



Defect Minimization Methods for a No-Clean SMT Process

Our thanks to Kester for allowing us to reprint the following article.

By Kurt Rajewski, Assistant Manager - Market Technology, Kester

Key competitive advantages can be obtained through the minimization of process defects and disruptions. In today's electronic manufacturing processes there are many variables to optimize. By gaining an understanding of what the defects are, and where they come from, is a key step in the process towards defect free/six sigma manufacturing. In the last decade, Surface Mount Technology processes have been slowly converting towards the No-Clean philosophy. This new trend has spawned new processing issues which need to be addressed. This paper will investigate solutions to current problems in the processing of No-Clean SMT processes. These solutions will be critical in the development of successful processes in the electronics industry in the years ahead.

Introduction:

This paper will discuss commonly experienced defects associated with No-Clean surface mount processes and propose methods to solve these issues. Much of the discussion can be applied to any surface mount process (i.e. water soluble, RMA, or No-Clean), but some are directly associated with No-Clean processes. Some of these defects are due to inefficient reflow profiles - examples of these defects would include cold solder joints, non-wetting, solder balling, and tombstoning.

Other defects can be primarily attributed to the solder paste printing process - these would include insufficient solder joints, bridging, and solder balling; while others can be attributed to miscellaneous process variables - these would include skewed components, solder beading, and solder balling.

The effects of extreme temperature and humidity conditions will also be discussed in this paper. In addition to this, some processing tricks that are designed to maximize the solder paste performance will also be discussed.

Reflow Related Issues:

1. Cold Solder Joint (dull joint): is defined as solder connections exhibiting poor wetting and possessing a grayish, porous appearance after soldering. This is a phenomena that can be associated with all processes.

One cause of this would be insufficient heat present to reflow the solder adequately. It may also be due to the flux's inability to accomplish the soldering task. This may be linked to inadequate cleaning of the component and PCB pads prior to soldering, or it may simply be due to excessive impurities in the solder solution. Possible solutions to 2 this problem include 1) raising the maximum reflow temperature high enough to reflow the material thoroughly, 2) preventing vibration of assembly during and immediately after reflow, 3) accelerating the cool down rate after reflow, and 4) checking the alloy analysis for high levels of contaminants.

2. Non-wetting: is defined as a condition whereby a surface has contacted molten solder, but has had part or none of the solder adhere to it. Again, this is a phenomena that can be associated to all processes.

There are various causes of non-wetting. It could be due to the base metal being visible, and since this is typically more difficult to solder to, non-wetting occurs. It might also be due to too long of a soak time in the reflow process using up the flux prior to soldering. The flux being used may be ineffective from an activity standpoint. Or, it could possibly be due to insufficient heat during the reflow process where the flux doesn't see the correct activation temperature.

Therefore, the solutions include 1) rectifying the situation with the PCB manufacturer if the base metal is present, 2) reducing the total profiling time prior to the reflow stage, or following the recommended reflow profile (see Exhibit #1: No-Clean Solder Paste Reflow Profile), and 3) increasing the flux activity, or using the correct flux for the given soldering task (see Exhibit #2: Metal Solderability Cross Reference Chart).

3. Solder Balling: is defined as the formation of very small spherical particles of solder separating from the main body of solder which forms the joint. This is a primary concern for No-Clean processes since a large number of solder balls can create an artificial bridge between two adjacent leads causing functional problems to the electrical circuit. Solder balling is not as big of a concern with water soluble processes since they typically are removed during the cleaning process.

One possible cause of solder balling may be moisture contaminated solder paste. The moisture splatters during reflow leaving solder spheres behind. An improper reflow profile can also cause solder balling. The temperature ramp rate is commonly too high which increases the probability of paste splattering. Solder balling can possibly be due to excessive oxides on the solder powder in the solder paste which inhibits solder coalescence during reflow.

Among the ways to approach this problem are the following 1) select a reflow process which best fits the paste selected (see Exhibit #1: No-Clean Solder Paste Reflow Profile), 2) minimize solder paste's exposure to high temperatures and humidities whenever possible.

4. Tombstoning: is defined as a soldering defect in which a chip component has pulled into a vertical or near vertical position with only one termination soldered to the PCB, resulting from force imbalances during the reflow soldering process. This is also known as drawbridging, the manhattan effect, and the stonehenge effect.

Tombstoning can be caused by uneven heating causing a differential across the component terminals. In other words, the solder melts at different rates and one side reflows before the other forcing the other lead to stand upright. The solderability between two terminations of the component or PCB pads can also be blamed. Uneven paste deposition on the two solder pads has also yielded tombstoning defects. Insufficient tack force of the solder paste to hold the component in place during reflow can also be a factor, but this is typically due to temperature and humidity effects on the solder paste. Excess movement during and after the reflow operation can cause component misalignment which results in tombstoning, and inadequate placement force to make intimate contact between the paste and the termination of the pads can also be a cause.

Solutions to this surface mount defect include 1) increasing the preheating temperature (following the recommended guidelines) so that the temperature differential between the two terminations is low at the time of reflow, 2) selecting components and PCBs with consistently solderable leads and pads, 3) ensuring consistent solder paste deposit heights between pads via a vision system designed to measure solder paste deposition height, 4) avoiding inefficient tack force by avoiding extreme environmental conditions, 5) minimizing the amount of movement the assembly sees during reflow, and 6) increasing component placement force to ensure contact of the component terminal to the solder paste deposit (not too much because bridging may occur if it is too high).

Recommended Reflow Profile:

The majority of no-clean solder pastes are rosin or resin based, and therefore can withstand the traditional reflow profiles used on RMA based fluxes. The goal of the reflow process is to melt the powder particles in the solder paste, wet the surfaces being joined together, and solidify the solder to create a strong metallurgical bond. The profile can be broken down into four zones - the preheat, soak, reflow, and cool down zones. An illustration of this profile can be found in Exhibit #1 at the end of this paper.

This profile should be used in order to avoid the following process problems. The Pre-Heat section helps prevent against insufficient solvent evaporation, component/PCB shock, and solder ball formation due to splattering. The Soak section guidelines prevent insufficient flux activation and excess oxide formation. And the Reflow section guidelines avoid flux entrapment, void formation, flux discoloration, and component and board damage.

Printer Related Issues:

1. Electrical Opens (due to insufficient solder joints): can be defined as the result of two electrically connected points becoming separated, or as an area on the PCB which interrupts the intended design on the circuit.

Causes for this type of defect are commonly attributed to the solder paste printing stage of a surface mount process. However, other non-printer related factors can also cause electrical opens. Solder paste can clog in the apertures of the stencil, never being released to the PCB pad. This will create an insufficient solder joint due to insufficient solder being placed prior to reflow. Component lead coplanarity (the distance between the PCB pad and the component lead) can also contribute to opens. The solder volume may be adequate, but if it is not in contact with both the lead and the pad during reflow, an open will occur. Finally, opens may also be a function of the PCB fabrication process itself.

Solutions for electrical opens include 1) correcting the aspect ratio. If solder paste is clogging the apertures it may be due to the aspect ratio being too small. The aspect ratio is defined as the ratio of aperture width to stencil thickness - use a ratio of 2.0 as a guideline for fine pitch applications. 2) Avoid solder paste contamination by avoiding extreme environmental effects in the manufacturing process, 3) investigate lead coplanarity issues and monitor operator material handling procedures, and 4) investigate fabrication issues with PCB supplier.

2. Electrical Bridges (excess solder): are defined as solder that 'bridges' across two conductors that should not be electrically connected, causing an electrical short.

Bridging can be caused by a variety of factors, but they are most commonly caused by problems in the solder paste printing process. The print alignment, or the alignment of the stencil to the PCB pad design, may be slightly off. Bridging can also be caused from too much solder paste being deposited. This may be due to the stencil aperture to pad ratio being too high (1 to 1 for example, where the stencil aperture and the PCB pad have the exact same dimension). Solder paste cold slump can also lead to bridging. Slump is typically a problem if the incorrect solder paste metal to flux weight ratio is being used. High temperatures and humidities can also lead to solder paste slump. Typically, however, if the solder paste includes a thixotropic thickening agent within the flux formula the solder paste will maintain its shape. The reflow profile may also contribute to bridging if the pre-heat section has too slow of a ramp rate. And finally, component contact with solder paste deposit may skew the deposit causing the solder paste to bridge.

Bridging can be avoided by doing the following 1) use the appropriate solder paste metal to flux weight ratio for the appropriate application. Typically either a higher viscosity or a higher metal content can solve the problem (i.e. dispensable solder paste has a metal content of 85-87% metal typically, this material will slump if used to print fine pitch surface mount. Typically 90% metal is used for a stencil printing solder paste application). 2) Use the appropriate reflow profile (see Exhibit #2), 3) ensure paste deposition is in good resolution and quality without slump or smear prior to reflow (this can be done manually or with an automated vision system), 3) pay close attention to alignment of stencil apertures to pads (automatic printer alignment will alleviate this issue), 4) reduce stencil aperture dimensions by 10% or reduce the thickness of the stencil to reduce the amount of solder paste being deposited, and 5) ensure proper pressure and accuracy for component placement.

3. Solder Balling: was defined earlier, but it can also be attributed to problems with the solder paste printing process.

Solder balling can be caused by poor solder paste printing alignment where solder paste is printed on the solder mask instead of the pad. The solder paste may not be able to coalesce into the joint and may solidify across two adjacent pads causing an undesirable bridge. Solder paste may also get smeared on the bottom side of the stencil during the printing process.

Solutions to these problems include 1) verifying print alignment prior to reflow on a consistent basis (this can be done manually or automatically via an electronic vision system), and 2) ensuring frequent cleaning of the bottom of stencil (this can be done automatically on automatic printers or manually with a lint free cloth and alcohol).

Miscellaneous Issues:

1. Skewed Components (components falling off the pads): can be defined as a descriptive term used to describe the misalignment of an item to its target.

The reasons for the occurrence of this defect are much more straight forward. Insufficient tack characteristics of the solder paste, typically a by-product of extreme temperature and humidity effects on solder paste, is the primary reason for this. Component placement inaccuracies, as well as too much movement of the assembly prior and during reflow, can also contribute to this problem. Poor component or PCB solderability characteristics may also create skewed components.

These problems can be avoided by trying the following 1) abide by recommended temperature and humidity requirements, 2) improve the accuracy of component placement, 3) minimize the amount of movement the unreflowed assembly sees, and 4) improve the solderability of the components or PCBs (this may also be accomplished by using a more aggressive flux).

2. Solder Beading (or Side Balls): is defined as the formation of larger solder balls located near discrete components possessing very low standoff distances.

This defect is similar to solder balling, but it is distinctive in the fact that these solder beads adhere to discrete components as opposed to multi-leaded devices.

The majority of this type of problem is due to excessive amount of solder paste being deposited. Another reason may be the flux outgassing which overrides the paste's cohesive (coalescence) force during the preheat stage. Component placement pressure may also be too high. Excessive pressure may push deposited solder paste out onto the solder mask where it can not coalesce back into the joint.

Solving this defect can be easily accomplished by doing one of the following 1) reducing the stencil thickness or reducing the aperture dimensions (a 10% reduction on the side where the solder bead occurs should solve this problem), 2) using the recommended temperature profiling guidelines, and 3) reducing the component pick and place pressure.

3. Solder Balling: can also be attributed to the solder paste. Solder balling can occur if the solder paste contains a large percentage of ultra fine powder particles (sub 25 micron in diameter) which can be carried away from the main solder pool by flux during heating. It may also be found that misprinted boards, inadequately cleaned prior to reprocessing, will have solder balls.

If these are the reasons that solder ball formation is occurring, then the following two solutions will help 1)

monitor solder paste vendor's control of oxide and fine particles in their powder distributions, and 2) use an automated cleaning system to clean misprinted board with an approved solvent (IPA is commonly used).

Temperature/Humidity Effects

Solder paste printing performance, tack life, and reflow characteristics can be greatly affected by the temperature and humidity characteristics of the manufacturing environment. The recommended temperature the solder paste should be at when printing is 70-77°F. The recommended humidity is between 35-65% RH. Ideal temperature storage for solder paste is typically 0-5°C (32- 40°F), but some newer formulas do not require refrigeration (it actually makes the solder paste perform poorly).

If the solder paste is too hot due to the working environment, paste slump is more likely to occur. As discussed previously, this can lead to bridging and other defects. Print definition is also commonly affected and this may lead to solder ball or solder bead formation.

If the solder paste is too cold due to the working environment, the solder paste may become too thick to yield good print definition. This may lead to stencil aperture clogging which in turn will result in insufficient or open joints. If the solder paste is exposed to extreme high or low humidity, similar problems will result.

Tricks to Extend Stencil Life

Stencil life can be lengthened by using a continuous replacement method for keeping the solder paste in a fresh condition. By simply adding new paste to the stencil as you use it, you can lengthen the life of the paste without degradation of tack or reflow characteristics.

Solder paste volume on the stencil also has a relationship to how long the solder paste will last. It is recommended to have adequate solder paste volume on the stencil so that a 1/4" to 1/2" roll of solder paste is in front of the squeegee at all times. This volume leads to better print definition as well.

The effects of time on reflow and tack characteristics for solder paste that has been printed on a PCB are greatly influenced by the environmental conditions of the manufacturing facility.

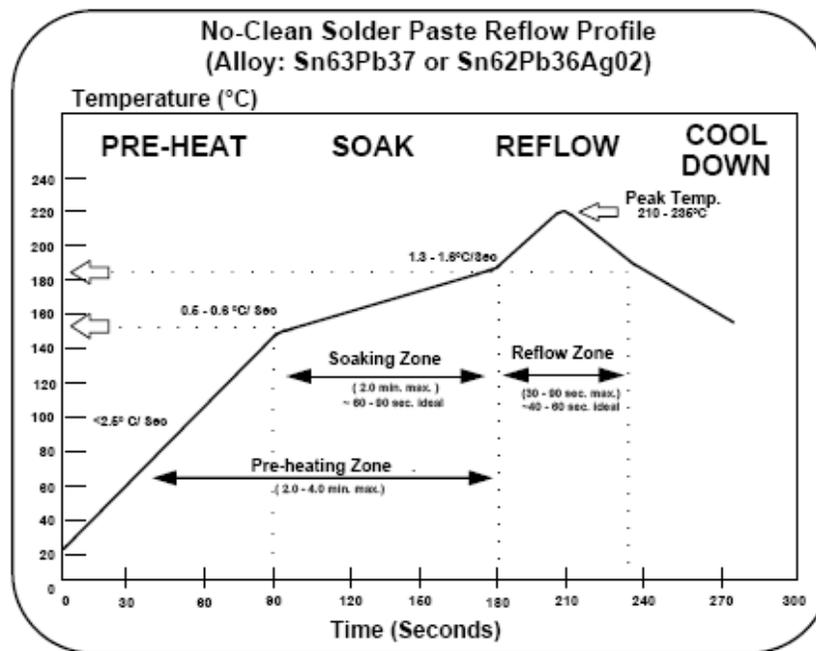
The general recommendation is to keep the time between printing and reflow down to a minimum so solderability characteristics are not sacrificed.

REFERENCES

1. Eric Slezak, Kester, Applying Kester Solderpaste to Surface Mount Assemblies
2. Phil Zarrow and Debra Kopp, ITM Inc., Surface Mount Technology Glossary: Terms and Definitions
3. C. Johnson and J. Kevra, Ph.D., Solder Paste Technology Principals and Applications
4. Tom Gervascio, Philips Circuits Assemblies, Solder Beads: How to Make Them A Vanishing Act
5. Surface Mount Technology Magazine, 10 Easy Steps: A Step by Step Guide to Surface Mount Technology
6. Howard Manko, Solders and Soldering, 3rd Edition.



Exhibit #1: No-Clean Solder Paste Reflow Profile:



Zone 1: Initial Pre-Heating Stage (Room Temperature to 150°C)

- Excess solvent is driven off
- PCB & Components are gradually heated up
- Temperature gradient shall be $< 2.5^\circ\text{C}/\text{Sec}$ to avoid:
 - Splattering: fast evaporation of solvent and air expulsion resulting in possible solder ball formation.
 - Slump: fast separation of paste flux resulting in possible bridge formation.

Zone 2: Soak Stage (150-180°C)

- Flux components start activation and begin to reduce the oxides on component leads, PCB pads, and solder paste powder spheres.
- PCB components are brought nearer to temperature when solder bonding can occur.
- Allows different mass components to reach the same maximum temperature.
- Activated flux keeps metal surfaces from re-oxidizing.

Zone 3: Reflow Stage (180-235°C)

- Paste is brought to the alloy's melting point
- Activated flux reduces surface tension at the metal interface so metallurgical bonding occurs.

Zone 4: Cool Down Stage (180°C to room temperature)

- Assembly is cooled evenly so that neither excess intermetallics form nor excess thermal shock to the components or PCB occurs.

Exhibit #2: Metal Solderability Cross Reference Chart:

Category	Flux	Solderability	Metals
1	R, RMA RA, OA, IA	Easy to Solder	Platinum Gold Copper Tin Solder Palladium Silver
2	RA, OA, IA	Less Easy to Solder	Nickel Cadmium Brass Lead Bronze Rhodium Beryllium Copper
3	OA, IA	Difficult to Solder	Nickel-Iron Kovar
4	IA	Very Difficult to Solder	Zinc Mild Steel Chromium Inconel Monel Stainless Steel

Key (listed in increasing order of activity):

R = Rosin Activated
RMA = Rosin Mildly Activated
RA = Rosin Fully Activated
OA = Organic Acid Activated
IA = Inorganic Acid Activated