Reflow Soldering: Meeting the SMT Challenge

by

Dr. Ning-Cheng Lee
Indium Corporation of America
Utica, NY, USA
Tel: 315-853-4900 Fax: 315-853-4320

ABSTRACT

Reflow soldering of solder paste is the primary interconnection method used in SMT assembly process. The major issues which plague the reflow soldering performance include, but not limited to, bottom-side-component-holding, bridging, dewetting, low-residue, opening, solder balling, solder beading, solder-fillet-lifting, tombstoning, defective balling for BGA, and voiding. The mechanisms, causes, and cures for each issue are briefly discussed in this article.

Key words: reflow, soldering, solder paste, SMT, bridging, dewetting, opening, solder balling, solder beading, solder-fillet-lifting, tombstoning, balling for BGA, voiding

INTRODUCTION

Reflow soldering of solder paste is the primary board-level interconnection method used in SMT assembly process. This bonding method offers the best combination of desired features including ease of processing, wide compatibility with varieties of SMT designs, high joint reliability, and low cost. However, while reflow soldering is being considered as the most vital SMT component-level and board-level interconnection method, it is also being challenged with the task of further improvement in the soldering performance. As a matter of fact, whether reflow soldering can meet this challenge will determine whether solder paste can remain as the primary SMT bonding material, particularly with the continuous advancement of ultra-fine pitch technology. The major issues which plague the reflow soldering performance are described below. The mechanisms and causes for each issue are also briefly presented in order to inspire the industry to develop new approaches to meet this challenge.

CHALLENGES

Bottom-Side-Component-Holding

Two-sided reflow soldering has been in use for a long time, with the first side printed, placed, reflowed, then turned over, and followed by the second side processing. In order to achieve a greater saving, certain process even skips the first side reflow and ends up reflowing both top side and bottom side in one reflow run. Typically, the bottom-side contains only small components such as chip capacitors or chip resistors. Due to the increasing complexity of PCB design, more and more larger components are included at the bottom-side. As a result, components dropping during reflow becomes an issue. Apparently, this component dropping phenomenon is a direct result of insufficient vertical holding force of the molten solder during reflow. This lack of vertical holding force can be attributed to the increasing weight of component, poor component solderability, insufficient flux wetting ability, or insufficient solder volume, with the first factor being the root cause. If the latter three factors are improved and the symptom still persists, use of SMT adhesive will be inevitable. Apparently, use of adhesive will suffer the lack of component self-alignment advantage during reflow.

Bridging

Bridging is the formation of a solder bridge between neighboring leads. In general, all factors contributing to paste slump will contribute to bridging. These include 1) too fast a ramp-up rate, 2) too low in paste thixotropic property or too slow in paste viscosity recovery after shearing, 3) too low in metal load or solid content, 4) too wide in powder particle size distribution, 5) too low in flux surface tension. However, slumping not necessarily will result in bridging. At reflow, a bridged molten solder will tend to break apart due to the surface tension driving force. Bridging problem will be aggravated by solder-robbing phenomenon[1, 2]. In this case, the excessive solder accumulated at certain area due to solder-robbing may become too much for the molten solder to break apart.

Besides the factors causing paste slumping problem, the following factors are also the frequently encountered causes of bridging[2, 3]: 1) too much paste deposited relative to the spacing between the pads, 2) heating temperature too high, 3) paste being heated faster than boards, 4) flux wetting speed too fast, 5) flux vapor pressure too low, 6) flux solvent content too high, 7) flux resin softening point too low.

Dewetting

Dewetting[2, 4, 5] of a solder film has the appearance of water on a greasy surface. It arises when a surface is covered initially by molten solder with the solder adheres to most of the solid metal surface and with some nonwetted spots hidden underneath this molten solder coverage. This meta-stable molten solder coverage retracts after a time due to minimal-surface-energy driving force, and consequently causing the solder to collect up into discrete globules and ridges[2, 6]. Dewetting can also result from gas evolution during exposure of the part to molten solder. Thermal breakdown of organics or the release of water of hydration from inorganics generates the gas. Water vapor is a common component of these released gases. At soldering temperature, water vapor is highly oxidizing and results in oxidation either of the surface or the molten solder film or in some subsurface interface (typically, the intermetallic surface at the molten solder interface). It has often been observed that higher soldering temperatures and longer dwell times result in more severe dewetting. This most often happens when the source of the gas release is in the base metal. This increase in reaction rate produces a more vigorous release, while the longer dwell time increases release time. Both result in an increased release volume. Approaches about eliminating the dewetting problem include: 1) reduce soldering temperature, 2) reduce dwell time at reflow, 3) use inert reflow atmosphere, 4) reduce contamination level.

Low-Residue

For no-clean reflow process, low residue is often desired either for cosmetic or for functional purpose. Examples of the latter need include 1) probing on a test pad through the flux residue at an in-circuit-test, and 2) making electrical contact between plug-in-connectors and pads or through-holes near reflowed solder joints. High level of flux residue often results in too much residue coating on the metallization-to-be-contacted thus hinders the establishment of electrical connection. This becomes an increasing concern when the circuitry density increases.

Obviously, a low-residue no-clean solder paste would be an ideal answer for this demand. However, this approach is complicated by the associated necessary reflow condition. A semi-empirical model is proposed[7] to predict the soldering performance of low residue solder pastes under various levels of inert reflow atmospheres. This model predicts that the soldering performance would improve rapidly then gradually level off with decreasing oxygen content. Experimental results indicate that the bond strength and wetting ability increases with decreasing oxygen concentration. Also, the bond strength increases with increasing flux solid content. The data match the model fairly well. This strongly proves that the model is valid and is very capable of predicting the soldering performance of solder paste materials. Therefore, it can be concluded that an inert reflow atmosphere should be used in order to successfully adapting a low-residue no-clean solder paste for soldering process.

Opening

Opening is a phenomenon where there is no solder joint formed between component lead and board pad. In general, this can be attributed to four main causes: 1) insufficient solder deposit, 2) poor lead coplanarity, 3) lack of wetting, and 4) solder robbing due to paste slumping, lead wicking[1, 2, 3, 8], or a nearby via-hole for a pretinned PCB[2]. The lead coplanarity issue is particularly a concern for the new, light weight 12 mil pitch QFPs. To address this issue, a method of solder pre-coating the pads prior to the assembly step has been proposed[9]. By forming a controlled local solder hump along the solder bump pre-coating with the use of an enlarged local pad dimension design, this method is reported to be able to compensate the variation in lead coplanarity and consequently prevent the opening. The lead wicking can be addressed by using a slower heating rate and more bottom-side heating than top-side heating. It can also be minimized by using a flux with slower wetting speed, a higher activation temperature, or by using a solder paste with retarded melting, such as a paste with a blend of Sn powder with Pb powder. The wicking due to nearby via-hole can be prevented by applying solder mask to cover the connecting trace prior to the application of a Sn-Pb coating for board finish.

Solder Balling

Perhaps the most common and also the most complicate problem should be the solder balling. At reflow, small spherical particles with various diameters are formed away from the main solder pool and do not coalesce with the solder pool after solidification. In most instances, the particles are composed of the solder powder particles used in the solder paste. In some other cases, the solder balls may be the result of several solder powder particles. Formation of solder balls causes concern for both circuit shorts or leakage currents as well as the possibility of insufficient solder in the joint. With the advancement of fine-pitch technology and no-clean approaches, the demand for solder balling free SMT processes is getting more and more stringent with time.

Causes of solder balling[1, 2, 4, 10] include: 1) smearing due to inadequate printing process, 2) excessive exposure of paste to oxidative environment, 3) extensive exposure of paste to humid environment, 4) inadequate heating methods, 5) too rapid a heating rate, 6) too long a preheat profile, 7) interaction between solder mask and solder paste, 8)

insufficient flux activity, 9) excessive solder powder oxides or contamination, 10) too much fines, 11) inappropriate volatiles incorporated in flux for specified reflow processes, and 12) paste slumping due to inadequate paste formulation.

Solder Beading

Solder beading is a special phenomenon of solder balling when using solder paste in certain SMT applications. In brief, solder beads refer to some very large solder balls, with or without the presence of tiny solder balls, formed around components with very low stand-off, such as chip capacitors or chip resistors[11]. Solder beading was caused by flux outgassing which overrode the paste cohesive force during the preheat stage. The outgassing promoted the formation of isolated paste aggregates underneath the low clearance components. At reflow, the isolated paste melted and, once emerged from the underside of the components, coalesced into solder beads.

Causes of solder beading include: 1) too high a print thickness, 2) too much overlap area between pads and component, 3) too much paste printed underneath the components, 4) component placement pressure too high, 5) preheat ramp-up rate too high, 6) preheat temperature too high, 7) moisture outgassing from components or from solder resist, 8) flux activation too high, 9) powder used too fine, 10) metal load too low, 11) paste slump too much, 12) solder powder oxide too much, 13) inadequate solvent vapor pressure (usually too low). Perhaps the easiest way to eliminate the solder beading problem is to alter the stencil aperture shape so that less paste is sandwiched between the low stand-off components and the pads.

Solder-Fillet-Lifting

Solder-fillet-lifting is the complete lifting of lead and solder fillet, particularly near the corner of the components, from the land pad observed on fine pitch QFP after wave soldering. A likely cause is the mechanical stress imparted to the leads during pre-wave pick test or mechanical damage imparted during board handling[12]. During pre-wave pick testing, a tweezer is drawn over the leads of QFP components to determine if all the leads had soldered in the reflow oven. This result in a non-alignment of toes when viewed from the top. If the underside heating of the wave cause a partial secondary reflow at the land/fillet interface, this inbuilt stress could be relieved by a lifting of the lead and fillet from the board. One way to prevent this problem is to conduct the pick test after wave soldering instead of before wave soldering.

Tombstoning

Tombstoning is the lifting of one end of a leadless component, such as capacitor or resistor, free of the substrate or even the component standing on one of its ends. Tombstoning is also known as the Manhattan effect, Drawbridging effect, of Stonehenge effect. It is caused by an unbalanced wetting of the two ends of the component at reflow and accordingly the unbalanced surface tension pulling force of the molten solder exerted onto the two ends. With the advancement of miniaturization of SMT, the components become more and more susceptible to this problem.

In most instances, tombstoning can be traced back to processing or design related issues[1, 2, 8], including: 1) too rapid a heating rate at reflow, 2) use of vapor phase reflow method, 3) inadequate spacing between the two pads of the chips, 4) too thick a solder paste print thickness, 5) the width and area of metallization under the chip component being too small, 6) too long an extension of the solder pad beyond the chip ends, 7) the solder pad width being too wide, 8) board finish being Sn-Pb coating, 9) uneven solderability of component termination metallization or PCB pad metallization, due to either contamination or oxidation, 10) component placement accuracy being too poor. Tombstoning can also be caused by materials[1, 2], such as 1) the wetting speed of the flux being too fast, 2) the outgassing of the flux being too much at reflow, 3) the wetting speed of the solder being too fast.

Defective Balling for BGA

Balling for BGA often encounters defects such as bridging, misalignment of balls, missing balls, and insufficient solder volume. This is usually caused by insufficient holding force toward the spheres during reflow, or insufficient self-centering force. The insufficient holding force can be a result of low viscosity, high barrier thickness, or high outgassing rate. The insufficient self-centering force is typically caused by relatively weak flux activity or by low solder volume.

The balling process for BGA can be achieved by using solder paste alone, solder spheres with solder paste, and solder spheres with fluxes. Also possible is the use of integrated preform together with either flux or solder paste. The most common approach appears to be using spheres with solder paste. For balling process involving Sn62 or Sn63 spheres, use of paste for sphere attachment produces excellent results. In the case of using fluxes for Sn62 or Sn63 sphere attachment, the defect rate increases with decreasing flux viscosity, decreasing solvent volatility, decreasing pitch dimension, increasing flux deposition thickness, increasing flux activity, and increasing pad diameter. For systems using pastes for high melting temperature sphere attachment, no missing is observed, and the alignment improves with decreasing paste deposition thickness, decreasing solvent volatility, increasing sphere solderability, increasing flux activity, increasing pad dimension, increasing metal load,

increasing pad solderability. Paste viscosity, pitch, and reflow profile has negligible effect on the high melting temperature balling yield using Sn63 solder paste. An easily releasable solder paste is crucial if a regular print-release process is desired for balling with solder paste alone. Balling with integrated preform is promising. Reducing the thickness and width of the solder link is considered essential for improving the balling success rate.

Voiding

Voiding is a phenomenon commonly associated with solder joints. This is especially true when reflowing a solder paste in the SMT application. In the case of LCCC, it was found that the overwhelming majority of large (>0.0005 in./0.01 mm) voids were located between the LCCC pads and PWB solder pads, while the fillets near the LCCC castellations contained very few small voids. Presence of voids will affect the mechanical properties of joints and deteriorate the strength, ductility, creep and fatigue life, due to the growth in voids which could coalesce to form ductile cracks and consequently lead to failure. The deterioration could also be due to the enhanced magnitude of the stresses and strains of solder caused by voids. In addition, voids could be attributed to 1) solder shrinkage during solidification, 2) laminate outgassing during soldering the plated through-holes, and 3) entrapped flux.

For solder paste processes, the voiding mechanism is more complicate[2, 13]. Generally, the voids are caused by the outgassing of entrapped flux in the sandwiched solder during reflow. The voiding is mainly dictated by the solderability of metallization, and increases with decreasing solderability of metallization, decreasing flux activity, increasing metal load of powder, and increasing coverage area under the lead of the joint. Decrease in solder particle size causes only a slight increase on voiding. Voiding is also a function of the timing between the coalescing of solder powder and the elimination of immobile metallization oxide. The sooner the paste coalescing occurs, the worse the voiding will be. Increase in voiding usually is accompanied by an increasing fraction of large voids, suggesting factors causing voiding will have an even greater impact on the joint reliability than what shown by the total-void-volume analysis results. Control of voiding may include 1) improving component/substrate solderability, 2) using fluxes with higher flux activity, 3) reducing solder powder oxide, 4) using inert heating atmosphere, and 5) slowing down the preheat stage to promote fluxing before reflow.

Voiding in BGA assembly follows a slightly different pattern[14] than the one shown above. In general, voiding in BGA assembly using Sn63 solder bumps is primarily introduced at board-level assembly stage. On the pretinned PCBs, voiding of BGA joints increases with increasing solvent volatility, increasing metal content, and increasing reflow temperature, and with decreasing powder size. This can be explained by a viscosity dictated flux-exclusion-rate model. In this model, a higher viscosity in fluxing medium at reflow temperature could hinder the exclusion of flux from the interior of molten solder, hence increase the chance of outgassing due to the increasing amount of entrapped flux, and consequently result in a higher voiding in BGA assembly. Flux activity and reflow atmosphere appear to have negligible effect on voiding when the solderability of the immobile metallization is not a concern. Increase in void content is accompanied by an increase in fraction of large voids. This suggests that, similar to voiding phenomena in SMT process, factors causing voiding in BGA will have an even greater impact on the joint reliability than what shown by the total-void-volume analysis results.

SUMMARY

Reflow soldering of solder paste is the primary board-level interconnection method used in SMT assembly process. The major issues which plague the reflow soldering performance include, but not limited to, bottom-side-component-holding, bridging, dewetting, low-residue, opening, solder balling, solder beading, solder-fillet-lifting, tombstoning, defective balling for BGA, and voiding. The mechanisms, causes, and cures for each issue are briefly discussed in this article. Issues not addressed here include leaching, intermetallics, non-wetting, skewing, lead-free soldering, etc. Only by meeting this challenge successfully will allow reflow soldering remain as the vital SMT assembly processing method in the ultra-fine-pitch era.

REFERENCE

1. N.-C. Lee and G. P. Evans, “Solder Paste - Meeting the SMT Challenge”, SITE Magazine (June, 1987).
2. N.-C. Lee, “How to Make Solder Paste Work in Ultra-Fine-Pitch and Non-CFC Era”, a short course in Nepcon West, Anaheim, CA (Feb. 1993).
3. J. S. Hwang, “Solder Paste in Electronics Packaging”, Van Nostrand Reinhold, New York (1989).
4. C. Lea, “A Scientific Guide to Surface Mount Technology”, Electrochemical Publications, Isle of Man, British Isles (1988).
5. J. A. DeVore, “To Solder Easily: the mechanisms of solderability and solderability-related failures”, Circuits Manufacturing, pp.62-70 (Jun. 1984).
6. R. J. Klein Wassink and E. E. de Kluizenaar, “De-wetting of Molten Solder from Copper”, in Proc. of Deutscher Veriag fur Schweisstechnik Conference on “Soldering and Welding in Electronics and Precision Mechanics”, Munich, Germany, Vol. DVS-71, pp. 16-21 (1981).

7. P. Jaeger and N.-C. Lee, “A Model Study of Low Residue No-Clean Solder Paste”, in Proc. of Nepcon West, Anaheim, CA (Feb. 1992).
8. R. J. Klein Wassink and J. A. H. van Gervan, “Displacement of Components and Solder during Reflow Soldering”, Soldering & Surface Mount Technology, pp.5-10 (Feb. 1989).
9. A. Wakigawa, “Solder Print for Super Fine Pitch SMDs”, EP&P, pp.42 (Jan. 1995).
10. M. Xiao, K. J. Lawless, and N.-C. Lee, “Prospects of Solder Paste in UFPT Era”, in Proc. of Surface Mount International 93, San Jose, CA (Aug. 1993).
11. W. B. Hance, P. A. Jaeger, and N.-C. Lee, “Solder Beading in SMT - Causes and Cures”, in Proc. of SMI, San Jose, CA (Aug. 1990).
12. J. Barrett, C. O Mathuna and R. Doyle, "Case Studies in Quality and Reliability Analysis of Fine Pitch Solder Joints", Soldering & Surface Mount Technology, No. 13, pp.4-11 (Feb., 1993).
13. W. B. Hance and N.-C. Lee, “Voiding Mechanisms in SMT”, China Lake’s 17th Annual Electronics Manufacturing Seminar, China Lake, CA (Feb. 1993).
14. W. B. O’Hara and N.-C. Lee, “Voiding in BGA”, in Proc. of SMT, San Jose, CA (Aug., 1995).