34.12 Clinching and cropping of component leads: (1) as inserted, and (2) after bending and cutting; leads can be bent either (a) inward or (b) outward.

work head designs must be used to handle each category. Accurate positioning of the board beneath the work head before each insertion is performed by a high-speed *x-y* positioning table.

Once the leads have been inserted through the holes in the board, they are clinched and cropped. Clinching involves bending the leads, as in Figure 34.12, to mechanically secure the component to the board until soldering. If this were not done, the component is at risk of being knocked out of its holes during handling. In cropping, the leads are cut to proper length; otherwise, there is a possibility that they might become bent and cause a short circuit with nearby circuit tracks or components. These operations are performed automatically on the underside of the board by the insertion machine.

The three types of insertion machines, corresponding to the three basic component configurations, can be joined to form an integrated circuit board assembly line. The integration is accomplished by means of a conveyor system that transfers boards from one machine type to the next. A computer control system is used to track the progress of each board as it moves through the cell and download the correct programs to each workstation.

Soldering The second basic step in PCB assembly is soldering. For inserted components, the most important soldering techniques are wave soldering and hand soldering. These methods as well as other aspects of soldering are discussed in Section 30.2.

Wave soldering is a mechanized technique in which printed circuit boards containing inserted components are moved by conveyor over a standing wave of molten solder (Figure 30.9). The position of the conveyor is such that only the underside of the board, with component leads projecting through the holes, is in contact with the solder. The combination of capillary action and the upward force of the wave cause the liquid solder to flow into the clearances between leads and through-holes to obtain a good solder joint. The tremendous advantage of wave soldering is that all of the solder joints on a board are made in a single pass through the process.

Hand soldering involves a skilled operator using a soldering iron to make circuit connections. Compared with wave soldering, hand soldering is slow since solder joints are made one at a time. As a production method, it is generally used only for small lot production and rework. As with other manual tasks, human error can result in quality problems. Hand soldering is sometimes used after wave soldering to add delicate components that would be damaged in the harsh environment of the wave-soldering chamber. Manual soldering has certain advantages in PCB assembly that should be noted: (1) Heat is localized and can be directed at a small target area; (2) equipment is inexpensive compared with wave soldering; and (3) energy consumption is considerably less.

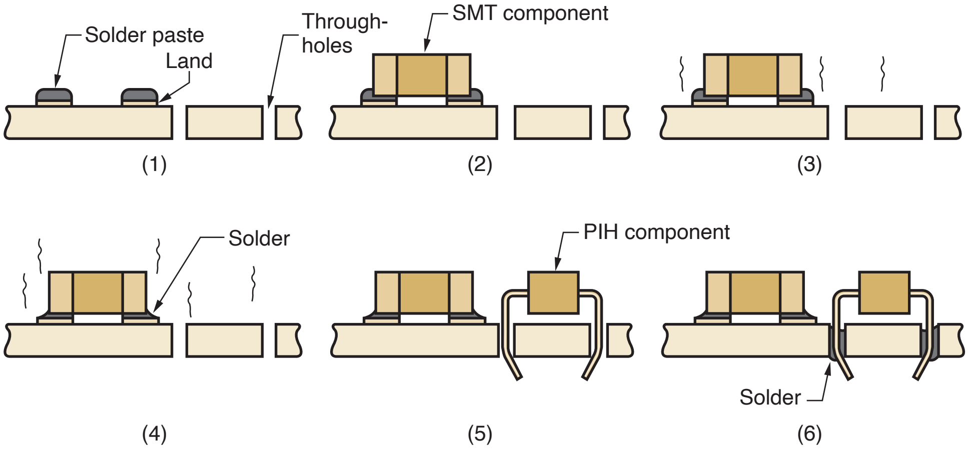

FIGURE 34.13 Typical process sequence for combined SMT-PIH assemblies with components on same side of board: (1) apply solder paste on lands for SMT components, (2) place SMT components on the board, (3) bake, (4) reflow solder, (5) insert PIH components, and (6) wave solder PIH components. This would be followed by cleaning, testing, and rework.

34.3.3 COMBINED SMT-PIH ASSEMBLY

The preceding discussion of SMT assembly methods assumed a relatively simple circuit board with SMT components on only one side. These cases are unusual because most SMT circuit assemblies combine surface-mounted and pin-in-hole components on the same board. In addition, SMT assemblies can be populated on both sides of the board, whereas PIH components are normally limited to one side only. The assembly sequence must be altered to allow for these additional possibilities, although the basic processing steps described in the two preceding sections are the same.

One possibility is for the SMT and through-hole components to be on the same side of the board. For this case, a typical sequence would consist of the steps described in Figure 34.13. More complex PCB assemblies consist of SMT-PIH components as in the figure, but with SMT components on both sides of the board.

34.3.4 CLEANING, INSPECTION, TESTING, AND REWORK

After the components have been connected to the board, the assembly must be cleaned, inspected for solder faults, circuit tested, and reworked if necessary.

Cleaning After soldering, contaminants are present on the printed circuit assembly. These foreign substances include flux, oil and grease, salts, and dirt, some of which can cause chemical degradation of the assembly or interfere with its electronic functions. One or more chemical cleaning operations (Section 27.1.1) must be performed to remove these undesirable materials. Traditional cleaning methods for PCB assemblies include hand cleaning with appropriate solvents and vapor degreasing with chlorinated solvents. Concern over environmental hazards in recent years has motivated the search for effective water-based solvents to replace the chlorinated and fluorinated chemicals traditionally used in vapor degreasing.

Inspection After cleaning, the PCB assembly is inspected for defective solder joints. Inspection of soldering quality is somewhat more difficult for surface-mount circuits (SMCs) because these assemblies are generally more densely packed, the solder joints are smaller, and joint geometries are different from those in through-hole assemblies. One of the problems is the way surface mount components are held in place during soldering. In through-hole assembly, the components are mechanically fastened in place by clinched leads. In SMT assembly, components are held by solder paste or adhesive. At soldering temperatures this method of attachment is not as secure, and components sometimes shift. Another problem with the smaller sizes in SMT is a greater likelihood of solder bridges forming between adjacent leads, resulting in short circuits.

Visual inspection is used to detect for board substrate damage, missing or damaged components, soldering faults, and similar quality defects that can be observed by eye. Machine vision systems are being used to perform these inspections automatically in a growing number of installations.

Testing Test procedures must be performed on the completed assembly to verify its functionality. The board design must allow for this testing by including test points in the circuit layout. These test points are convenient locations in the circuit where probes can make contact for testing. Individual components in the circuit are tested by contacting the component leads, applying input test signals, and measuring the output signals. More sophisticated procedures include digital function tests, in which the entire circuit or major subcircuits are tested using a programmed sequence of input signals and measuring the corresponding output signals to simulate operating conditions.

Another test used for printed circuit board assemblies is the substitution test, in which a production unit is plugged into a mock-up of the working system and energized to perform its functions. If the assembly performs in a satisfactory way, it is deemed as passing the test. It is then unplugged, and the next production unit is substituted in the mock-up.

Finally, a burn-in test is performed on certain types of PCB assemblies that may be subject to "infant mortality." Some boards contain defects that are not revealed in normal functional tests but which are likely to cause failure of the circuit during early service. Burn-in tests operate the assemblies under power for a certain period of time, such as 24 or 72 hours, sometimes at elevated temperatures, such as 40°C (100°F), to force these defects to manifest their failures during the testing period. Boards not subject to infant mortality will survive this test and provide long service life.

The smaller scale of SMT poses problems in circuit testing because less space is available around each component. Contact probes must be physically smaller and more probes are required because SMT assemblies are more densely populated. One way of dealing with this issue is to design the circuit layout with extra lands whose only purpose is to provide a test probe contact site. Unfortunately, including these test lands runs counter to the goal of achieving higher packing densities on the SMT board.

Rework When inspection and testing indicate that one or more components on the board are defective, or certain solder joints are faulty, it usually makes sense to try to repair the assembly rather than discard it together with all of the other components that are not defective. This repair step is an integral part of electronic assembly plant operations. Common rework tasks include touchup (repair of solder faults), replacement of defective or missing components, and repair of copper film that has lifted from the substrate surface. These tasks are manual operations, requiring skilled workers using soldering irons. Rework of surface-mount assemblies is more difficult than in conventional PIH assemblies, again because of the smaller component sizes. Special tools are required, such as small-bit soldering irons, magnifying devices, and instruments for grasping and manipulating the small parts.

34.4 Electrical Connector Technology

PCB assemblies must be connected to back planes, and into racks and cabinets, and these cabinets must be connected to other cabinets and systems by means of cables. The growing use of electronics in so many types of products has made electrical connections an important technology. The performance of any electronic system depends on the reliability of the individual connections linking the components of the system together. This section is concerned with connector technology that is usually applied at the third and higher levels of electronics packaging.

To begin, there are two basic methods of making electrical connections: (1) soldering and (2) pressure connections. Soldering was discussed in Section 30.2 and throughout the current chapter. It is the most widely used technology in electronics. ***Pressure connections*** are electrical connections in which mechanical forces are used to establish electrical continuity between components. They can be divided into two types: permanent and separable.

34.4.1 PERMANENT CONNECTIONS

A permanent connection involves high-pressure contact between two metal surfaces, in which one or both parts are mechanically deformed during the assembly process. Permanent connection methods include crimping, press fit technology, and insulation displacement.

Crimping of Connector Terminals This connection method is used to assemble wire to electrical terminals. Although assembly of the wire to the terminal forms a permanent joint, the terminal itself is designed to be connected and disconnected to its mating component. There are a variety of terminal styles, some of which are shown in Figure 34.14, and they are available in various sizes. They all must be connected to conductor wire, and crimping is the operation for doing this. ***Crimping*** involves the mechanical deformation of the terminal barrel to form a permanent connection with the stripped end of a wire inserted into it. The crimping operation squeezes and closes the barrel around the bare wire. Crimping is performed by hand tools or crimping machines. The terminals are supplied either as individual pieces or on long strips that can be fed into a crimping machine. Properly accomplished, the crimped joint will have low electrical resistance and high mechanical strength.

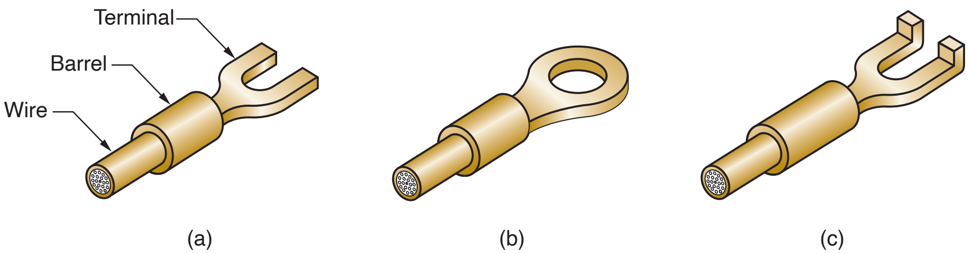

FIGURE 34.14 Some of the terminal styles available for making separable electrical connections: (a) slotted tongue, (b) ring tongue, and (c) flanged spade.

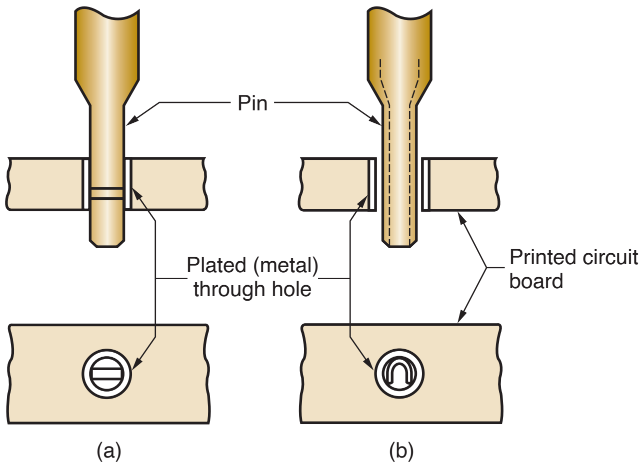

FIGURE 34.15 Two types of terminal pins in electronics press fit technology: (a) solid and (b) compliant.

Press Fit Technology Press fit in electrical connections is similar to that in mechanical assembly, but the part configurations are different. Press fit technology is widely used in the electronics industry to assemble terminal pins into metal-plated through-holes in large PCBs. In that context, a ***press fit*** involves an interference fit between the terminal pin and the plated hole into which it has been inserted. There are two categories of terminal pins: (a) solid and (b) compliant, as in Figure 34.15. Within these categories, pin designs vary among suppliers. The solid pin is rectangular in cross section and is designed so that its corners press and even cut into the metal of the plated hole to form a good electrical connection. The compliant pin is designed as a spring-loaded device that conforms to the hole contour but presses against the walls of the hole to achieve electrical contact.

Insulation Displacement Insulation displacement is a method of making a permanent electrical connection in which a sharp, prong-shaped contact pierces the insulation and squeezes against the wire conductor to form an electrical connection. The method is illustrated in Figure 34.16 and is commonly used to make simultaneous connections between multiple contacts and flat cable. The flat cable, called ***ribbon cable***, consists of a number of parallel wires held in a fixed arrangement by the insulation surrounding them. It is often terminated in multiple pin connectors that are widely used in electronics to make electrical connections between major subassemblies. In these applications, the insulation displacement method reduces wiring errors and speeds harness assembly. To make the assembly, the cable is placed in a nest and a press is used to drive the connector contacts through the insulation and against the metal wires.

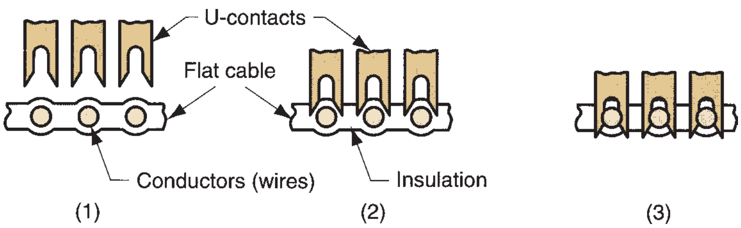

FIGURE 34.16 Insulation displacement method of joining a connector contact to flat wire cable: (1) starting position, (2) contacts pierce insulation, and (3) after connection.

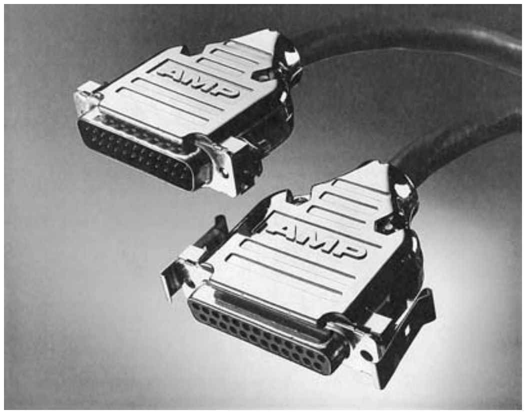

FIGURE 34.17 Multiple pin connector and mating receptacle, both attached to cables. (Photo courtesy of Tyco Electronics Corporation, a TE Connectivity Ltd. Company.)

34.4.2 SEPARABLE CONNECTORS

Separable connections are designed to permit disassembly and reassembly; they are meant to be connected and disconnected multiple times. When connected they must provide metal-to-metal contact between mating components with high reliability and low electrical resistance. Separable connection devices typically consist of multiple contacts, contained in a plastic molded housing, designed to mate with a compatible connector or individual wires or terminals. They are used for making electrical connections between various combinations of cables, printed circuit boards, components, and individual wires.

A wide selection of connectors is available to serve many different applications. The design issues in choosing among them include (1) power level (e.g., whether the connector is used for power or signal transmission); (2) cost; (3) number of individual conductors involved; (4) types of devices and circuits to be connected; (5) space limitations; (6) ease of joining the connector to its leads; (7) ease of connecting with the mating terminal or connector; and (8) frequency of connection and disconnection. Some of the principal connector types are cable connectors, terminal blocks, sockets, and connectors with low or zero insertion force.

Cable connectors are devices that are permanently connected to cables (one or both ends) and are designed to be plugged into and unplugged from a mating connector. A power cord connector that plugs into a wall receptacle is a familiar example. Other styles include the type of multiple pin connector and mating receptacle shown in Figure 34.17, used to provide signal transmission between electronic subassemblies. Other multiple pin connector styles are used to attach printed circuit boards to other subassemblies in the electronic system.

Terminal blocks consist of a series of evenly spaced receptacles that allow connections between individual terminals or wires. The terminals or wires are often attached to the block by means of screws or other mechanical fastening mechanisms to permit disassembly. A conventional terminal block is illustrated in Figure 34.18.

A ***socket*** in electronics refers to a connection device mounted to a PCB, into which IC packages and other components can be inserted. Sockets are permanently attached to the PCB by soldering and/or press fitting, but they provide a separable connection method for the components, which can be conveniently added, removed, or replaced in the PCB assembly. Sockets are therefore an alternative to soldering in electronics packaging.