1

Lead-Free Surface Mount Technology

Jennifer Nguyen[1] and Jasbir Bath[2]

[1] *Flex, Milpitas, California, USA*
[2] *Bath Consultancy LLC, San Ramon, CA, USA*

1.1 Introduction

Surface mount technology (SMT) involves the assembly or attachment of surface mount devices (SMDs) onto the printed circuit board (PCB). Today, the majority of the products are built using surface mount technology and lead-free process. This chapter will review the surface mount process for lead-free soldering, including printing, component placement, reflow, inspection, and test. The chapter also discusses some advanced miniaturization technologies used in the SMT process.

1.2 Lead-Free Solder Paste Alloys

Today, there are a variety of lead-free solder paste alloys available in the market. SnAgCu (SAC) materials with 3.0–4.0% Ag and 0.5–0.9% Cu and remainder Sn are widely accepted within the industry. Among them, Sn3.0Ag0.5Cu (SAC305) is still the most common alloy used in the SMT process. These SnAgCu alloys have the liquidus temperature of around 217 °C. As the cost of Ag has increased over the past years, the use of low Ag alloy materials such as Sn0.3-1.0AgCu or SnCu/SnCuNi has increased. These alloys have approximately 10 °C higher melting temperature than SAC305 and may need to be processed at slightly higher temperature during the reflow process.

Low temperature lead-free alloys which contain SnBi/SnBiAg are also used. These alloys have melting temperature around 140 °C and can be processed at 170–190 °C. These low temperature alloys usually have high bismuth content and they create some reliability concerns, especially on mechanical reliability. These

Lead-free Soldering Process Development and Reliability, First Edition. Edited by Jasbir Bath.

low temperature alloys are used on certain applications such as light-emitting diode (LED)/TV products. In recent years, there is a desire for low temperature lead-free alloy alternatives with better reliability. The drivers for these low temperature alloys include component warpage, low energy consumption, and component or board sensitivity to the higher temperature lead-free process. These alloys typically have higher liquidus temperature than traditional SnBi/SnBiAg alloys, but they still have lower liquidus temperature than SAC305. These alloys have gained a lot of interest in the industry in the recent years, and some are available in the market and used in production.

1.3 Solder Paste Printing

1.3.1 Introduction

One of the most important processes of the surface mount assembly is the application of solder paste to the PCB. This process must accurately deposit the correct amount of solder paste onto each of the pads to be soldered. Screen-printing the solder paste through a foil or stencil is the most commonly used technique, although other technique such as jet printing is also used.

There is no major change to solder paste printing for lead-free processes. The same printer can be used for tin-lead and lead-free printing. In general, the same stencil design guidelines can be used for lead-free process.

1.3.2 Key Paste Printing Elements

Solder paste printing process is one of the most important processes in surface mount technology. This process can account for the majority of the assembly defects if it is not controlled properly. For effective solder paste printing, the following key factors need to be optimized and controlled:

- PCB support
- Squeegee (type, speed, pressure, angle)
- Stencil (thickness, aperture, cleanliness, snap off, separation speed)
- Solder paste (including type, viscosity)

PCB support is important to the printing process. Good PCB support holds the PCB flat against the stencil during the screen-printing process. PCB support is generally provided with the screen-printing machines. If the board is not properly supported, solder defects such as bridging, insufficient solder, and solder smearing can be seen. For fine pitch printing such 0.3/0.4 mm pitch chip scale package (CSP), 0201/01005 (Imperial) chip component, a dedicated custom-made fixture for printing or vacuum support should be used.

Squeegees, squeegee pressure, and speed are other critical parameters in the screen-printing process. Metal squeegees are commonly used for printing solder paste, and rubber or polyurethane squeegees are used for epoxy printing. A squeegee angle of 60 °C to the stencil is typically used [1]. Squeegee speed and squeegee pressure are critical for good printing. The speed of the squeegee determines how much time the solder paste can roll and settle into the apertures of the stencil and onto the pads of the PCB. In the beginning of lead-free conversion, a slower printing speed was used because the lead-free solder paste was stickier than tin-lead solder paste. Today, many lead-free solder pastes can print well at high speed.

The speed setting is widely varied from a typical range of 20–100 mm/s^{-1} depending on the size of the aperture, the size of PCB, and the quantity of boards being assembled, etc. Printing speed used depends on the solder paste supplier or is optimized by a Design of Experiment (DOE). It is typically between 40 and 80 mm s^{-1}. During the solder paste printing, it is important to apply sufficient squeegee pressure and this pressure should be evenly distributed across the entire squeegees. Too little pressure can cause incomplete solder paste transfer to the PCB or paste smearing. Too much pressure can cause the paste to squeeze between the stencil and the pad.

Stencil is another key factor in the solder paste printing. Metal stencils are used in solder paste printing. Stainless steel material is commonly used; however, metal stencils can be made of copper, bronze, or nickel [2]. There are several types of screen-printing stencil, including chemical etch, laser cut, and electroformed [2]. The thickness of the stencil is typically 125 μm (5 mil) or 150 μm (6 mil). Stencils with the thickness of 100 μm (4 mil) or thinner have become more popular with the high density and fine pitch components such as 0201/01005 (Imperial) chip components or 0.4/0.3 mm pitch CSP or quad flat no-leads/bottom termination component (QFN/BTC) components. Thicker stencils than 150 μm are typically used when more paste is needed. Stencil thickness and aperture size determine the amount of paste deposited on the pad. In general, stencil aperture must be three times and preferably five times the diameter of the solder particles. To ensure the proper paste release and efficient printing, the aspect ratio should be greater than 1.5, and the area ratio should be greater 0.66.

The aspect ratio is defined by Eq. (1.1), and the area ratio is shown in Eq. (1.2).

$$\text{Aspect ratio} = \text{Aperture Width/Stencil Thickness} \tag{1.1}$$

$$\text{Area Ratio} = \text{Area of the Aperture } (\text{L} \times \text{W})/\text{Area of Aperture Walls } (2 \times (\text{L} + \text{W}) \times \text{T}) \tag{1.2}$$

Snap off and stencil separation speed are also important for good printing quality. Snap off is the distance between the stencil and the PCB. For metal stencil

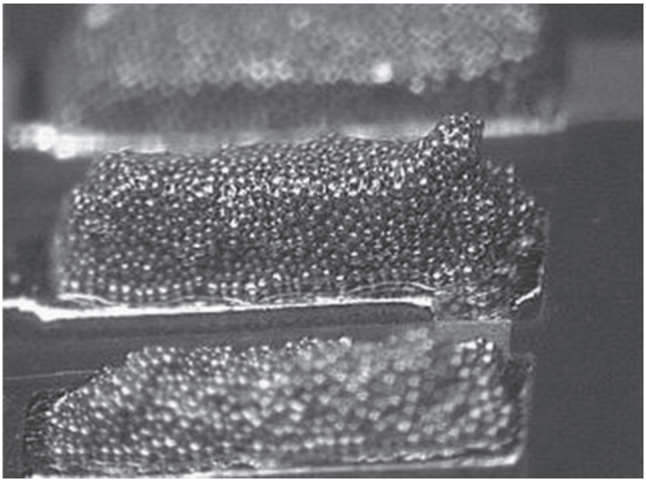

Figure 1.1 Example of tailing at the edge of the paste due to high separation speed.

printing, the snap off should be zero. This is also called contact printing. A high snap off will result in a thicker layer of solder paste. Stencil separation speed is the speed of separation between the stencil and PCB after printing. Traditionally, high separation speed will result in clogging of the stencil apertures or tailing at edges around the solder paste deposited (Figure 1.1). However, lead-free pastes tend to have a higher adherence than tin-lead pastes and may prefer high separation speed than tin-lead solder paste. Separation speed varies depending on the solder pastes and its supplier, and the supplier's recommendation should generally be followed.

Last but not least, the correct solder paste type and material should be used. The correct type of solder paste should be selected based upon the size of the apertures within the stencil. Type 3 was commonly used in the tin-lead process; however, Type 4 has become a more common lead-free solder paste type in the recent years due to the increase in miniaturized components on the printed circuit board. The

Table 1.1 General solder paste type and particle sizes.

Paste type	Particle size (μm)
3	45–25
4	38–20
5	25–15
6	15–5

release from the apertures of the stencil is affected by the particle size within the selected solder paste. Table 1.1 lists the particle size of different solder paste type.

Both tin-lead and lead-free solder paste should be refrigerated while being stored to maintain its shelf life but must be brought to room temperature before use to maintain quality. Some new lead-free solder pastes require no refrigeration and can be stored at room temperature. The solder paste should be mixed properly before use to ensure even distribution of any separated material throughout the paste. It is recommended to follow the solder paste manufacturer's recommendations for storage and handling conditions.

1.4 Component Placement

1.4.1 Introduction

After the correct amount of solder paste is applied, components are placed on the PCB at the specific locations. The component placement process includes board loading and registration, fiducial vision alignment, component pick-up, component inspection, and alignment and placement. The component placement must be precise and in accordance with the schematics. Pick and place machines are used in this process. There are different types of pick and placement machine available in the market. Some machines are designed specifically for speed whereas others are more focused on flexibility. The machines designed for speed are generally referred to as "chip shooters" and can achieve component placement rates of up to 100 000 cph (components per hour). The flexible pick and place machine can handle components ranging from 01005 (Imperial) chips to large components such as ball grid arrays (BGAs), connectors, etc. Flexible machines typically have slower pick and place speed than the chip shooter. The machines are selected depending on the types, sizes, and volumes of the surface mount components. The same pick and place equipment can be used for tin-lead and lead-free components.

1.4.2 Key Placement Parameters

Component placement is an important factor in surface mount assembly. It affects not only the assembly time but also the reliability of the solder joint. Placement accuracy and placement speed are critical in this process. To achieve accurate placement and high output, the following factors need to be considered:

- Nozzle
- Vision system
- PCB support
- Component size, packaging
- Feeder capacity

1.4.2.1 Nozzle

It is very important that the correct nozzle be selected for each different part to be placed to ensure accurate and consistent placement. There are many different types of nozzle for pick and place components. Most nozzles use a vacuum to hold the components. For handling small components, positive pressure is often supplied in addition to vacuum at the moment of placement so that the component would be completely released from the nozzle. Component flatness at the top surface is important for the pick and place process. Certain components such as connectors that do not have a flat top surface can have a pick-up pad inserted or pre-attached by the supplier for pick and place purposes. Some alternative nozzles have a gripper, which grips the component sides instead. The gripper is typically for placing some odd-shaped components. However, the placement speed is typically slower as compared to the nozzles that hold the component by vacuum. In addition, extra space is required between the components to accommodate the grippers.

1.4.2.2 Vision System

The vision system inspects every component before placement. It checks the part dimensions and any component damage before placement. It is important to program each component with the correct tolerance parameters to allow the machine to determine if an incorrect part has been loaded and also not to reject acceptable components.

1.4.2.3 PCB Support

The PCB needs to have adequate support during component placement. Improper PCB support can cause component misalignment or missing components.

1.4.2.4 Component Size, Packaging, and Feeder Capacity

The surface mount components on the PCB will differ in size. It is common to have small components positioned close to large components in high density design. All small components need to be placed before larger components so that the larger components do not get disturbed and misaligned during placement.

The surface mount components are supplied in different ways. The most common component packages are tape and reel, tubes, and trays.

1.4.2.5 Feeder Capacity

Feeders are used to feed components to a fixed location for the pick-up mechanism. Feeder types include tape and reel feeder, matrix tray feeder, bulk feeder, and tube feeder. The tape and reel feeders come in different sizes and are the most common feeder for placing large quantities of small components. The number of tape feeders that can be loaded into the machine at a time will play an important role in determining the speed of component placement. The matrix tray feeders are typically used for large and/or expensive components such as BGA or QFN/BTC components. The tray holds the components securely without damaging the body or leads. However, the pick and place process for the tray feeder is often slower than the tape feeder.

1.5 Reflow Process

1.5.1 Introduction

In reflow soldering, the solder paste and solder balls for the case of a BGA component must be heated sufficiently above its melting point and become completely molten, in order to form reliable joints. In the case of components with leads, the solder paste must wet the plating on component leads to form the desired heel and toe fillets.

There is no one best reflow profile for all board assemblies. Ideally, a reflow profile must be characterized for each board assembly using thermocouples at multiple locations on and around the component devices and board. The solder paste type, component, and board thermal sensitivity must be considered in reflow profile development.

Lead-free solders typically process at higher temperature than tin-lead solder due to the high melting temperature of typical lead-free solders. Lead-free solder such as SAC305 (Sn3Ag0.5Cu) have an initial melting point of 217 °C and a final melting point of 220 °C. Lead-free reflow typically has a narrower process window than tin-lead reflow due to the component or board maximum temperature limitations.

1.5.2 Key Parameters

Solder joint formation depends on temperature and time during reflow. There are four phases of a reflow process, including preheat, soak, reflow, and cooling. In addition, reflow atmosphere plays an important role in the reflow process. The key parameters for reflow will be discussed in the following sections.

1.5.2.1 Preheat

The preheat phase prepares the PCB and components for actual reflow. It helps to reduce the thermal shock and temperature difference between the PCB and components and reflow temperature. A quick ramp rate during the preheat can damage the component. In general, a ramp rate between 1.0 and 3.0 $°C/s^{-1}$ is recommended, and the temperature change should be evenly distributed throughout the PCB. Preheat also removes some flux volatiles and prepares the solder paste material for reflow.

1.5.2.2 Soak

Soak is also known as the pre-reflow phase. In this phase, the flux in the solder paste gets activated, and this helps to remove oxidation on the component leads, PCB pads or on the solder particles' surface. Also, soaking phase allows the thermal gradient across the PCB to equilibrate prior to reflow. In this way, the entire assembly sees nearly the same reflow conditions to form consistent solder bonds. For large boards or boards with a large range of component sizes, a longer soak time is usually helpful to achieve successful assembly to help ensure the delta Temperature across the board is reduced. Soak profiles are also used to minimize voiding when assembling such components as BGA, land grid array (LGA), and QFN/BTC.

1.5.2.3 Reflow

As the solder reaches the solder melting temperature, the board enters the reflow phase. Peak temperature and time above liquidus temperature are important factors in this phase. The peak temperature is generally 20–30 °C above the liquidus temperature of the alloy, and reflow time is typically 30–90 seconds in order to form a good solder joint and proper intermetallic formation at the interfaces.

A typical reflow profile chart is shown in Figure 1.2, and typical profile parameters are listed in Table 1.2.

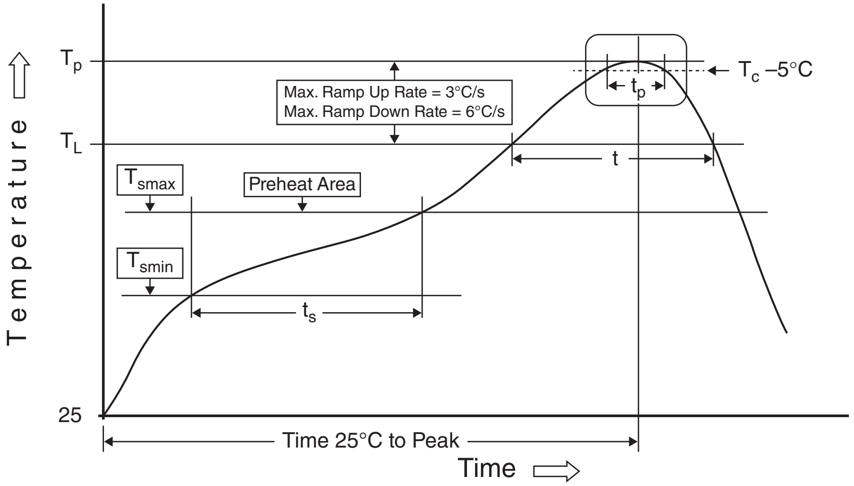

Figure 1.2 Reflow profile chart [3]. Copyright 2019 by IPC International, Inc. and is used with IPC's permission. This image may not be altered or further reproduced without the prior written consent of IPC.

Table 1.2 Typical lead-free profile parameters for lead-free.

Parameters	Typical lead-free profile
Preheat ramp rate	1–3 °C s^{-1}
Preheat and soak temperature range	110–210 °C
Preheat and soak time	60–180 s
Reflow time	30–90 s
Peak temperature	235–255 °C (for alloy liquidus temperature of ~217 °C)
Cooling rate	~3–4 °C s^{-1}

1.5.2.4 Cooling

Cooling affects the grain structure of the solder joint. Fast cooling rate results in fine grain structure which is assumed to have a more reliable solder joint and bond. However, too fast a cooling rate can exert thermal stress on the solder joint, which can result in fractures or tears on the solder joint. In general, cooling rate should not exceed 3–4 °C s^{-1}.

1.5.2.5 Reflow Atmosphere

Lead-free reflow can be done in both air and nitrogen environment. Reflow soldering in an inert atmosphere such as nitrogen reduces the solder oxidation during reflow and results in better wetting and appearance of the solder joint. However, nitrogen adds additional cost to the reflow process. Today, most lead-free reflow in manufacturing can be done in an air environment. For fine pitch components and some advanced assembly such as flip chip assembly and package on package (PoP) components, a nitrogen atmosphere is recommended

1.6 Vacuum Soldering

Lead-free soldering results in more voiding in the component solder joint as compared to tin-lead soldering. In addition, the assembly of BTCs generates more voids at the thermal pad solder joints and causes some quality and reliability concerns. Reflow soldering using vacuum or vacuum soldering becomes an attractive solution. Vacuum soldering uses gases and pressure to create a vacuum environment for reflow. Vacuum soldering helps to eliminate or minimize voiding

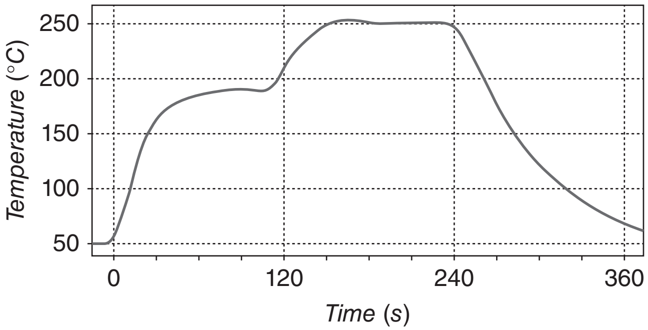

Figure 1.3 Example of lead-free profile using a vacuum reflow oven.

to a few percent. Hydrogen atmosphere is also used. Vacuum soldering is not a new technique and it has been used for a long time for various applications such as brazing (hard soldering). The use of vacuum in soldering electronics has increased in recent years.

Although vacuum soldering can significantly reduce voids for most electronics components, vacuum soldering has its own concern. Vacuum soldering is typically a batch process which can limit the throughput of the assembly. However, some in-line vacuum soldering equipment is also available. In addition, vacuum soldering requires a higher temperature rising rate and a longer reflow time which generates some concerns regarding the components and solder joint reliability. An example of a lead-free profile generated using vacuum soldering is shown in Figure 1.3.

1.7 Paste in Hole

Paste-in-hole or pin-in-paste technology is also known as through-hole reflow. Paste in hole is a method for inserting and soldering plated-through-hole (PTH) components during the surface mount process. The process involves screen-printing in and around PTH barrels, inserting PTH components after SMT placement, and reflowing the solder paste in the reflow oven to form the SMT and PTH solder joints simultaneously. Paste-in-hole reflow eliminates the wave solder process on selected PTH components and may reduce manufacturing costs and improve cycle times.

However, there are some restrictions with the paste-in-hole process. First, the plated-through hole component must be able to survive the reflow process temperature. Second, all PTH components for pin-in-paste process must be on one side of the PCB. In addition, the pin and paste process typically works well with only thin boards. The board thickness for pin-in-paste assembly should be less than 2.4 mm (0.093″), and the ideal thickness for this process should be ≤1.6 mm (0.062″).

A large amount of solder paste is required to fill the hole for a larger board thickness. When the pin-in-paste technology is used for thicker printed circuit boards (≥2.4 mm), an overprinted stencil aperture and/or solder preform can be used to be able to achieve good hole fill for PTH components.

1.8 Robotic Soldering

Robotic soldering is a soldering method which uses robots to solder surface mount or pin through hole components onto the printed circuit board. A soldering robot is an automated system that performs soldering tasks with precision and repeatability. Robotic soldering can help to eliminate human error from manual soldering and help to improve throughput and yield. Robotic soldering is typically used for point to point soldering applications, but it is also available for line soldering applications. Robotic soldering is used in many industries, such as Automotive Electronics and Solar Modules, Aerospace, Military and Medical. An example of a soldering robot is shown in Figure 1.4. Both benchtop and in-line robotic soldering systems are available in the market.

Figure 1.4 Example of a soldering robot. Source: Courtesy of Japan Unix. https://www.japanunix.com/en/products/automation.

1.9 Advanced Technologies

Advanced surface mount technologies relate to advanced packaging and product miniaturization. Miniaturization is the trend in surface mount technology today. Both PCB and component sizes are getting smaller and smaller while the number of signal inputs and outputs is increasing along with better performance expected.

The following sections will discuss some advanced technologies that are getting popular in surface mount technology such as Flip Chip and PoP.

1.9.1 Flip Chip

Flip chip assembly is the direct connection of active die onto printed circuit boards, regardless of interconnect material and method. In typical component packaging, the interconnection between the die and the carrier is made using wire. The die is attached to the carrier face up. A wire is bonded first to the die, then looped and bonded to the carrier. In contrast, the interconnection between the die and carrier in flip chip packaging is made through a conductive bump that is placed directly on the die surface. The bumped die is then "flipped over" and placed face down, with the bumps connecting to the carrier directly. A bump is typically 60–100 μm high, and 80–125 μm in diameter. The flip chip connection is formed by using solder or conductive adhesive. By far the most common flip chip interconnect method is solder, and will be discussed in the next section.

The flip chip assembly to the board typically requires no paste. Instead the bumped flip chip is dipped in flux before placing and reflow. Flux is used to hold the die in place during the assembly and to help remove oxide from the pads and component bumps during the reflow process. Nitrogen atmosphere is required for flip chip assembly. The nitrogen prevents oxidation during reflow and promotes good wetting of the pads. The bare die is typically underfilled for protection and enhancement of thermal and mechanical reliability.

1.9.2 Package on Package

PoP is a method to solder two or more BGA packages vertically. This allows higher component density in devices. This component packaging is commonly used in consumer products such as mobile phones, digital cameras, etc. An example of package on package is shown in Figure 1.5.

Package on package technology provides many benefits. First, PoP uses less PCB real estate of the motherboard. Second, PoP results in better electrical performance of devices because PoP shortens the length between different interoperating parts,

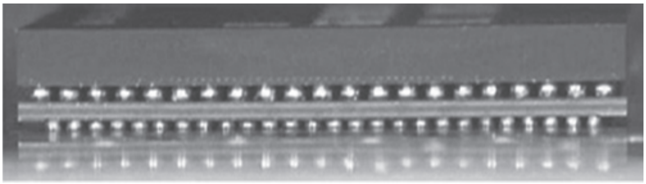

Figure 1.5 Package on package (PoP) component.

such as the controller and the memory. Shorter routing between the circuits yields faster signals and reduces noise and cross talk. In addition, PoP allows the memory package to be separate from the logic packaging. The user can test these components separately, change the components based on the product requirements or select different suppliers for the package.

Package on package can be processed using standard surface mount machines equipped with a dip-fluxing unit. The bottom package, along with the other SMT components on the board, is placed first. The top part is then dipped in flux (or solder paste) and placed. The flux (or paste) material must be carefully selected, and the flux amount and thickness must be controlled to get a high yield. Nitrogen is highly recommended for better yield.

1.10 Inspection

1.10.1 Solder Paste Inspection (SPI)

Since most PCB solder joint defects relate to the solder paste printing process, it is critical to have a good solder paste printing inspection method. In the beginning, solder paste height was manually inspected and tracked for solder paste printing control. However, solder paste volume was then shown to link to the solder joint quality and reliability, and thus it needs to be monitored closely. Automatic solder paste inspection systems were then developed to address the need for manufacturers to be able to monitor solder paste alignment and volume during the printing process. Solder paste inspection can be 2D or 3D inspection. 2D solder paste inspection checks the area of the paste deposit only while 3D inspection captures the height of the solder paste and enables the equipment to accurately measure the total volume of the paste deposited. The 3D Solder Paste Inspection (SPI) is commonly used to monitor and control the paste printing process and helps to correct any defects that are caused by solder printing early in the assembly process before the solder joint is formed. An example of 3D solder paste inspection image is shown in Figure 1.6.

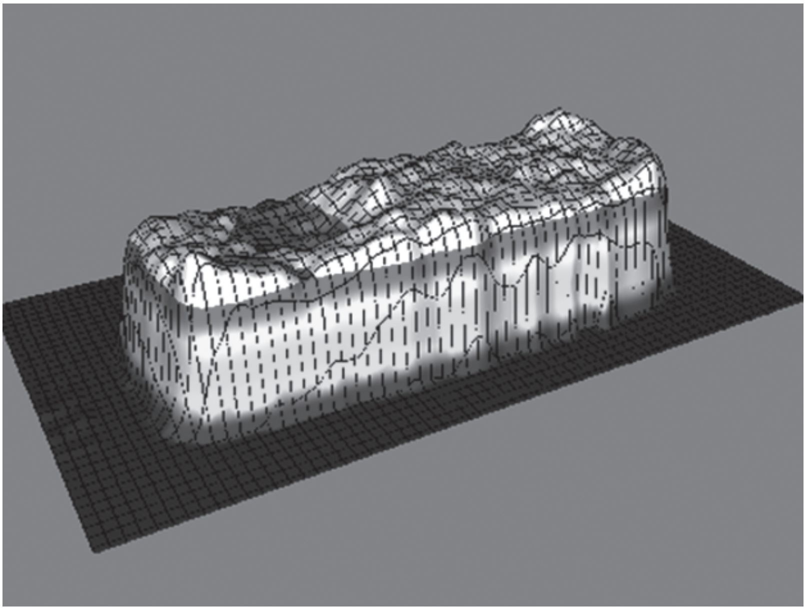

Figure 1.6 Solder paste inspection image. Source: Courtesy of Koh Young America, Inc.

1.10.2 Solder Joint Inspection

Solder joint inspection is necessary to ensure the quality and reliability of the solder joint. There are different ways to inspect the solder joint such as visual inspection (by the human eye or by using optical devices (magnifying glass, microscope, etc.)). Some solder joints are hidden under the components and cannot be visually inspected and need to be X-rayed for inspection of assembly defects. For example, the solder joints of BGA or BTC components are inspected using this technique. Automated optical inspection (AOI) and automated X-ray inspection (AXI) are common systems used in manufacturing to ensure the quality and reliability of the solder joint.

1.10.2.1 Automated Optical Inspection (AOI)

AOI is an automated visual inspection system used in the SMT process to identify failure and quality defects such as wrong component placement, missing component, component misalignment, bridging, missing solder, etc. Automated optical inspection can be done in line or off line, and it can be used pre or post reflow process. AOI is much more reliable and repeatable than manual visual inspection. AOI uses camera and machine vision systems to provide images for defect analysis. The surface of the board is visually scanned using several light sources and high definition cameras to create the picture of the board. The captured image is then

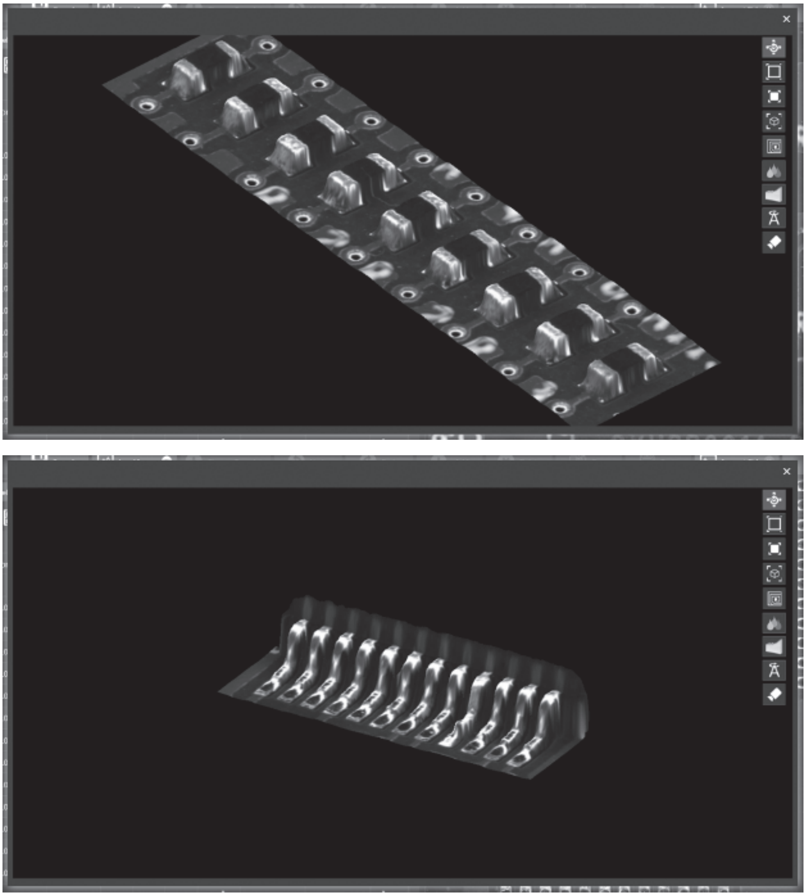

Figure 1.7 Example AOI inspection images of lead-free soldered chip (top) and lead-frame components (bottom). Source: Courtesy of Cyberoptics Corporation.

processed and compared with the knowledge the machine has of what it should look like. Using this comparison, the AOI system can detect defects and generate a report. A golden good board or known status board and design information is needed for generating the database of what the solder joint should be. Examples of AOI images for lead-free soldered chip and lead-frame components are shown in Figure 1.7.

1.10.2.2 X-ray Inspection

Automatic optical inspection works well for printed circuit boards where the solder joints are visible. When the solder joint is under the package and hidden such as in the case of BGA, CSP, and BTC components, X-ray inspection is

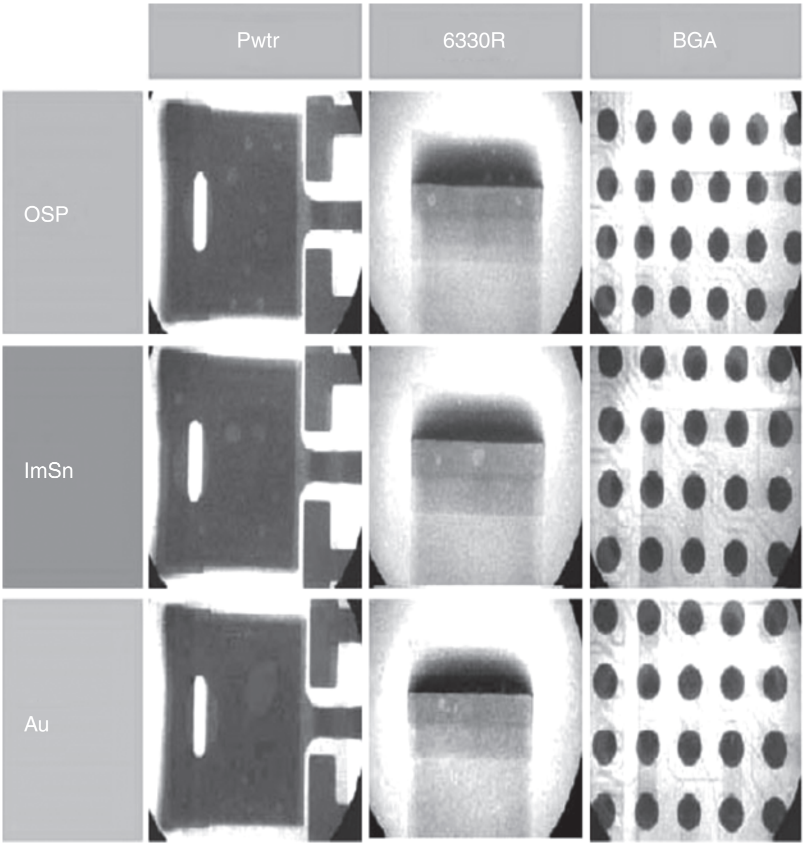

Figure 1.8 Example of power transistor/BTC, chip, and lead-free Sn3Ag0.5Cu BGA component solder joints soldered with lead-free Sn0.7Cu1.1Ag0.06Ni1.8Bi solder for three different board surface finishes (organic solderability preservative, immersion tin, NiAu) under X-ray inspection [4].

needed. X-ray inspection can also reveal other solder joint defects such as voiding or head-on-pillow (HoP). Manual or automated x-ray inspection is used. AXI is typically used in the production environment. Example images of X-ray inspection for lead-free soldered Power Transistor/BTC, 6330 chip and BGA components are shown in Figure 1.8 for three different board surface finishes.

1.11 Conclusions

In general, surface mount technology for lead-free soldering is similar to tin-lead soldering. Lead-free soldering requires higher process temperatures and has a narrower process window than tin-lead. Therefore, the process parameters should be closely controlled and optimized for high yield.

Besides traditional pick and place and reflow methods, the surface mount components can also be assembled using some alternative methods such robotic soldering or vacuum soldering. As the products get smaller while the performance is increased, the use of advanced miniaturized technologies such as fine pitch components (0.3/0.4 mm pitch component), smaller chip components (0201/01005 Imperial), PoP components and flip chip component has increased with surface mount technology becoming the major and dominant technology to assemble most electronics components today.

References

1 Mohanty, R., Claiborne, B. and Andres, F. (2010) Effect of squeegee blade on solder paste print quality. SMTnet, 17 June. https://smtnet.com/library/index.cfm?fuseaction=view_article&article_id=1630\ignorespaces2010 (accessed 3 February 2020).

2 Gopal, S., Rohani, J., Yusof, S., and Bakar, Z. (2006). Optimization of solder paste printing parameters using design of experiments. *Journal Teknologi* 43 (A): 11–20.

3 IPC/JEDEC J-STD-020E (2015) *Moisture/reflow sensitivity classification for non-hermetic surface mount devices*. January 2015. IPC International.

4 Nakatsuma, M., Wada, T., Mori, K. et al. (Koki Company Limited) (2013) A study of lead-free low silver solder alloys with nickel additions. SMTAI conference.