Vitronics Soltec

By G. Schouten

Solderball formation in reflow soldering

Information Sheet 111

Introduction

Solderpaste is used for joint formation in reflow soldering.
The paste consists of solderparticles bonded by a flux.
Although this is a simplification, basically that's the system that is deposited at the joint areas before soldering.
The solderlumps should be deposited as a coherent dot without loose particles.

Paste deposition

In most cases solderpaste is deposited by a screen printing process.
The paste should be so deposited that it behaves as one small lump of paste for every joint.
If in this process some paste is deposited outside that lump it can form solitaire solderballs as the paste starts to melt. Also the paste should be deposited on the metal part that should be soldered. If paste is deposited outside the joint area it might not be able to be part of the solderjoint.

Component placement

The lump might also be separated by the placement of the component in the solderpaste. Displaced paste particles might in that case not be able to coalesce with the original lump and create a separate solderball.

Too high amount of paste at the connection

It is also possible that the solderpaste during the placement of the component is squeezed underneath a chip component. During reflow this paste will then create a solderball that is squeezed out from the side of the gap between component body and substrate.

Paste quality

Another reason for solderball formation might be that the paste contains too much oxide. In that case the flux is unable to melt all the solderparticles together and a ring of often small satellites will surround the solderjoint.

Preheating rate

If during the solderprocess in the preheat stage the temperature rises at a too high rate, the volatiles from the flux will escape so fast that they can displace the solderparticles, so that they will separate from the original lump. These solderparticles will during melting create separate solderballs.

Should solder balls be removed?

According to the ANSI/IPS-S-815B document § 3.6.6 solder balls are allowed if not greater then 0.13 mm diameter. They should not violate minimum electrical conductor spacings and the amount of solder balls should not exceed five per square inch.

If we assume that the maximum solder ball is adhering then this ball has a weight of 1.15 microgram. The adhesion force is normally much stronger than the weight of the solderball, which makes it difficult to remove them with brushes.
This is especially the case when there are also SMD's on the solder side.
The brushes may then not even touch the solder ball. A matter of even greater concern is where do the solder balls go? They may be wiped underneath components where they are just fixed in a wedged gap.
During operation of the board this gap will change due to thermal expansions. This may give pressure on the solder ball and this may cause damage to the solder resist. What is underneath? Another possibility is that the ball comes free, now what will it do?
It is good to know that such small solderballs that are fixed to the solderresist can withstand an acceleration of 40g, before they get loose from the solderresist.

All together it is more safe to leave the allowed balls where they are, or better, prevent their formation by optimising the process.