The Application of PCB, Mounted-Components and Solder Paste in Surface Mount Technology Assembly (SMTA)

Noor Mariamadzliza Mohd Nan*, Ir. Assoc. Prof. Mohabattul Zaman Bukhari, Noor Azira Mohd Noor, Mohd Nazry Salleh.

School of Materials Engineering,
Northern Malaysia University College of Engineering (KUKUM),
Block B, Kompleks Pusat Pengajian KUKUM, Jalan Kangar-Arau,
02600 Jejawi, Perlis, Malaysia.
Email: maria@kukum.edu.my*

ABSTRACT Surface Mount Technology or SMT is a rapidly evolving field in the manufacture of electronic components, involving either active (transistors, integrated circuits, diodes, etc) or passive (capacitor, resistors, inductors, coils, etc) which do not have leaded connections. This technique which is SMT, replaces the old technique which the older package tended to use through-hole pins, which pass through holes in a printed circuit board and were soldered from below. The pins contributed inductance, and the size of the holes limits the density of the pins. SMT packages are soldered to the surface of a printed circuit board to alleviate this problem. These techniques are applied to the PCB, mounted components and solder paste. The comparison among the application of PCB, mounted components and solder paste in the SMTA will be shared and compared.

INTRODUCTION

The use of surface-mounted devices was the fastest growing development in the printed circuit board's industry, in the second half of the 1980s. Surface mounting is the technique of attaching components and devices only to the surface of the board. No holes or terminals are used in this process: only the board pads are soldered. If any plated-through holes are used, they serve as via-holes or interconnect-holes. The pad size corresponds to the foot-print of the surface mounted device (SMD) and these foot-prints are very small and allow a high density of component population.

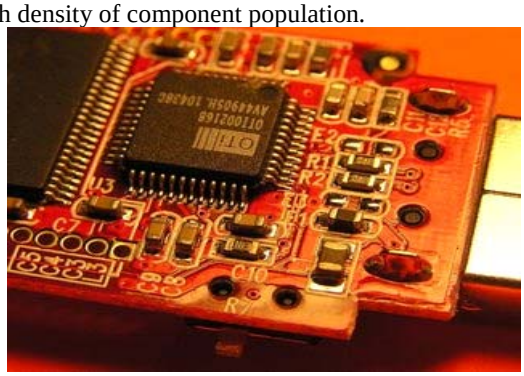

Figure 1: Surface-mount components on a keydrive's circuit board

In industry, SMT has largely replaced the previous construction method of fitting components with wire leads into the holes of the circuit board (also known as through-hole technology). Through-hole component, also spelled "thru-hole", is an electronic component that has pins designed to be inserted into holes and soldered to pads on a printed circuit board (PCB). The soldering operation can be performed using wave-soldering machine, a selective soldering system or a common solder iron. The components can be fixed in their positions before soldering by clinching the leads on the backside of the board. This was done in order to keep the components in their proper positions during transport, or to keep lighter components from floating up when they pass the solder-wave in a wave-soldering machine.

ADVANTAGES OF SMT

There are many undisputed advantages of the SMT over the older through-hole technique that propel the industry into this direction. Among those are:-

- Smaller components.
- No need to drill holes through abrasive boards.
- Simpler automated assembly.
- Minor errors during component placement will be corrected automatically (through the surface tension of the molten solder that pulls the component into alignment with the solder pads).
- Components can be fitted to both sides of the circuit board.
- Lower lead resistance and inductance (leading to better performance for high frequency parts).

TYPES OF SURFACE MOUNTED DEVICE

Several types of surface mounted devices are already being applied to the printed circuit boards. They can be divided into two major groups, which are the **leadless devices** and **leaded devices.** The leadless devices are devices in which the body of the component has metallized areas to serve as

in which the lead shape is conducive to a surface connection.

REASONS OF USING SMT

The driven force in the use of SMT are (each is important and is inter-related):-

- There will be cost reduction in materials and processes. This cost reduction stems from several factors, not just a reduction in size. The first factor is due to saving in devices in term of nano-size. No lead wires, lighter for shipping and less bulk to store.
- Due to simpler assembly and processing since SMT has ability to use less complex and cheaper boards for same density.
- With SMT, there would be increased density and speed in production. Semiconductor components can be mounted on top and bottom of a board, with little change. Their physical size also contributes to compactness and shorter delay line, thus increase the speed.
- With the increased density and reduction in size, there would be a reduction of interconnection too. In terms of reliability, the number of interconnections in a circuit always poses a problem. The fewer external joint produced at the user level, is the better. This is why there is such strong trend toward ever-larger scale of integration.

THE APPLICATION OF PRINTED CIRCUIT BOARD (PCB) IN SMTA

Printed circuits are found in virtually all electronics equipment manufactured in the last twenty years. Printed circuits are manufactured with use of printed circuits board and solder paste. Printed circuit boards are dielectric substrates with metallic circuitry photo-chemically formed upon substrate.

Dielectric Substrate in PCB

The dielectric substrate most commonly used is made from fiberglass sheets, which have copper foil, bonded onto both sides with epoxy resins. Other types are glass with polyimide, Teflon of trizine and paper covered with phenolic resins.

Metallic Circuitry Plating

Metallic Plating on PCB is the most crucial element in assuring a good and reliable circuitry in the component. Few types of plating were being used in the industry.

A printed circuit board (PCB) also known as printed wiring board (PWB) which interconnects electronic components without discrete wires. The “printed wires” which call “traces” are attached to a sheet of insulator. The vast majorities are manufactured by gluing a layer of copper foil over the entire substrate and the unwanted copper is then removed.

There are three general major classification of PCB that is being used in the industry of SMT which are single-sided boards, double-sided boards and multiplayer boards.

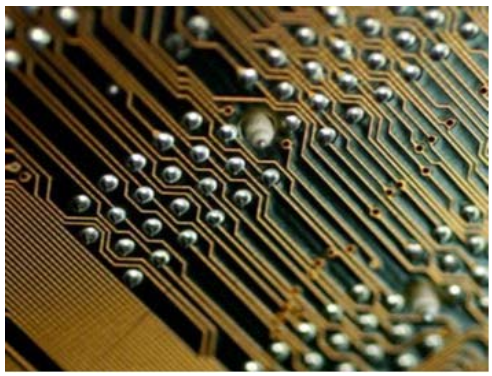

Figure 2: Close-up photo of one side of a motherboard PCB.

Figure 2 showing the close-up photo of one side of a motherboard PCB. The conductivity traces, vias and solder points for through-hole components on the opposite side are also shown (in Figure 2). The Printed Circuit Board (PCB) has traces and all the components are attached to the traces by soldering technique. Two or more of the layers are dedicated to provide ground and power which detune accidental antennas and provide efficient distribution of power.

Figure 3: The smaller components mount on single-side PCB.

Figure 3 shows the smaller components mount on single-sided PCB. The components smaller than leaded counterpart and has either short pins or flat contacts are soldered on the PCB board. They are designed to be handled by machines rather than by humans. It is easier to mount the components and look nicer than the through-hole technique. PCB in SMT is simpler automated assembly and small errors in component placement are corrected automatically. This is because the surface tension of the molten solder pulls the component into alignment with the solder pads. Besides, the component can be fitted to both sides of PCB compared to through-hole technique.

Another advantage of the PCB in SMT is that there is no need to drill holes through abrasive boards. It is a lower lead

resistance and inductance. It is leading to better performance for high frequency parts.

In other hand, PCB also applied in many field of industry such as aerospace, automobile, biomedical, materials development and etc. In industry, PCB in SMT has largely replaced the previous construction method of fitting components with wire leads into holes in the circuit board which also called through-hole technology.

THE APPLICATION OF MOUNTED COMPONENTS IN SMTA

Surface-mount components are usually much smaller than their leaded counterparts, and are designed to be handled by machines rather than by humans. It usually is smaller than an equivalent through-hole leaded device and, in some cases, less expensive. The pitch of the surface mount component was smaller than the pitch of through-hole components. Equipment and design engineers can save valuable wiring board area by mounting surface mounted leadless components on the underside of a board, and conventional leaded components on the top side of boards. Moreover, the components use less area on the top-side and do not use any area inside, or on the bottom-side, of the board.

Some of the more difficult hurdles to overcome with fine pitch surface mount assembly are:

1. Smaller apertures in solder-paste printing, requiring smaller powder sizes in the solder-paste and thinner screen's stencils to avoid clogging in the apertures.
2. Tighter process control in screen-printing and more careful choice of process materials and process parameters.
3. More difficult pick up and handling of the small components, and the need for better placement accuracy for the pick-and-place machine.

In the last three decades, the evolution of single chip packages depicts three primary paths for packaging ICs:

1. *Peripheral* : DIP to PLCC to QFP to fine pitch QFP
2. *Area array* : Ceramic and plastic PGA to BGA to fine pitch BGA
3. *Flip chip* : Ceramic flip chip to organic flip chip

The electronics industry has defined a collection of standard package shapes and sizes. The examples of SMT components are listed below[5]:

- Rectangular passive components (mostly resistors and capacitors):
  - 0201 - 0.02" × 0.01" (0.6 mm × 0.3 mm), two terminals
  - 0402 - 0.04" × 0.02" (1.0 mm × 0.5 mm), two terminals
  - 0603 - 0.06" × 0.03" (1.5 mm × 0.8 mm), two terminals
  - 0805 - 0.08" × 0.05" (2.0 mm × 1.3 mm), two terminals
  - 1206 - 0.12" × 0.06" (3.0 mm × 1.5 mm), two terminals

- Tantalum capacitors:
  - Size A (EIA 3216-18): 3.2 mm × 1.6 mm × 1.6 mm
  - Size B (EIA 3528-21): 3.5 mm × 2.8 mm × 1.9 mm
  - Size C (EIA 6032-28): 6.0 mm × 3.2 mm × 2.2 mm
  - Size D (EIA 7343-31): 7.3 mm × 4.3 mm × 2.4 mm
  - Size E (EIA 7343-43): 7.3 mm × 4.3 mm × 4.1 mm

- SOIC - small-outline integrated circuit, dual-in-line, 8 or more pins, gull-wing lead form, pin spacing 1.27 mm

- PLCC - plastic leaded chip carrier, square, J-lead, pin spacing 1.27 mm

- TSOP - thin small-outline package, thinner than SOIC with smaller pin spacing of 0.5 mm

- SSOP - shrink small-outline package, pin spacing of 0.635 mm

- QSOP - quarter-size small-outline package, with pin spacing of 0.635 mm

- VSOP - even smaller than QSOP; 0.4, 0.5 mm or 0.65 mm pin spacing

- SOT - small-outline transistor, with three terminals
  - SOT-23 - 3 mm × 1.75 mm × 1.3 mm body - three terminals for a transistor, or up to eight terminals for an integrated circuit
  - SOT-223 - 6.7 mm × 3.7 mm × 1.8 mm body - four terminals, one of which is a large heat-transfer pad

- PQFP - plastic quad flat-pack, a square with pins on all four sides, 44 or more pins

- CQFP - ceramic quad flat-pack, similar to PQFP

- TQFP - thin quad flat pack, a thinner version of PQFP

- BGA - ball grid array, with a square or rectangular array of solder balls on one surface, ball spacing typically 1.27 mm
- CGA - column grid array, circuit package in which the input and output points are high temperature solder cylinders or columns arranged in a grid pattern.

- CCGA - ceramic column grid array, circuit package in which the input and output points are high temperature solder cylinders or columns arranged in a grid pattern. The body of the component is ceramic.

- µBGA- micro-BGA, with ball spacing less than 1 mm

- Chip-on-board - a silicon chip containing an integrated circuit is supplied without the usual plastic encapsulation, and soldered directly to the board.

THE APPLICATION OF SOLDER PASTE IN SMTA

In the reflow soldering of surface mount assemblies, solder paste is used for the connection between the leads or terminations of surface mount components and the lands or PCB.

Solder paste is a homogeneous and kinetically stable mixture of solder alloy, flux and vehicles, which capable of forming metallurgical bonds at a set of soldering conditions and can be readily adapted to automated production in making reliable and constant solder joints.

Placing components onto a PCB for surface mount technology requires the application of solder paste. Stencil printing of solder paste is a critical step in the surface mount process. The quality of the print is directly correlated to the process yields. Any contamination, or dust, that is drawn into the stencil aperture directly affects the quality of printing. As the pitch and aperture get smaller for fine pitch technology, then the maximum stencil thickness becomes thinner, and the size of the particulate contamination becomes increasingly detrimental to the manufacturing process.

Application of adhesive paste by the needle must be precise, to ensure accurate positioning and that the appropriate quantity is applied. To apply the paste, pressure is passed down the nozzle. Once paste is released, a small backward suction can draw up any particulate on the board, or stencil, creating a blockage. The lines must then be stopped while the nozzle is changed. The removal of particulate, directly before printing, reduces the downtime experienced through blocked adhesive nozzles, as well as, reducing the frequent cost of replacement.

Using solder paste to build SMT assemblies that have leaded components is not something new. By using this technique we may find, as others have, that there are several advantages. Using solder paste for leaded components can simplify the design, improve component density, simplify the process, eliminate equipment, lower manufacturing costs and improve the quality and reliability of the finished product. To implement using solder paste for leaded components, it begins with the design. It is possible for many older designs to be adapted to this process. Using solder paste to its full advantage we can fill vias that are exposed. The vias may be test pads with open vias or plated holes that have leads of components protruding through them. For a complex, high density double-side mounted assembly; this process can have a significant impact. By elimination of wave soldering, we have many advantages. The component density can be much higher using solder paste. The component limitations of the types of components that we put on the wave solder side of the board are enhanced by using solder paste.

Solder paste consists of small solder spheres, flux and solvent. It is a thick gray paste with a viscosity close to that of toothpaste. The solder spheres are usually of the eutectic tin / lead alloy (63 % tin and 37 % lead). The pastes are separated into types 1, 2, 3 etc, as shown in Table 1. The higher numbers have smaller solder spheres and are more suitable for fine-pitch devices. Solder paste for stencil printing has a metal content or around 90 weight-percent (wt %). Note that the flux has a much lower density than the solder spheres, so the volume percentage of solder the paste is only about 50 %. The solder volume after re-flow is therefore mush less than the paste volume printed. Paste for dispensing has a lower metal content to make it flow better through a nozzle.

A minimum amount of paste is required on the stencil to keep the printing results consistent. As the solder paste is consumed, a dispenser inside the machine can add more automatically. This way, the machine can run continuously without operator interventions. However, if the paste is left on the stencil during longer stops, the viscosity changes due to the evaporation of the solvent, eventually leaving the paste dry and useless. The viscosity of the paste will increase when the printer is idle and decrease when in use. Viscosity is also temperature dependent; it decrease with rising temperature. The paste on printed boards that are not assembled soon loses tackiness, and components may fall off before re-flow soldering. Solder paste contains unhealthy materials, and the use of proper ventilation and protective gloves are absolutely necessary. Used solder paste must be handled appropriately and should be sent for recycling.

Table 1: Solder – paste types

Type	Solder Spheres Diameter	
	µm	mil
1	75 – 150	3 – 6
2	45 – 75	1.8 – 3
3	20 – 45	0.8 – 1.8
4	20 – 36	0.8 – 1.4
5	15 – 25	0.6 – 1
6	5 – 15	0.2 – 0.6

applied to all the solder pads (for connecting the surface-mount components) and the plated-through contacts (PCB holes for "Pin in Hole Intrusive Re-flow" insertion). Usually a screen printing process is used for this purpose. A squeegee moves across the PCB, which is masked with screens, and presses the solder paste into all the unmasked areas. To ensure that the plated-through holes are completely filled, significantly more solder paste must be applied than for the solder pads on the PCB surface. The required quantity can be set exactly via several parameters.

As an alternative to screen printing, the solder paste can be dispensed by means of a pipette. A high-precision robot moves the pipette to all the required positions on the PCB in succession. The dispensing method is particularly suitable for small PCBs or applications, which demand high precision and flexible dispensing volumes.

COMPARISONS BETWEEN PCB, MOUNTED-COMPONENTS AND SOLDER PASTE IN SURFACE MOUNT TECHNOLOGY ASSEMBLY (SMTA).

	PCB	**Mounted-Components**	**Solder Paste**
1) Definition	- A printed circuit board or **PCB** interconnects electronic components without discrete wires. Alternative names are printed wiring board or **PWB**. [(4)]	-Surface-mount components are usually much smaller than their leaded counterparts, and are designed to be handled by machines rather than by humans.[(2)]	- Solder paste is a mixture of solder alloy particles, flux and other materials for use in re-flow soldering (oven, vapor phase or infrared) of surface mount technology (SMT) components.
2) Application	- The vast majority is manufactured by gluing a layer of copper foil over the entire substrate and the unwanted copper is removed. The PCB has 2 types which are single layer and multi layer. (9)	- Equipment and design engineers can save valuable wiring board area by mounting surface mounted leadless components on the underside of a board, and conventional leaded components on the top side of boards[(13)]. Moreover, the components use less area on the topside and do not use any area inside, or on the bottom-side, of the board.	- Stencil printing of solder paste is a critical step in the surface mount process. The quality of the print is directly correlated to the process yields. Any contamination, or dust, that is drawn into the stencil aperture directly affects the quality of printing. - As the pitch and aperture get smaller for fine pitch technology, then the maximum stencil thickness becomes thinner, and the size of the particulate contamination becomes increasingly detrimental to the manufacturing process.[(1)]

3) Made of	- The simplest PCB is a layer of copper foil glued to a sheet of plastic (referred to as the substrate), often an epoxy glue reinforced with fiberglass. The circuit board is formed by removing foil to form the conductors (referred to as "traces") to which components are attached, usually by soldering. (5)	-Different components are applied with different types of techniques. For example in SMTA mounted components are PLCC,TSOP, SSOP,QSOP, VSOP,SOT and etc. (12)	- Solder preforms are generally made from solder alloy wire or stamped from solder alloy sheet material, and formed into specific shapes (typically toroids, washers or donuts) for use in reflow soldering (oven, vapor phase or infrared) of Plated Through Hole (PTH) components and some Surface Mount Technology (SMT) components.()
4) Advantages	- Simpler automated assembly and small errors in component placement are corrected automatically. (7) - The component can be fitted to both sides of PCB compared to through-hole technique. (7) -No need to drill holes through abrasive boards.*10 It is a lower lead resistance and inductance. It is leading to better performance for high frequency parts.*11	-the components use less area on the topside and do not use any area inside, or on the bottom-side, of the board. (6) - can save valuable wiring board area by mounting surface mounted leadless components on the underside of a board, and conventional leaded components on the top side of boards(3)	- Using solder paste for leaded components can simplify the design, improve component density, simplify the process, eliminate equipment, lower manufacturing costs and improve the quality and reliability of the finished product. - Using solder paste to its full advantage we can fill vias that are exposed. The vias may be test pads with open vias or plated holes that have leads of components protruding through them.(8)

CONCLUSION

The implementation of the surface mounted technology is not simple. Few reliability data have been accumulated to define its use at the higher end of the printed circuit industry. It must be used with new and more costly laminates, specialized assembly technique, and so on.

Therefore, this technology is expected to grow in the areas involving mass production at lower end of the industry. Surface mounting of devices for mass production is already underway. The use of SMT in high technology applications like computer is undeniable. These can justify the increased investment because of the circuit speeds gained.

Surface mount assembly is done by stencil- printing solder paste to a board, placing the component on the board and then heating the entire assembly, so that the solder melts and forms solder joints. By SMT, the benefits are:

- Reduction in package size resulting in greater functionality in the same board area.
- Reduction in weight. Mobile and handheld electronic items such as video camera, cellular telephone are examples which have low weight and high performance.
- Reduction in noise, this is primarily due to smaller electrical paths compared to leaded components. This feature is very useful in RF and microwave circuits
- where low noise contribution is mandatory and is a design feature.
- higher operating speed resulting from shorter interconnect distances.

REFERENCES

1. Manko, H.H (1986) Soldering Handbook for Printed Circuits and Surface Mounting, New York, Van Norstrand Reinhold.

2. Clark, R.H. (1985) Handbook of Printed Circuit Manufacturing, New York Van Norstrand Reinhold.

3. Hwang, J.S (1992) Solder Paste in Electronics Packaging: Technology and Application in Surface Mount, Hybrid circuits, and component Assembly, New York, Van Norstrand Reinhold.

4. Colin C. Johnson, and Joseph Kevra "Solder Paste Technology Principles and Application", Division of TAB Books Inc., Blue Ridge Summit, PA 17294 0850.

5. Tummala, R.R. (Fundamentals of Microsystems Packaging, McGraw-Hill Book Company (2001).