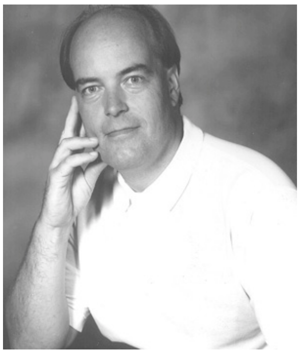


Solder Balling? – No It's Called Solder Beading!!!

Bob Willis

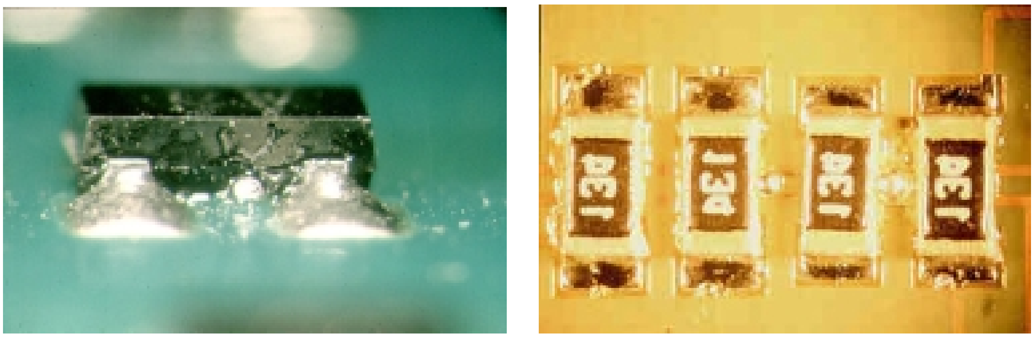
First what is a Solder Bead?

The term solder bead is used to differentiate it from solder balls. A solder bead is a solder ball, but its location is normally constant unlike solder balls. Solder beads are located on the side of chip components between the terminations and away from the lands as illustrated in some of the photographs included in this text.

Solder beads are seen more commonly with no clean processes due to the fact that there is no cleaning process to wash them away. The solder beads are normally 0.010" in size but this can vary depending on the amount of solder paste available to form them. On some occasions there can be more than one ball at any one location, possibly on each side of the chip.

Solder beads can also be found at the side of SOT89, SOT23 and on leadless ceramic chip carriers. Many of your production and quality engineering colleagues look to a specification for guidance on the acceptable size of a ball as they are very difficult and time consuming to remove. IPC does define what is acceptable and rejectable. A quick reference guide to size is available and produced by the author as part of the PPM Monitoring Project run for the SMART Group, go to downloads at www.ppm-monitoring.com

The real solution, find the cause, that will help you to eliminate the problem.



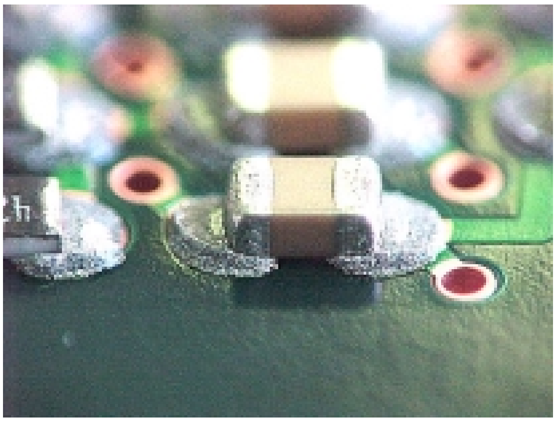
Example of solder beads on a SOT23 and chip components

The solder bead is formed when solder paste is present under a component and can not form a joint at the solderable land or component termination. Solder paste can become trapped under the body of a component and then when the paste particles reflow it starts to coaless together to form a larger ball. Normally speaking the solder paste at the end terminations has already reflowed and wet the joint so the paste under the device can not form due to the limited space available under the part. This causes the liquid solder to flow from under the part at this point the solder is free to form a bead and hence the defect. You can see the solder beading phenomena demonstrated on a video tape titled "***Reflow Soldering and Temperature Profiling***" or on the interactive CD-ROM "***Surface Mount Process Defect Guide 2***" both produced by the author. A version of the video is available to on line with RealPlayer at www.bobwillis.co.uk/video/bobwillis016.rm

After explaining what it is and how it forms let us together consider the causes and corrective action. Stencil Printing can cause the problem if the incorrect process parameters are used. The stencil aperture should be slightly reduced from the pad size 10% reduction is quite a common rule in our industry. As we go down in component geometry the size reduction does change and on 0201 the size reduction may be zero.

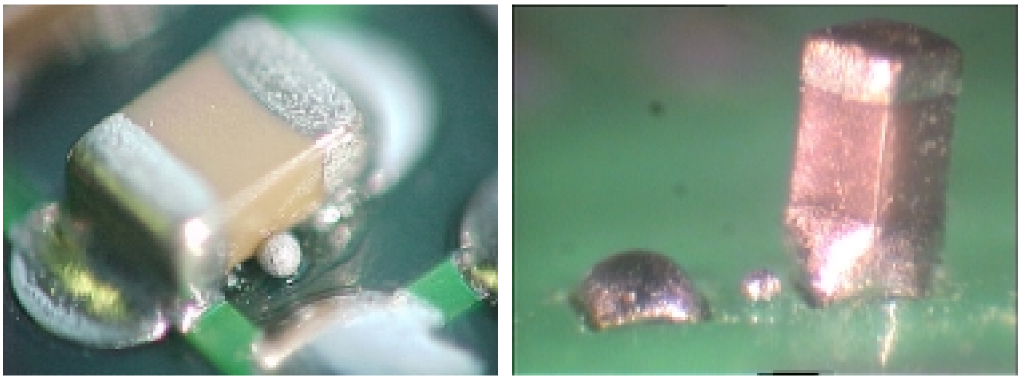
The stencil should ideally make contact with the surface of the pad before and during printing this prevents paste being forced under the stencil causing it to be transferred to the surface of the next board. This correct initial set-up prevents the need for high squeegee pressures it also reduced the rate of under stencil wiping. Excess pressure will cause you problems paste is forced under the stencil and possibly where parts are to be placed. The solder finish and the thickness of the solder resist can have an impact on this part of the process.

It is normal now to use a 0.006" stencil in manufacture today, dropping down from 0.008" in the past. Increasingly engineers are using 0.005" for fine pitch and even 0.004" but this stretches all the tolerance on component coplanarity and PCB flatness to the limit. Using a 0.006" without steps is the limit for good design with pin in paste reflow. The thicker the paste deposit the more likely beading is to occur.



Placement can cause the paste to be forced under the component body. Setting excessive placement force will displace solder paste between the component and resist coating. During reflow the beads will form. Removing components before reflow in areas where beads have formed in the past you can generally detect if this is the cause of the problem. The same thing can occur if you have a variation in the paste height. With the same placement force if the paste is higher you get the same end result.

Reflow Soldering inspection is where the solder beading fault is generally first detected. In-process inspection after reflow will allow you to detect the rate of solder beading. In the case of reflow it is the rate of paste slump which causes the solder paste to be present under the parts. When evaluating solder paste a key factor is paste slump which can occur during the early stages of pre heat or just before the paste moves into the reflow stage.



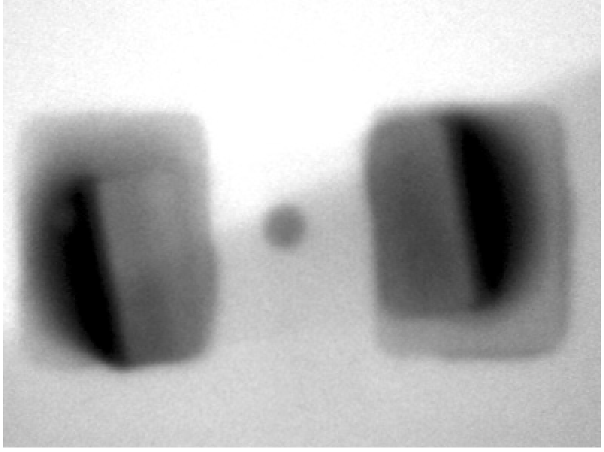
The most common place to see solder beads are at the side of a chip components like resistors and capacitors. It can however be seen on other parts and lift small 0201 chips, two defects for the price of one!!

Simple reflow tests using a board which has been printed and components placed can help you understand the process. Placing the board on a hot plate using similar profile parameters as the oven will allow the whole sequence to be viewed. An alternative is to take a populated board and pass it through the reflow oven without the final reflow spike. This can be achieved by lowering the final zone temperature which will give the sample board the complete pre heat cycle without reflow. The sample board can then be examined by removing selected chips and examining the amount of paste under the parts.

Solder slump tests on paste is a standard paste test you should conduct using a ceramic tile and printing small dots of paste. A measure of the diameter of the paste is made before and after heat exposure. As the paste does not reflow you can measure the worst possible slump for a given paste and oven profile. This test is normally refereed to as hot slumping.

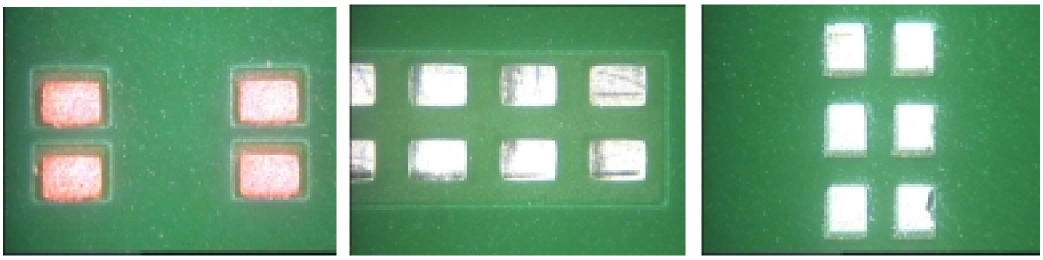
As previously stated the oven profile and the point when the flux activates can also increase or decrees the amount of solder beads. That is why so often changing a paste can reduce the problem. This demonstrates that the original profile used was incorrect and poorly specified by the engineering staff. Remember that a poorly set-up oven profile can make the best paste on the market look poor.

Solder paste particles rely on their initial surface cleanliness and the activity of the paste to allow them to reflow and coaless in to a single ball. If the activity is low then it is quite possible for any paste to leave multiple balls at the side of chip parts. Inevitably as the components get smaller the distance between terminations on the chips becomes smaller. This may increase the presence of solder beads, beading has been seen with 0201 and lead-free alloys.



Solder bead that has not escaped from under the chip, give time it may have put in an appearance

The solder mask thickness between the chip and board will exaggerate the problem as a limited space is available to allow the paste to flow back to the original print area. Leaving the solder mask in place but decreasing the web width works well as does the correct pad size and width in the first place. From a design point of view solder mask under chip components is only required if a track is placed under a part. If there is no mask the paste will flow back to the termination points more readily. Solder mask apertures are a necessity around pads when a track or via hole is present under a component. Some companies have suggested printing advantages of the defined pad for smaller components. Resist defined pads does increase solder bead formation.



Here are the different approaches to pad and resist definition

The gap between the chip/solder mask/laminate surface can change due to:

Resist thickness across the board and from batch to batch
Plating thickness of the copper on the foil
Alternative solderable surface plating
Original copper foil thickness on the PCB
Etching of the copper can decrease the pad area

There are many variable but a solution can be found to eliminate beading, an alternative is to use lower magnification and not see them as many people do?????

Bob Willis is a process engineer providing engineering support in conventional and surface mount assembly processes. He runs production lines for suppliers at exhibitions and also provides seminar and workshops world wide. Bob has one of the largest collections of training videos, interactive CD-ROMs and training material in the industry. Bob will be presenting four Master Classes at APEX in California, he will also be presenting classes at SMT Nuremberg in Germany for those engineers visiting the show. For further information on how Bob may be able to support your staff contact him via his web site **www.bobwillis.co.uk**