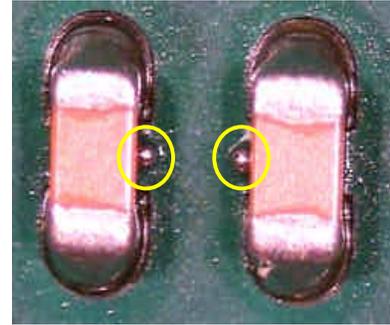


AIM Tech-Sheet

Solder Beading in SMT: Causes & Cures

Solder Beading Defined

When discussing solder beading, the first priority is to accurately define the SMT defect. Solder beads are found on boards that have been reflowed and are recognized by a large ball of solder embedded in a pool of flux located next to discrete components with very low stand-offs such as TSOPs, SOTs, and resistor packs. Because of their location in relation to these components, solder beads often are referred to as “satellites”. Solder beading is also sometimes referred to as “mid-chip squeeze balls”, or something similar, for apparent reasons. Unfortunately, solder beading also is often referred to as “solder balling”. Contrary to solder beading, solder balling is characterized by several tiny balls trapped along the peripheral edge of flux residue and/or balls stuck around fine pitch lands and solder mask. When asked to remedy a case of solder balling or beading, your first questions should be, “What does it look like and where is it?”.



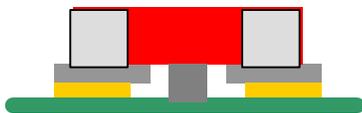
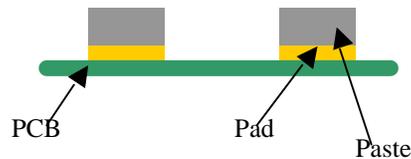
Why Is It Problematic?

Basically, solder beads may form a “bridge” of solder that runs from one component termination to another, thus causing an electrical connection that was not designed to be there. This poses the threat of resulting in a short circuit. This may occur where the bead was originally formed or elsewhere on the assembly if vibration causes the bead to break loose and move around. While the above may not necessarily occur if the solder beads are present, solder beading obviously remains a defect that should be minimized or eliminated if possible.

How Does It Happen?

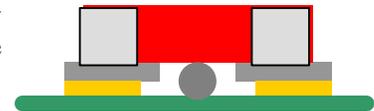
Before we discuss the actual causes of solder beading, it will be valuable to discuss the dynamics by which it occurs. It will be most effective to accomplish this via pictures:

1. Solder paste is printed on the pads of a circuit board.



2. During component placement some solder is squeezed underneath the body of the component and broken off from the solder on the pads.

3. During reflow, the solder trapped underneath the component does not flow back to the solder pads. Contrarily, its cohesive properties (surface tension) cause it to form a large ball (bead).



4. The surface tension of the cooling solder draws the component closer to the pads. As the body of the component is drawn down, the solder bead squeezes out the side and remains there.

Solder beads also can occur from paste bleeding under the stencil as a result of excess squeegee pressure or improper gasketing between the stencil and the PCB. This paste is transmitted to the PCB outside the aperture dimensions. When reflow occurs it may remain on the PCB adjacent to the component aperture in solder bead form.

Why Does It Happen?

Simply stated, solder beading generally is related to an excessive deposit of solder paste that, because of its lack of “body”, is squeezed underneath a discrete component and then becomes a solder bead as is

described above. The increase of solder beading as a common solder defect may be traced back directly to the increased usage of no-clean solder pastes. Back in the old days (pre-early 1990s), when rosin-based (RMA) pastes were still prevalent, solder beading was a rare defect. Why did the frequency of solder beading increase with the use of no-clean pastes? Simple: rosin-based pastes are less likely to be squeezed underneath the body of a component than are no-clean pastes; no-clean pastes have less blanket-material than rosin-based pastes, and thus do not have the rigidity or body of these thicker pastes. Thus when a chip-type component is placed into no-clean paste, this paste is more likely to be squeezed underneath the component. Of course, this squeeze out is all the more likely when there is an excess of solder paste deposited.

Just as there are a number of causes of solder balling there are also several factors which can lead to or encourage the formation of solder beads.

Mis-registration between the stencil aperture and the pad can lead to solder paste printing onto the board mask resulting in beading.

Operators can inadvertently transfer solder paste to the mask in an attempt to straighten out a misplaced component. Usually it is advisable NOT to attempt to straighten a component prior to reflow. Tweaking the placement, usually done with tweezers, can also lead to insufficient, shorts, and voiding. Allow the paste and the reflow oven to float the component to the lands and straighten the component.

Worn equipment, stencils, and squeegees as well as warped boards or insufficient stencil wiping can also contribute to beading and micro balls as well.

How Can It Be Prevented?

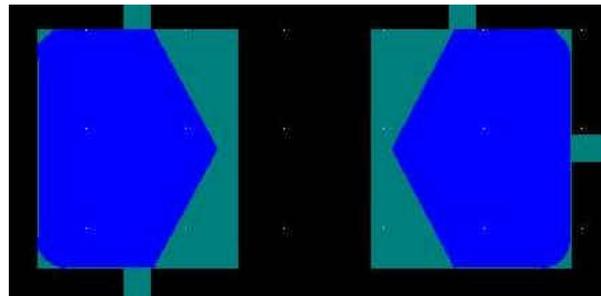
It probably is unnecessary to mention that manually removing solder beads is prohibitively expensive and impractical. Therefore, it is desirable to eliminate solder beads before they appear. As the above states, solder beading usually is related to an excessive deposit of solder paste. This being the case, the obvious solution to solder beading is to reduce the amount of solder paste deposited during the print. This may be accomplished by making adjustments to several variables:

- Aperture Size/Shape
- Stencil Thickness
- Snap-off Distance
- Squeegee Pressure and Speed
- Paste Viscosity
- Reflow Profile

Aperture Size/Shape

Probably the most viable method to prevent solder beading is through the adjustment of the size and shape of stencil apertures. In the past, printing 1:1 was common; that is, the dimensions of an aperture were precisely the same as that of the pad to which it corresponded. Today, when a no-clean paste is in use it is common to utilize an aperture reduction and/or shape adjustment. A ten percent reduction on each side of the aperture or larger reduction on the lead edge of the component is common.

A variety of shapes such as ovals, triangles, and odd-form shapes have been used to eliminate solder beading with varying success. One of the most common and successfully used aperture reduction methods is the home plate design (see image). This reduces the amount of paste printed and helps to keep the paste from spreading off the pads, where it may ball up and become



a bead. Remember that it's not only the reduction that matters; the location of the paste on the pad also is important. For many manufacturers, this may mean bringing in hundreds or thousands of new stencils, which obviously can be quite costly. However, when weighed against the cost of high defect PPMs, this cost may appear negligible.

Stencil Thickness

Another common and viable method to prevent solder beading is through the adjustment of the stencil thickness. Stencils commonly range in thickness from .004" to .008", with .006" often used as a default thickness. The amount of solder paste deposited through a stencil (depending on several factors, especially snap-off) is .001" less than to .002" more than the stencil thickness. Obviously, reducing the thickness of one's stencil(s) will also reduce the volume of solder deposited on the PCB. As the use of high solvent, low solids no clean solder fluxes have become more prevalent, many manufacturers have had to get new stencils cut in thinner dimensions in order to cure/prevent solder beading. Another option is to utilize a "step-down" stencil, which offers a reduced thickness in certain areas in order to reduce the paste deposition in key locations.

The thickness(es) of the stencil and dimensions of its apertures need to be considered in conjunction with one another as well as the components and PCB design in use, as the length, width, thickness, and shape of an aperture can have an impact upon paste release. Your solder manufacturer should be able to provide additional advice and information on recommended aspect ratios and area ratios.

Snap-off

Also known as print-gap, snap-off is the programmable distance between the topside of the PCB and the bottom side of the stencil during the print cycle. While sometimes used to aid the release of solder paste from the stencil apertures, snap-off also results in an increased paste volume deposited on an assembly.

When solder pastes are used in conjunction with a print gap, the increased paste deposition may result in solder beading. For this reason, solder paste manufacturers generally recommend on-contact (zero snap-off) printing. In addition to helping to prevent solder beading, on-contact printing also provides for a more uniform paste deposition, a more consistent paste height, and gasketing between the stencil and the PCB which, as stated above, can help to eliminate paste bleed. As with all settings, however a print gap may exist even if the printer is set at zero snap-off due to PCB, stencil, and/or equipment irregularities. For this reason it is always critical to manually verify settings after they have been programmed.

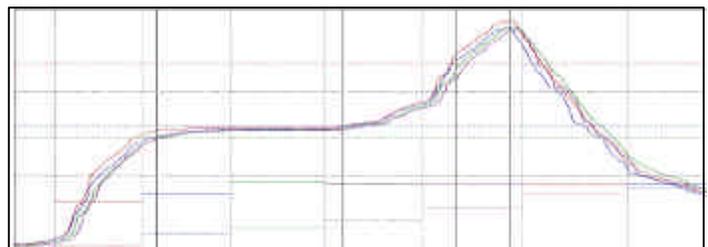
Paste Viscosity

As stated above, lighter-bodied solder pastes are more likely to exhibit solder beading than are heavier-bodied pastes. Increasing the viscosity of solder pastes may help to reduce solder beads, as the thicker paste will be more resilient to being squeezed out the sides of components. However, as with adjusting the squeegee speed and pressure, increasing the paste viscosity is a defect "bandage", could in fact cause printing difficulties, and avoids the key issue of the solder beading problem- excessive paste deposition.

Reflow Profile

Solder beading sometimes can be resolved with a profile adjustment. As with solder balling, solder beading may result from too slow a ramp rate. In this case, the slow ramp rate causes capillary action to draw the unreflowed paste away from the pad on which it was deposited to a place under the component. The paste reflows there forming a bead of solder that comes out to the side of the component. Adjusting the profile to the paste manufacturer's recommendation may help to rectify the issue.

In addition, utilizing an altered reflow profile has proven to aid in the reduction/elimination of solder beads. It has been theorized by some that solder



beading is often the result of the solder paste outgassing during reflow, wherein some paste is broken off from the main body of paste where it forms a solder bead as explained above. The newly developed profile allows the paste to outgas at a slower rate, thus reducing the impetus to expel paste from the main deposit.

Other Factors

Although solder beading normally is the result of an excessive paste deposit during printing, it sometimes can be caused and resolved by other means.

Metal Load

Solder pastes with higher metal contents (e.g., 90% versus 89.5%) have demonstrated a reduced tendency to solder bead. This may occur for multiple reasons: the higher metal load results in a higher viscosity paste that is less likely to lose its integrity during PCB assembly; or perhaps the increased metal content of the paste results in metal packed more closely together, which is more prone to weld upon itself than paste with more widely dispersed metal powder.

Powder Size

It also has been demonstrated that the size of the metal powder in solder paste has some impact upon solder beading. In short, the larger the powder size, the less likely it is to solder bead. For example, 45 micron (-325/+500) powder is less likely to form solder beads than 25 micron (-500/+625) powder when used under the same conditions. In general, larger powders, due to their less overall surface area, normally exhibit less oxide content than finer powders. As stated above, this reduction in oxides can translate into a reduction in solder beading.

How Is It Tested?

As stated above, although solder beading generally is application-related, it may be influenced by the solder paste in use. Thus, it is important to develop a test methodology for determining whether a particular paste is more likely to solder bead than another. Basically, this testing should attempt to result in the maximum amount of solder beading. Solder paste manufacturers such as the authors' utilizes test methods such as the below to develop pastes which significantly reduce the formation of solder beads.

When testing a solder paste for its propensity toward beading, a standard 1:1 ratio aperture to pad should be used. Six-mil stencil test boards are populated with misplaced components as well as properly placed components. Most commonly 1206 size capacitors or resistors are used and placed with sufficient pressure to cause a squeeze out onto the mask itself. During the reflow process a linear reflow profile is used, thus reducing the effects that a long low soak might have in reducing the beading. These variables assist in determining which formulations of paste tend to form beads. It is then a simple process to qualify each board and corresponding paste by the quantity and size of the beads themselves.

Conclusion

As the above indicates, solder beading is a phenomenon that may be caused by several contributing factors. In nearly every case, however, solder beading may be substantially reduced or eliminated by reducing the volume of solder paste deposited on the PCB. In addition, the location of the solder paste on the pad is of critical importance. A viable method used to prevent solder paste is through revised stencil design, wherein the thickness of the stencil is reduced, step-down areas are utilized, and/or apertures are reduced in size and shape.

**For additional information, please contact AIM tel 401-463-5605, fax 401-463-0203,
email info@aimsolder.com, or visit AIM on the web at www.aimsolder.com**