SURFACE MOUNT TECHNOLOGY

Surface mount assembly (SMA) is the description of a technology that incorporates the electrical and mechanical components on printed wiring boards (PWBs) or a similar type circuit substrate. The name surface mount comes from the methodology of attaching the components. Surface mount places the components on the surface of the supporting media, the PWBs, or other types of substrate. There are two types of connections between the discrete packages and the supporting board material—those with leads, which require plated through hole (PTH), and others with leads of pads for surface mounting the devices. The surface mount packages have connections that are significantly below those available for through hole components. This yields a more compact design that reduces the area required and provides a denser circuit.

There are many categories of packages that can be employed in surface mount assembly. They can be classified in two categories: leadless devices and leaded chip carriers. The leadless devices include leadless chip carriers, surface mount arrays, capacitors, resistors, and inductors. Leaded devices include packages for active devices, of either plastic or ceramic, and have a particular lead configuration. The resistors, capacitors, inductors, and various types of semiconductor circuits can come in two configurations, leaded and chip. The leaded components are traditionally employed in through-hole mounting and are not normally employed in SMA. The

J. Webster (ed.), *Wiley Encyclopedia of Electrical and Electronics Engineering*. Copyright © 1999 John Wiley & Sons, Inc.

chip components are available in protected and unprotected states. The chip is a minimum configuration for the particular device. In the case of capacitors, it may be the actual device that is provided in the leaded package. The bare device has connection points that may be solder plated.

The semiconductors are typically manufactured in multileaded ceramic or plastic packages, which can be with leads extending from the package or leadless (connections are on the package). (This provides the capability of testing the devices to insure conformance to specifications.) The specifications for the devices conform to a Joint Electronic Development Engineering Council (JEDEC) standard, so that the design of the PWB for attachment can be standardized. Bare chip circuits are also employed in special circumstances. One arrangement, chip-on-board (COB), requires the application of wire attachment from the chip (semiconductor device) to the PWB. The other configuration is flip chip. Flip chip employs the bare semiconductor die (chip) with special solder placement on the surface to provide connection pads (bumps) for the semiconductor device. The chip is mounted circuitry toward the PWB. The processing of the assembly causes the bumps to reflow and attach the chip to the PWB.

Printed wiring boards (PWBs), also known as printed circuit boards (PCBs), are the most commonly used interconnect mechanism to complete circuitry. The pattern of electrical circuitry, also known as paths or traces, is manufactured to provide the appropriate connections between various devices. Surface mount is a card assembly method that relies on a solder connection for both physical and electrical packages to board connections. The PTH joining method employs a close mechanical fit between the components and the board via hole for the location of the component, and the solder provides the final bond as well as the electrical connections. PWBs are manufactured from nonconductive, stable materials. These materials can be employed for the manufacture of single-sided, two-sided, or multilayer PWBs. The single sided PWB has circuitry on only one side. The components are inserted through the boards and soldered to the bottom side, which has the circuitry. Two sided boards have circuitry on both sides, and either side or both sides can be employed for soldering components in place. Multilayer boards are built of many levels of circuitry that are laminated to form many layers of interconnections. Most PWBs are designed on a grid pattern which specifies the minimum dimension between different circuit elements. The standard package of the mid 1980s had a lead pitch, center to center distance between the individual leads, of 0.1 in. or 100 mil. Applying the design of complex circuitry with traces of 20 mil provided adequate space to connect the circuit elements. As the circuitry became more complex, the need for more connections witnessed the manufacture of multilayer circuits that could provide more interconnects at the expense of additional processing layers. Also, work is being done on packages with spacing of less than 20 mils to provide more dense packing. The smaller dimensions provide challenges for both the designer and the manufacturer. As the density of the interconnects increases, the manufacturing tolerances decrease. Decreasing tolerances has a direct impact on yields. The better the design, the greater the manufacturing tolerances. The miniaturization of circuitry causes a greater need for a complete design for manufacturing concept. [See Boothroy et al. (1) and White et al. (2) for more details.] The advantage of miniaturization is a decrease in the package size, a decrease in product volume, a decrease in cost, and an increase in the reliability due to better manufacturing control.

HISTORY

Surface mount assembly has evolved in response to the manufacturing need for more miniaturization and for more complex products. The start of surface mount began with the development of thick film hybrid technology. The first attempts at thick film hybrids were made during the Second World War. Variable resistors were produced by printing a resistive paste on a carrier and firing the circuit. The design of the circuit was such that a movable contact provided the ability to change position on the printed resistor and, consequently, change the resistance in the circuit. The application of hybrid circuits came into widespread use in the 1960s. The needs of military applications, that required more stringent operating environments than the conventional PWBs, required a radical solution. The development of ceramic substrates and conductive pastes provided the ability to develop a stable circuit. Populating the circuit with devices required providing holes in the ceramic substrate, which made the substrate more expensive and weakened it, or developing a means of mounting components directly onto the substrate. The development of the integrated circuit provided another impetus to mount devices to the substrate.

The hybrid circuits have two configurations, thick and thin film. The thick film circuits are produced by printing conductive and resistive pastes and, subsequently, firing the pastes to stabilize the material and provide the operational characteristics. The thin film circuits were similar to thick film except that the conductors and resistors were vacuum deposited. This process provides for finer geometries but cannot handle large amounts of power. Both types of circuits needed additional components to complete their functionality. There was a need to provide a means of mounting and connecting the additional components. Devices can be connected to this circuitry by soldering, eutectic and lower temperature materials, or by epoxy, both conductive and nonconductive. The start of these technologies led to the development of other surface mount techniques.

The early application of high performance circuitry required a good thermal and electrical contact with the semiconductor. Through a combination of heating and mechanical motion, a eutectic bond can be produced. This was normally a gold-silicon interface that required processing temperatures in excess of 400°C. As less stringent circuits were developed, the application of tin-lead solders was employed to attach prepackaged circuits. These were lower cost devices. As miniaturization of devices accelerated, the drive to produce low cost devices provided a rationale for experimenting with new approaches to circuit assembly. Prepackaged plastic devices could be attached to the substrate inexpensively, resulting in less expensive products. The driving force behind the initial development of surface mount assembly was high volume, low cost consumer products. In the late 1970s, the Japanese were investigating high volume manufacturing of consumer electronics. The cost of the ceramic substrate was too expensive for the consumer market. Conventional PWBs were made, and components were attached to the surface. Since the surface mount component is smaller than the corresponding leaded one, the corresponding package, a SMT assembly, is smaller than a discrete assembly. (This is very understandable since the leaded component can contain the entire surface mount one.) While the initial surface mount assemblies were very elementary circuits, the potential cost savings provided the push needed to look at automating the process. During the early 1980s, the assembly equipment manufacturers in the United States emphasized the development of equipment for placing through hole components, while the Japanese were developing high speed chip placement equipment. The requirements for high speed placement equipment require tighter tolerances and fewer selections of dimensions for devices. Those volumes, which were required for the consumer market, caught the attention of the component suppliers. The result was that in the early 1980s, the manufacturers of chip capacitors went from almost 20 different sizes to three. This change precipitated lower prices and an increased ease of manufacture. With the availability of inexpensive production, the application of surface mount technology increased. The mid 1980s witnessed the expansion of surface mount into higher quality products. As the reliability of the devices improved, this acceptance increased even more.

The late 1980s witnessed a broad acceptance of surface mount technology in a large number of consummer applications. The development of the multichip module (MCM) during this time provided a high reliability package that could apply the automated surface mount placement equipment. Successful high-reliability applications, like the Fully Automated Digital Electronics Module (FADEC) that General Electric developed for jet engine controls, proved that surface mount assembly in modules provided a packaging technology that could withstand almost any environment. As microprocessors kept shrinking and their functionality kept increasing, the personal computer market applied surface mount technology to increase reliability and reduce costs.

TERMINOLOGY

The terminology employed with surface mount assembly is derived from the two affiliated technologies, PWB assembly and device manufacturing.

Package Types

The development of packaging standards provided the basis for uniformity and the ability to develop high speed manufacturing equipment. The Joint Electronic Development Engineering Council (JEDEC) develops the packaging standards. Some of the more complex surface mount packages are shown in Fig. 1.

Three Letter Acronyms (TLAs)

With any technology, a code language develops that simplifies its terminology. The resulting acronyms are usually two to four letters but are classified as three letter acronyms. Some of the more typical ones are given below.

- CC—chip carrier: This refers to the packaging of an electronic device, usually a semiconductor device, into a package that protects the device and provides for electrical contact and mechanical mounting to a more complex circuit. The package can have leads extending from the device or be leadless.
- COB—chip on board: refers to a packaging technique where bare semiconductor die are epoxied to PWBs and wire bonded to the conductive paths on the PWB. (Wire

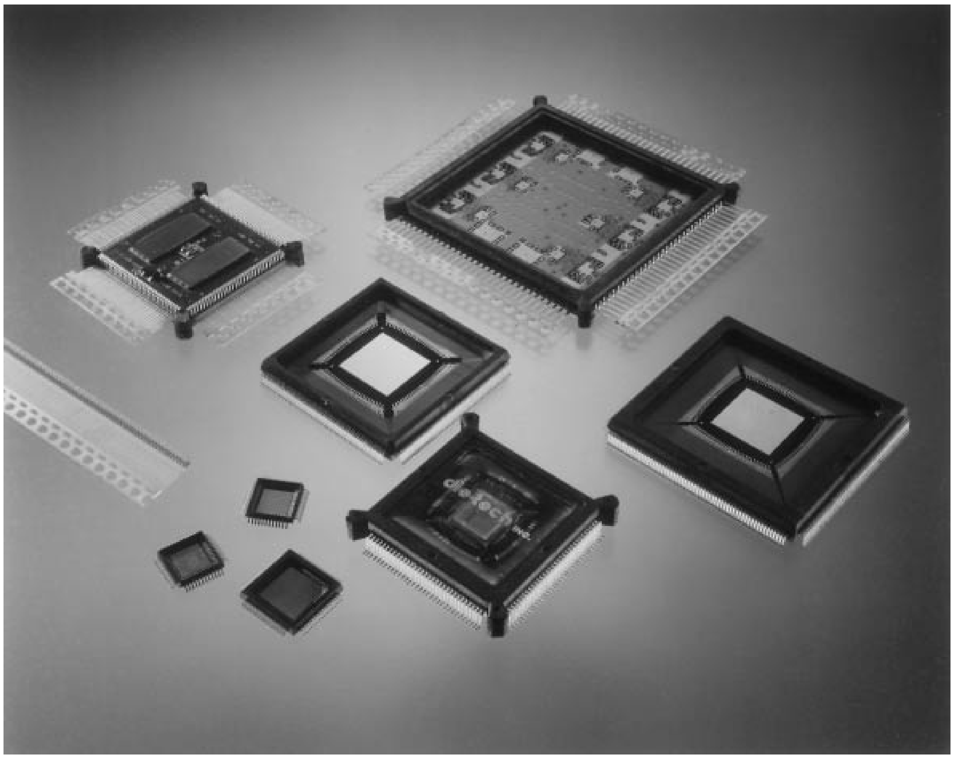

Figure 1. Complex surface mount packages that contain smaller surface mount devices. (Courtesy of DieTech, Inc.)

bonding is the traditional method for attaching semiconductors to their packages.) The resulting interconnnection is then protected by a hard epoxy that provides a thermal expansion match for the PWB and the wirebonds.

- CLCC—ceramic leadless chip carrier: refers to the package type shown in Fig. 1(a) that contains the device in a ceramic package.
- LCC—leadless chip carrier: This is a packaging technique that incorporates a device into a carrier that does not have external leads extending from the package. The leads are actually on the package's periphery and are an integral part of the package.
- MCM—multichip module: This is a more complex arrangement than the chip carrier in that any number of devices can be interconnected with the package and a larger package results that provides complex functionality. The advantage of the higher level package is that more complex functionality can be verified.
- MELF—metalized electrode face: This is a packaging technique for cylindrical parts without leads. Both ends of the cylinder have metalized terminations.
- PCB—printed circuit board: The basic building block of the electronic circuitry, the printed circuit board is also known as the printed wiring board (PWB).
- PLCC—plastic leaded chip carrier: This is the plastic equivalent of the CLCC. This part has leads coming off all four sides that wrap underneath the part in a "J" shape.
- PWB—printed wiring board: The material that provides both the platform for supporting the electronic devices and the means of interconnection for these devices. Typically, the material is resin based and capable of withstanding high stress environments. The fabrication of the circuitry results in the type of circuit, single sided, two sided, or multilayer. Single sided refers to material with the circuitry on only one side. Double or two sided refers to material with the circuitry on both the top and bottom. Multilayer refers to material that has more than one level on one of the sides.
- QFJ—quad flat pack J lead: This refers to a QFP with "J" leads extending from the package. The leads provide both electrical conductivity and mechanical mounting.
- QFP—quad flat pack: This is the description of a package that typically is square with contacts on all four sides. The configuration of this package is either leaded or without leads.
- RA—rosin activated: This term refers to additives that are added to solder that increase the ability of the solder to adhere to the circuitry and the device leads. There are potential problems from using this activation that require especially good cleaning of any residues after the solder process.
- RMA—rosin mildly activated: Less active than the RA, mildly activated is less aggressive in chemical reactions with the devices or circuitry.
- SMA—surface mount assembly: The term that refers to a process that employs electronic components attached to the surface of the circuitry.
- SMT—surface mount technology: The term refers to the technology that involves any portion of the surface mount assembly process.
- SOIC—small outline integrated circuit: Generic term for parts with multiple leads on two opposing sides and no leads on the other two sides. Typically, there will be the same number of leads on both sides of the part, and the lead spacing will be consistent throughout the part, although the center leads may be missing.
- SOJ—small outline J lead: This part is a special case of the SOIC where all the leads are bent in a "J" shape.
- SOP—small outline package: Frequently, a part of the name for various types of SOICs. Q = quarter, S = shrink, T = thin, and V = very.
- SOT—small outline transistor: Rectangular plastic packages that house transistors or diodes. Typically, these have three or four leads.

SURFACE MOUNT ASSEMBLY PROCESSES

The critical element in surface mount assembly is providing the proper mechanism for attaching the components to the substrate. The process of attaching the components to the circuitry involves either solder attachment or epoxy mount.

Solder

The process of attaching components by soldering involves raising the temperature of the solder until it becomes molten and cooling the circuitry until the solder hardens. The solder consists of metals that have a relatively low melting point, usually below 315°C. This liquid metal composition is structured to adhere to both the components and the conductive material on the substrate. The ability of the material to adhere to the surfaces is a function of its wetability. Properly prepared surfaces show good adhesion. Due to the nature of the alloys involved, inorganic materials and metallic oxides can inhibit the formation of the bonds. In cases like these, there are chemicals, that is fluxes that can be employed to enhance the bondability of the materials.

Fluxes

The function of the flux is to provide an untarnished surface that is capable of easily spreading the liquid metal to cover the desired surfaces. Since there are varying degrees of contamination and different solubilities of materials, the characteristics of the fluxes have been tailored for specific circumstances. In general, there are three types of fluxes employed in electronics manufacturing: activated, nonactivated, and mildly activated rosin fluxes. Water–white rosin flux is a chemically inactive and electrically insulating flux. When heated above its melting point, it becomes active and reacts with some metals. Activated fluxes have additives that exhibit more aggressive behavior and promote solderability in more difficult conditions. The mildly activated fluxes are more aggressive than the nonactivated but they do not have the residue problem of the activated fluxes. The activated fluxes require thorough cleaning to insure removal of the flux, which can cause damage to the components if left on the circuit.

Composition

The main constituents of solders have been tin and lead in various compositions. Recently, other materials have been incorporated into the metallurgy in order to reduce the amount of lead employed in the products. Eutectic solder refers to the composition of the material that has either a solid or a liquid state and no plastic state. For tin–lead, this composition is 63/37%. The melting point is 183°C. Other percentages provide melting points that differ slightly based on the percentages of the metals, but they exhibit a transition state where the solder becomes plastic and moldable before melting. Many different compounds are being used in production, based on the constraints of the product or the environmental considerations of employing lead in products.

Adhesives

There are two types of adhesives employed in surface mount assemblies: epoxies and acrylics. The epoxies are further subdivided into thermosetting, thermoplastic, elastomeric, or alloy. Thermosetting involves a chemical or thermally induced reaction. Single part epoxies normally require thermal elevation to initiate the chemical reaction. Two part epoxies provide the catalyst by the combination of the two elements. Elastomer is named after material with elongation properties that can be employed in situations where a degree of stress relief is required. The alloy is a combination of materials from the preceding three categories.

Cleaning

The application of solder has the potential of leaving an undesirable residue after the solder process. Cleaning is a critical element of the process. The application of water-white rosin should leave minimal residue and has been employed without cleaning. The slightly more active flux, mildly activated, should always be cleaned from assemblies. Activated flux must be cleaned because the residue is still active and can cause contamination and corrosion. There is a direct correlation between cleanliness and circuit reliability.

SURFACE MOUNT PROCESSING

In the development of a product, the capabilities of the manufacturing process as well as the functionality of the circuit determine the configuration of the design and the components that are required for the manufacturing process. The production of a high volume, high reliability product, like a digital cellular phone, has different constraints from a portable, digital clock. The purpose of this section is not to develop the methodology of design for manufacturing but to overview the processes required for manufacturing. For small volume or prototype quantities, manual assembly can be employed to produce a working product. The potential problem with small quantity manual assemblies is that the learning required to produce good quality manual assembly is not immediately available. Consequently, the application and testing of prototype product should not be expected to approach that of a debugged assembly process.

For this article, assume that the product being introduced into production has been through a prototyping process that eliminated any design flaws that would require revision of the assembly. The revision process will not be considered. Finally, we will assume that the design and manufacturing teams have been working together to develop a manufacturable product that does not require modifications or changes to achieve a shippable product. The selection and application of equipment is dependent on the desired volumes. The descriptions below cover manual, semiautomatic, and automatic assembly tools. The selection of the appropriate equipment is left to the reader.

Assembly Equipment

Once the type of board that will be built is identified, it is necessary to select the equipment that will be used to build it. This article considers the more common methods of assembly and identifies the equipment that would be typically involved in each stage. The steps discussed will follow the typical order in building a board. There are many reasons for deviating from this model. Some of these are discussed at the end.

Solder Paste Application. Frequently when building a board, the top side is built first. The first step in top side assembly is normally placing solder paste on the board. The paste serves many functions; it holds the part in place during assembly, provides the material to clean the contacts on the part, and provides the material to form the solder joints.

Solder paste is made up of two key components: flux and solder. The flux has the job of holding the compound together and, when heated, cleaning the pads on the board, the terminations on the components, and the solder particles. The solder has the job of forming the electrical connections when it is melted and cooled.

Because of these functions, the paste must be put on the board in the correct quantities and aligned on the pads. If there is too much solder, there might be shorts; if there is too little solder, the mechanical bond may not be sufficiently strong to hold the components. There are two basic ways of putting the paste on the board: dispensing and screening.

Dispensing Solder Paste. Solder paste dispensing involves delivering the paste in a syringe (typically 10 cc or 30 cc). The paste is forced out onto the PWB at the desired locations. (The methods of forcing the paste out are the same as dispensing glue.) There are many machines available to do the dispensing.

The simplest machine is a very simple pump with a hand held syringe that the operator moves from location to location. The operator usually activates a foot switch that will start the dispensing process. At the other extreme, a gantry style robot moves the dispensing head to the desired location, and the controller dispenses the paste. The advantage of the robots is that they are faster and more repeatable. The advantages and disadvantages of the various dispensing techniques are discussed in the section entitled “Glue Application.”

Dispensing solder paste offers the advantages of being flexible, requiring no custom tooling, and being easy to clean up. The machines that dispense paste are normally programmable to allow quick changes to be made. These machines typically require no custom tooling to be created per assembly. Clean up normally includes only cleaning the nozzle, which is one small piece. For these reasons, paste is frequently dispensed for prototype boards.

Dispensing also has its disadvantages: it is slow, require low viscosity paste, and lacks repeatability. Because each location on a board must have the paste forced onto it one location at a time, and since most assemblies contain multiple locations, dispensing a board can take a long time. In order to dispense paste, the paste must be thin enough to be forced through the needle. This means that the paste should not be left standing on the pads because will slump, leading to potential shorts. Depending on the dispensing method, the volume of paste and the location of the paste may be difficult to repeat. For these reasons, dispensing paste is usually not used in production.

Screen Printing Solder Paste. Screen printing is a simple process in which a template is held over the substrate and a viscous fluid is then forced through. In electronics manufacturing, the substrate is the PWB, and the viscous fluid is frequently solder paste. The template is either a stencil or a screen.

A stencil is a metal plate, the same thickness as the desired paste height, with holes cut out in it where the paste is desired. Almost any type of metal may be used, but stainless steel is the most common material currently used. The holes may be formed by any number of methods, with chemical etching and laser cutting among the most common. The holes are made the same size and shape as the desired pattern of paste on the board.

A screen is a mesh, typically made of steel threads closely spaced, that is covered with a coating. (The gauge of the mesh indicates the number of wires per inch, so the higher numbers provide the ability to make thinner coating; for example, 325 mesh is finer than 200.) The coating is removed from the areas where the paste is desired (the mesh is still there). Each location where paste is desired is actually formed of many smaller openings, forcing the screen to be thicker to achieve the same volume of paste. Because of the many smaller openings, the choice of paste to use with the screen must be thought out very carefully and may limit the applications that can be used. A screen, while cheaper than a stencil, is not used very often because it does not last as long, gives less desirable results, and is harder to clean.

The viscous fluid, typically paste, is moved along the top of the stencil or screen from one end to the other by a squeegee, which also pushes the paste downward through the openings in the stencil or screen. Typically, three variables concerning the squeegee are controlled: hardness, pressure, and speed. The hardness of the squeegee determines how it will react when it is used. Pressure and speed are inversely related to each other. Increasing pressure will have similar effects as decreasing speed. The objective with these two variables is to find the right balance that yields good coverage on the board.

A machine is not required for this process. There are many companies that use an operator to do the screening. However, this method has several drawbacks, the largest of which is the lack of consistency. Typically, a machine will be used to improve the process. The machines range from semi automatic to fully automatic. The semi automatic printers require the operator to place the boards on the machine, verify and align the board and stencil, tell the machine to print, and then remove the board. The automatic machines will perform all the tasks automatically, including passing the board to the next machine.

Due to all the reasons mentioned above, screen printing has many advantages: repeatable results, speed, choice of pastes, and ease of use. The disadvantages of screen printing are few, mostly related to the cost of the stencil or screen. For these reasons, screen printing is usually the preferred choice for placing solder paste on the board for production runs.

Glue Application. Glue is applied to the board to hold the components in place for the rest of manufacturing and for certain user requirements. The bottom side SMD components are typically glued, while the top side of the board usually is not. The choice of the glue must be considered carefully considered since it must have several properties: The glue must have strength for manufacturing, must be easily broken for rework, must not be harmful to the components or board, and must be easily worked with and cleaned. There are several ways to put the glue on the board: dispensing, pin transfer, and screen printing.

Dispensing. Dispensing refers to taking the glue from a container (typically a syringe) and forcing the desired amount through a needle onto the desired location. There are two basic categories for dispensing: time-based and positive displacement. Each has its advantages.

Time-Based Dispensing. Time-based dispensing involves applying pressure for a certain amount of time to dispense the glue. Air is usually applied to the back of the syringe to supply the pressure to dispense the material. This system is usually referred to as "air over." Air over is inexpensive, easy to maintain, and easy to clean. Unfortunately, as the level of glue in the syringe becomes lower, more time is required to achieve the same volume of glue. However, this can be compensated for through various machine controls.

Positive Displacement. In order to overcome the disadvantages of time-based dispensing, various methods of positive displacement have been created. The two most common are piston and screw.

An extremely common method of positive displacement is using a piston to force an exact amount of adhesive out the end of the dispensing tip. This is accomplished by keeping a steady pressure on the syringe and pulling the piston up high enough to allow the adhesive to enter the dispensing chamber. The piston is then brought down to force an amount of glue (equal to the volume of the chamber that the piston then occupies) out onto the substrate.

The screw method uses a screw which, when turned, forces an amount of adhesive out the chamber which is proportional to the amount that the screw is turned. This format is very similar to many pumps used for other applications. This method is easy to vary, as the more the screw is turned, the more glue comes out the end.

Positive displacement systems provide reliable consistent glue dot sizes. With this advantage comes a few disadvantages; more expense and more complexity.

Pin Transfer. The methods of dispensing adhesive mentioned so far are extremely flexible but can require a long time to apply all of the adhesive required on the board. There are several industries that do not need the flexibility mentioned above but need very quick cycle times (less than 8 s to apply all the adhesive). In order to meet this need, pin transfer was created. Pin transfer allows the whole board to be covered at once and allows reloading while the board is being transferred. There is a die made for each different type of board that has pins sticking down from it at each of the locations where adhesive is desired. The die is loaded by dipping the pins in a tub of adhesive. A small amount of adhesive will stick to each pin, with the quantity varying depending on the size of the pin. The die is then brought over to the substrate, and when contact is made, the adhesive transfers to the substrate. Pin transfer is fast but also expensive due to the tooling, and it requires long cleanup times.

Stencil Printing. Many manufacturers wanting to make use of their current equipment look at another method of adhesive application, stencil printing. Stencil printing adhesive is very similar to stencil printing paste. The advantages include that no new equipment needs to be bought, but the board must be flat with nothing sticking out on the side of the board. This limits the number of options that are available in the order of manufacture.

Placement Equipment

After the material that is to hold the parts on the substrate during assembly is applied, the next step is to start populating the substrate. This requires that the parts be placed on the substrate in the correct location and the correct orientation. There are many standards designed to define what is acceptable and what is not. The standards are a compromise between perfect placements, production speeds required, and requirements of the finished product. Whatever standard is used, the process used must be capable of building to that standard in an efficient and timely manner. There are three basic ways to place parts on a substrate, each with its unique advantages.

Manual. Placing parts manually involves setting up an operator with a picking tool and all the parts to be placed. A machine is optional for this type of placement. The machines can provide instructions and even some assistance in picking and holding the parts. The operator follows a set of directions to determine where each part goes on the board. This is the least expensive and slowest way to build boards, but it is appropriate for prototype work. While this method can be used for production, it relies completely on the operator skills.

Semiautomatic. There are many machines that fall in the semiautomatic range. The machines provide help to the operator in many ways. The machines may require the operator to place the parts individually in a location that the machine can reach and then tell the machine to go. The other extreme for this class of machines requires the operator only to place the substrate in the machine, let the machine place all the parts, and then remove the board. The machines require less precision from the operator since the operator is not actually placing the parts. This class of machines is frequently used in contract manufacturing and other environments where batch processing of the substrates is preferred.

Automatic. The automatic machines require the least amount of operator intervention. These machines automatically bring the substrates in, populate them, and then pass them to the next machine. See Fig. 2. This type of machine is used in environments where the continuous building of product is required. The machines in this class frequently are specialized in order to gain more speed. The machines are typically divided into two categories: chip shooters and fine pitch placers. The chip shooters are designed to put down the small parts (typically resistors, capacitors, and diodes) very quickly. The chip shooters, Fig. 3, typically use tape and reel parts so that the feed action can be very quick. These machines are designed so that the operator just has to replenish the parts as the machine uses them. The fine pitch placers, Figs. 4 and 5, usually take many types of inputs as the parts frequently come on trays or in sticks. These machines are very flexible and can do just about any type of part, but are much slower than the chip shooters. A typical line would consist of a chip shooter and a fine pitch placer. The automatic machines are

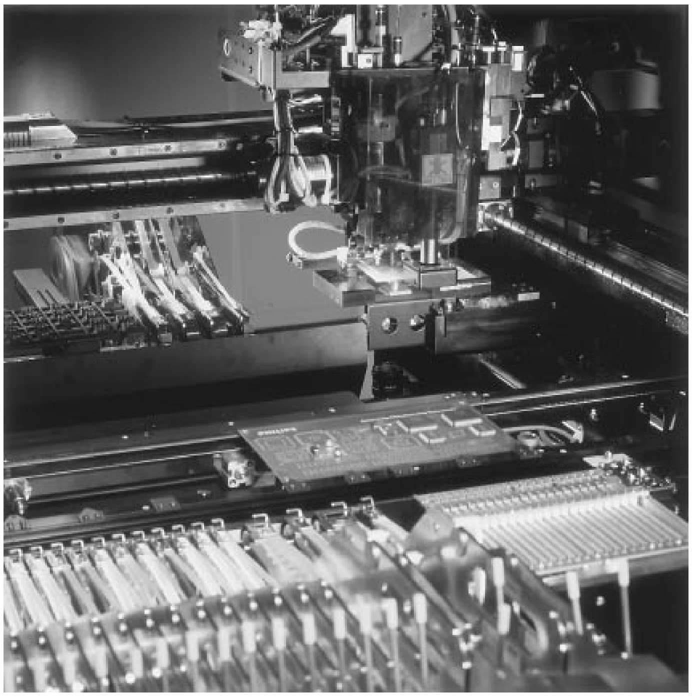

Figure 2. Example of placement mechanism of fine pitch equipment with flexible placement head. (Figure of Philips Eclipse courtesy of North American Philips.)

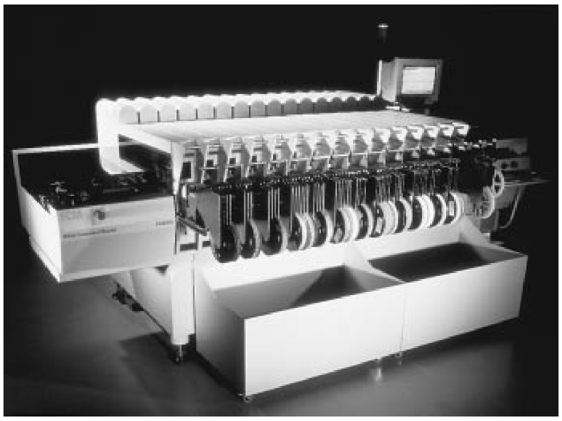

Figure 3. Example of high speed chip placement equipment with placement rates in excess of 60,000 devices per hour. (Figure of Philips FCM courtesy of North American Philips.)

Figure 4. Example of medium speed flexible placement machine with placement rates in excess of 14,000 devices per hour. (Figure of Philips Topaz courtesy of North American Philips.)

usually the most expensive to buy, but are frequently the most cost effective solution for placing parts.

Reflow/Curing Equipment

Once the parts have been placed on either adhesive or solder paste, it will be necessary to process the boards through an oven. In the case of adhesive, the oven cures the adhesive so that the parts will stay put through the rest of the process [see Fig. 6(a)]. For solder paste, the oven has to activate the flux and allow the flux to eliminate oxidation, and then melt the solder and allow it to form the solder joints between the board and the parts [see Fig. 6(b)]. In both cases, the oven temperature must be carefully controlled so as not to throw the boards and parts into thermal shock. Most surface mount parts can withstand temperature changes up to four degrees Celsius per second, but it is not recommended to exceed two degrees per second of heating or cooling.

The ovens come in two basic categories, batch and flow line. Batch ovens can only work on one group at a time. The flow line ovens are designed for the continuous introduction and removal of work. Each has its unique advantage.

Figure 5. Example of flexible placement machine. (Figure of Philips Eclipse courtesy of North American Philips.)

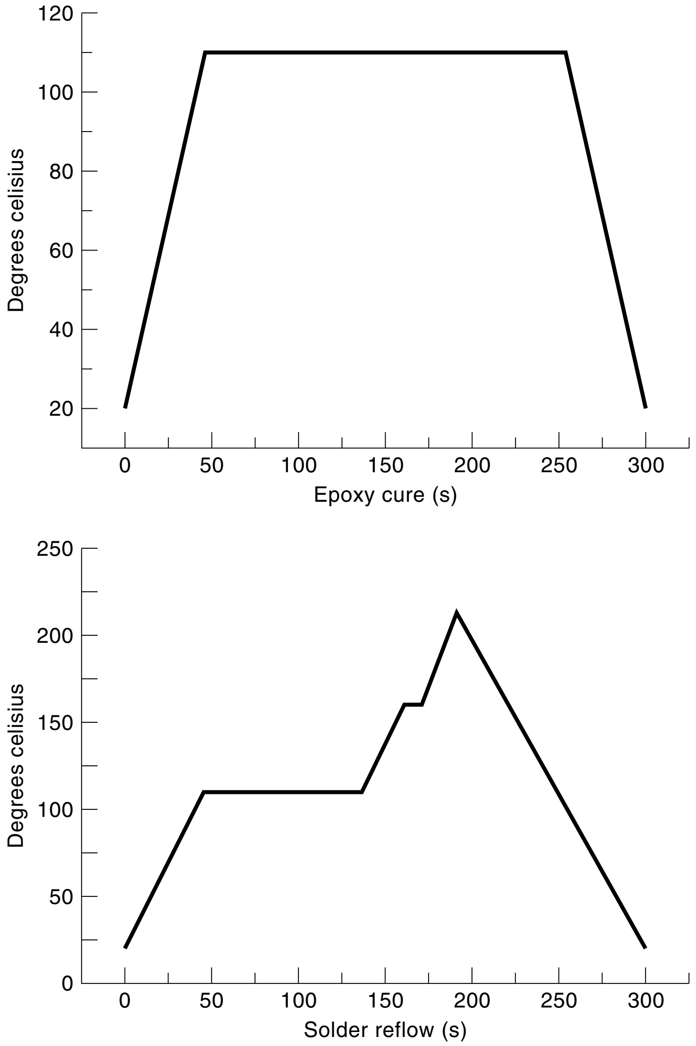

Figure 6.

Batch ovens typically have a sealed chamber to do their work. Because the chamber is sealed, the control of the environment is easy to maintain. This makes it easier to achieve the ideal profile for the work desired. Batch ovens expose the whole board to the same temperatures at the same time, so that the board is not subjected to two different extremes in temperature at the same time. These ovens are primarily for small batches of work and are frequently used in prototype environments where workflow may be erratic.

Flow line ovens are designed for the continuous introduction of boards into the process cavity. This means that the entry and exit points are not completely sealed. (Various techniques are implemented to achieve some degree of isolation.) The flowline ovens will have several zones to control how much energy is being put into the board to heat it up as it goes through the various phases. By controlling the speed that the board passes through these phases and the amount of energy applied in these zones, the desired profile can be achieved. Flow line ovens are typically used in production environments.

Regardless of the choice of batch or flow line oven, there are four main types of energy transfer that occur inside the oven. Infrared (IR), convection, vapor phase, and Ultraviolet (UV) are the main techniques.

Infrared ovens are used to cure adhesive and reflow solder paste. Infrared ovens rely on IR transmitters to input energy into the oven cavity. As this IR energy is absorbed, it will heat up the object and, through conduction, whatever the object is touching. The choice of IR was made because the green printed wiring boards (typical color) absorb the infrared light. This technique was very common several years ago, but as the process cavity, oven, became better sealed, the energy transfer method started becoming half convection and half IR. Due to the sensitivity of the IR ovens to the color of the substrate (prototypes are typically reddish in color), these ovens lost favor with a lot of people.

Convection ovens are used to cure adhesive and reflow solder paste. Convection ovens rely on hot gases to provide the transfer of energy to the substrate. The gases in a convection oven are either nitrogen or air. This transfer of energy is not dependent on the color of the substrate. Many different techniques are used to generate the heat and distribute it throughout the various parts of the oven. This technique for energy transfer is very stable and will work well in a wide variety of situations, making it a popular choice for ovens today.

Vapor phase ovens, both batch and continuous belt, are used to reflow solder paste. Vapor phase ovens rely on a combination of fluids (fluorocarbons) brought to boiling temperature. The combination will have two separate boiling temperatures, and each will be used and kept in place by cooling coils. This results in vapors at two separate temperatures. The substrates will then be passed (lowered in batch ovens) through these gases, and the gases will efficiently heat up anything that they come in contact with. The substrates will then be cooled (removed in batch ovens) slowly to allow the solder to gradually solidify. Vapor phase ovens are used primarily for high reliability (military) applications and design critical applications. The main reasons this technique is not more popular are the expense of the chemicals and the potential of these chemicals to damage the environment.

Ultraviolet ovens are used for curing adhesive. UV ovens rely on ultra-violet light to transfer the energy to the adhesive. There are several adhesives that may be used in electronics manufacturing that will only cure if exposed to strong UV light for a period of time. Consequently, there is a better control of the curing process, because it will not start without the UV energy. A second reason for using UV adhesives is that most substrates do not absorb UV light well, so this does not heat the substrate significantly.

Wave Solder Equipment

If through hole parts are used, it is necessary to solder the parts in place and form the electrical connections. The parts may be soldered by hand or, more commonly, by processing in a wave solder machine. A typical wave solder machine will consist of a fluxer, a preheated, and a solder wave. The fluxer will apply flux to the bottom of the substrate by spraying or foam. The only requirement is that the whole area be covered, so that the flux will contact all the surface area to be soldered. After the substrate is fluxed, it moves to the preheaters. The preheaters will activate the flux, so that the flux can remove oxidation from the components and the substrate. The preheaters also elevate the substrate temperature too, so that the solder temperatures will not provide a thermal shock when the substrate encounters the solder wave. When the soldering takes place, the substrate is moved across a wave of solder so that only a small line of solder is in contact with the substrate at once. Depending on the system, a second wave may also be used. The waves are generated by pumping molten solder into a pot and allowing it to spill over or are generated directly from a pump pushing the solder upward.

Cleaning Equipment

After assembly, it may be necessary to clean the substrate. Cleaning is required if the flux residue is corrosive or if the cosmetic appearance of the substrate is critical. The chemicals used will vary, depending on the type of flux used. The substrate will be exposed to various washing stages in order to remove all the corrosive material. Some of the newer fluxes available permit the elimination of this step.

Testing Equipment

After all the assembly steps are completed, it will be necessary to test the assembly. At this point, the assembly will be hooked up to various pieces of equipment to find out if it works and what is wrong with it. The testers may test each component separately (using a bed of nails to probe every node of the circuit) or use a functional test (plug it in and see if it works). All the nonconforming boards then get separated out and are reworked or scrapped.

Rework Equipment

If the assembly is to be repaired, an operator will remove the problem parts and replace them. The equipment to do this may be as simple as a soldering iron or complex enough to have robotic arms to help solder the parts and hold them in place. The choice is dependent on the parts to be replaced.

CONCLUSION

Surface mount assembly is a very dynamic field. The cost, size, and reliability advantage of surface mount implies that this technology will be around for some time. There are many publications that address this field, as well as many organizations. Additional information can be obtained from both IEEE/CPMT (3) and IMAPS (4). Web page information is provided in the references as a starting point for further investigation. There are so many different challenges to implementing surface mount that the authors are not providing details about research on one part of the process, like using cobalt composite solder, because there are too many areas being worked on. The recommendation is to investigate the publications and Web sites to find the latest information.

BIBLIOGRAPHY

1. G. Boothroyd, P. Dewhurst, and W. Knight, *Product Design for Manufacture and Assembly,* New York: Marcel Dekker, 1994.
2. K. Preston-White, W. J. Trybula, and R. N. Athay, Design for semiconductor manufacturing, *IEEE Trans. Compon. Packag. Manuf. Technol., Part C: Manufacturing,* **20** (1): 58–86, 1997.
3. IEEE/CPMT [online]. Available Website: http://www.cpmt.org
4. IMAPS [online]. Available Webside: http://www.imaps.org

Reading List

C. Capillo, *Surface Mount Technology,* New York: McGraw-Hill, 1990.

H. H. Manko, *Solders and Soldering,* New York: McGraw-Hill.

R. Prasad, *Surface Mount Technology,* Upper Saddle River, NJ: Prentice-Hall, 1997.

R. Tummala and E. Rymaszewski, *Microelectronics Packaging Handbook,* New York: Van Nostrand-Reinhold, 1989.

WALTER J. TRYBULA
SEMATECH

MICHAEL E. TRYBULA
Philips Broadband