

## Solder Beads - What to do about them

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

### What are solder beads?

Solder beads are often confused with solder balls on the surface of a surface mount technology (SMT) board. Solder balls are small balls of solder that are randomly scattered across the surface of a SMT board. They can appear on the solder mask or on the board metallization and they are present around all different component types. Solder beads comprise a specific category or subset of solderballs. Solder beads are solder balls that occur in close proximity to chip resistors and chip capacitors. A typical example of a solder bead is shown in Photo 1.



Photo 1

### How are solder beads formed?

There are several steps in the formation of solder beads. The first step is when the solder paste deposit starts to slump as the paste is heated in the reflow oven. If there is too much paste or the pads are too close together the paste can bridge between the adjacent component lands. When the solder paste reflows or melts the solder pulls back to the pads but in the process a small solder ball or solder bead can break off the main mass of the solder. This solder bead can be located closer to one land than the other but often times the solder bead appears equidistant between the two pads on one side of the component. The process of solder bead formation is shown schematically in Diagram A.

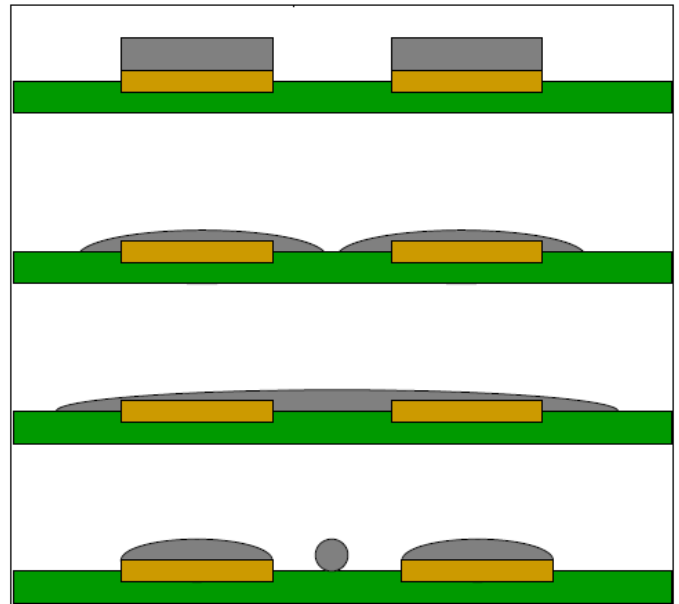


Diagram A.

### Why do solder beads form?

There are three primary problem areas that contribute to the formation of solder beads. These problem areas are correctable at the manufacturing level. This discussion will focus on the problem areas and what can be done in the manufacturing area to overcome them.

The first condition is that there is too much paste being deposited. When solder paste is deposited a stencil of a given thickness is used. The stencil thickness and the aperture opening determine the amount of solder paste deposited. If the amount of solder paste deposited is too much for the size of the pads on the circuit board solder beads can form.

The second and third conditions that can contribute to the formation of solder beads have to deal with the solder pad design. If the pads are too close together or too large for the intended component, solder beads can form (Diagram B).

The shape of the solder pad also plays a role in solder bead formation.

Other factors that can contribute to the formation of solder beads can include; solder paste rheology, reflow

profile and ambient conditions. These factors are often not changeable. Even if minor adjustments are possible their over-all contribution to solder bead formation is minimal and will not be addressed in this discussion.

### Excess Solder Paste

Excess solder paste is probably the single biggest contributing factor to the formation of solder beads. Fortunately this is a correctable condition. Solder paste is deposited on the pad by pushing the paste through a stainless steel stencil. The thickness of the stencil and the cross sectional area of the stencil opening determine the volume of paste printed. The first consideration is the stencil thickness. A 5 or 6 mil thick stencil puts down an adequate amount of solder paste to form a good solder fillet. If a 7 or 8 mil stencil is being used, the assembler is most likely putting down too much paste. Changing to a 5 or 6 mil stencil may be all that is needed to alleviate the problem. If the assembler is already using a 5 or 6 mil stencil and still has solder beads, doing an aperture reduction on the stencil can reduce the amount of paste deposited. Aperture reduction involves reducing the cross sectional area of the opening or aperture in the stencil. Representations of stencil thickness reduction and aperture reduction to reduce solder paste volume are shown in Diagram B.

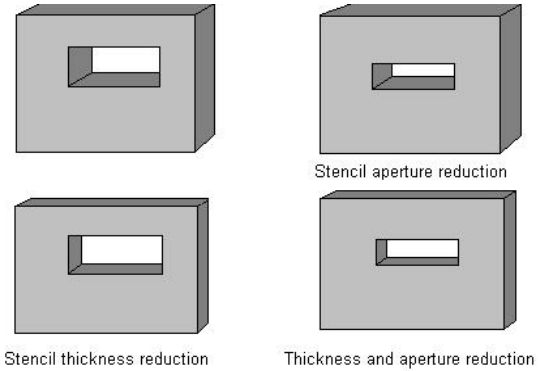


Diagram B

### Solder Pad Placement

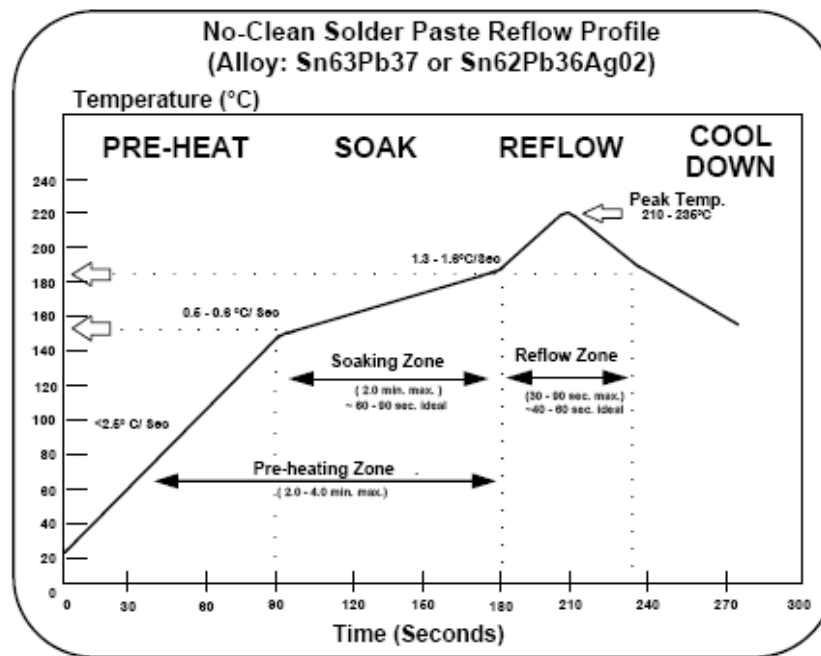
Another potential cause for the formation of solder beads is can be traced to the placement of the pads on the surface mount board. Pads that are too close together relative to the size of the component can contribute to solder bead formation.

### Solder Pad Size

The size of the pads relative to the size of the component can also be a contributing factor to the formation of solder beads. Pads that are too large relative to the size of the component metallization also contribute to solder bead formation.



## 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.