Solder beading, a defect that can result in short circuits, may be reduced substantially by decreasing the solder paste volume deposited on a printed circuit board (PCB).

By Kevin Pigeon

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 chip-type resistors and capacitors, thin small-outline packages (TSOP), small-outline transistors (SOT), D-PAK transistors, and resistor packs (Figure 1). Because of their location in relation to these components, solder beads often are referred to as "satellites." Solder beading also is sometimes referred to as "mid-chip squeeze balls," or something similar, for apparent reasons. Unfortunately, solder beading also is 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 or balls stuck around fine-pitch lands and solder mask. When asked to remedy a case of solder balling or beading, the first questions should be, "What does it look like and where is it?"

Figure 1. An illustration of solder beading by discrete capacitor.

Figure 2. The first step in solder beading: solder paste is printed on circuit board pads.

Why It Is 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 a short circuit, which may occur where the bead originally was formed, or elsewhere on the assembly if vibration causes the bead to break loose and move. While the above may not necessarily occur if solder beads are present, solder beading remains a defect that should be minimized or eliminated if possible.

How It Happens

Before discussing actual solder beading causes, it is helpful to demonstrate the dynamics by which they occur step-by-step via imagery:

1. Solder paste is printed on circuit board pads (Figure 2).
2. During component placement, some solder is squeezed underneath the component and broken off from the solder on the pads (Figure 3).
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) (Figure 4).

4. The surface tension of the cooling solder draws the component closer to the pads. As the component is drawn down, the solder bead squeezes out the side and remains there (Figure 5).

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 It Happens
Simply stated, solder beading generally is related to an excessive solder paste deposit that, because of its lack of "body," is squeezed underneath a discrete component and then becomes a solder bead as is described above. Solder beading increase as a common solder defect may be traced directly to the increased use of no-clean solder pastes. Before the early 1990s, when rosin-based RMA pastes still were prevalent, solder beading was a rare defect. Rosin-based pastes are less likely to be squeezed underneath a component's body than are no-clean pastes; no-clean pastes have less blanket-material than RMA pastes, and do not have the rigidity or body of these thicker pastes. Thus, when a chip-type component is placed into no-clean paste, the paste is more likely to be squeezed underneath the component. This squeeze out is all the more likely when there is an excess of solder paste deposited.

Just as there are numerous causes of solder balling, there also are several factors that can lead to or encourage the solder bead formation:

- Misregistration between the stencil aperture and the pad can lead to solder paste printing onto the board mask, resulting in beading.
- Operators inadvertently can transfer solder paste to the mask in an attempt to straighten out a misplaced component. Typically it is advisable not to attempt to straighten a component prior to reflow. Tweaking the placement, usually done with tweezers, can lead to shorts and voiding. Instead, allow the paste and 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 also can contribute to beading and micro balls.

Preventative Measures
It generally 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 previously mentioned,
solder beading usually is related to an excessive solder paste deposit. 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 by adjusting stencil aperture size and shape. 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 used, it is common to utilize an aperture reduction or shape adjustment. A 10 percent or larger aperture reduction on the lead edge of each side of the component is common.

![Click here to enlarge image](image1.png)

*Figure 5. The final step in solder beading: the surface tension of the cooling solder draws the component closer to the pads. The solder bead squeezes out the side and remains there.*

Various shapes such as ovals, triangles and odd-form shapes have been used to eliminate solder beading with varying success. A common and successfully used aperture reduction method is the home plate design (Figure 6). This reduces the amount of paste printed and helps keep the paste from spreading off the pads, where it may ball and become a bead. It is not only the reduction that matters; the paste's location on the pad also is key. For many manufacturers, this may mean bringing in hundreds or thousands of new stencils, which, while costly, may appear efficient when weighed against the cost of high defect part per millions.

**Stencil Thickness**

Another common and viable method to prevent solder beading is by adjusting stencil thickness. Stencils commonly range in thickness from 0.004 to 0.008", with 0.006" often used as a default thickness. The amount of solder paste deposited through a stencil (depending on several factors, especially snap-off) is 0.001" less than to 0.002" more than the stencil thickness. Obviously, reducing stencil thickness also will reduce the solder volume deposited on the PCB. As the use of high-solvent, low-solids, no-clean solder pastes have become more prevalent, many manufacturers have had to get new stencils cut in thinner dimensions to cure/prevent solder beading. Another option is to use a "step-down" stencil, which offers a reduced thickness in certain areas to reduce paste deposition in key locations.

![Click here to enlarge image](image2.png)

*Figure 6. The homeplate aperture design reduces the amount of paste printed and helps keep the paste from spreading off the pads, where it may ball and become a bead.*

Stencil thickness and aperture dimension must be considered in conjunction with one another as well as the components and PCB design in use because aperture length, width, thickness and shape can impact paste release. Solder manufacturers typically can 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 PCB's top and the stencil's bottom during the print cycle. While sometimes used to aid solder paste release 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 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 PCB that helps eliminate paste bleed. As with all settings, however, a print gap may exist even if the printer is set at zero snap-off because of PCB, stencil or equipment irregularities. It is always critical to verify settings manually after programming.

**Paste Viscosity**
Lighter bodied solder pastes are more likely to exhibit solder beading than heavier bodied pastes. Increasing solder paste viscosity may reduce solder beads because the thicker paste will be more resilient to being squeezed out the sides of components. Unfortunately, as with adjusting the squeegee speed and pressure, increasing paste viscosity is a defect "bandage" that could cause printing difficulties, while avoiding 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 refloows there form a solder bead that comes out to the side of the component. Adjusting the profile to the paste manufacturer's recommendation may rectify the issue.

*Figure 7. The LSP Profile allows paste to outgas at a slower rate, reducing the impetus to expel paste from the main deposit.*

Additionally, using an altered reflow profile has proven to aid in solder bead reduction/elimination. It has been theorized by some that solder beading often is 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. The newly developed profile shown in Figure 7 allows the paste to outgas at a slower rate, thus reducing the impetus to expel paste from the main deposit.

**Metal Load**
Solder pastes with higher metal contents (e.g., 90 vs. 89.5 percent) 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 paste's increased metal content 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 μm
325/+500) powder is less likely to form solder beads than 25 μm (-500/+625) powder when used under the same conditions. Generally, larger powders exhibit less oxide content than finer powders because of less overall surface area. This oxide reduction can translate into a solder beading reduction.

**Test Methodologies**

Although solder beading generally is application-related, the solder paste in use may influence it. 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 result in the maximum amount of solder beading. Some solder paste manufacturers use test methods to develop pastes that significantly reduce solder bead formation. 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 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, reducing the effects that a long low soak might have in reducing the beading. These variables help determine which paste formulations tend to form beads. Then it is simple process to qualify each board and corresponding paste by bead quantity and size.

**Conclusion**

Solder beading is a phenomenon that may be caused by several contributing factors. In nearly every case, however, solder beading can be reduced or eliminated by decreasing the volume of solder paste deposited on the PCB. Additionally, solder paste location on the pad is important. A viable method to prevent solder paste is through revised stencil design, wherein stencil thickness is reduced, step-down areas are used, and apertures are reduced in size and shape.

Kevin Pigeon may be contacted at AIM, 25 Kenney Dr., Cranston, RI 02920; (401) 463-5605; Fax: (401) 463-0203; E-mail: kpigeon@aimsolder.com; Web site: www.aimsolder.com.

*Surface Mount Technology* November, 2001

**Links referenced within this article**

Find this article at: [http://smt.pennnet.com/Articles/Article_Display.cfm?Section=Archives&Subsection=Display&ARTICLE_ID=125633](http://smt.pennnet.com/Articles/Article_Display.cfm?Section=Archives&Subsection=Display&ARTICLE_ID=125633)