

## TYPICAL DEFECTS

1. Solder Balling
2. Solder Beading
3. Solder Balls caused by printing failures
4. Poor wetting
5. Open solder joint
6. Short circuits (bridges between near by pads)
7. Insufficient solder on soldering pads
8. Components movement/Insufficient tackiness
9. Tomb stoning (Manhattan) effect
10. Wicking

## 1. SOLDER BALLING

The solder particles do not melt evenly because the oxide layer present on them has not been completely removed or the surfaces are not wettable.

Possible reasons: Reflow profile not suitable i.e. Initial ramp too steep preheats too high/too long, peak too high.

Solder paste not active enough.

Solder pads or component leads have poor solderability.

The solder powder is too heavily oxidised.

Incorrect storage conditions for the solder paste.

Solder resist incompatibility.

## 2. Solder Beads

Caused by solder particles being present underneath the components Prior to reflow.

Possible reasons: Reflow profile not suitable, initial ramp too steep.

Too much solder on the pads caused by incorrect stencil specification.

Excessive solder paste slump.

Excessive placement pressure.

Incorrect print registration.

### 3. SOLDER BALLS CAUSED BY PRINTING FAILURES

Solder paste is forced onto non-wettable surfaces such as the solder Mask.

- Possible reasons:
- Bad seal between the stencil and the PCB during printing (gasketing).
  - Mismatch between the stencil and the PCB.
  - Excessive squeegee pressure.
  - Poor aperture to pad size relationships.
  - Contaminated underside of stencil.

### 4. POOR WETTING

The oxide layer has not been removed completely from the surfaces to be soldered.

- Possible reasons:
- Solder paste is not active enough.
  - Solder pads or component leads are not solderable or contaminated.
  - The solder powder is oxidised or is too fine for the application.
  - Reflow profile is not suitable i.e. Preheat too high/too long or peak too high.

## 5. OPEN SOLDER JOINTS

Solder has only wetted to one surface.

Possible reasons:      Bad solderability of the component soldering surfaces due to oxide or contamination.

SOICs, PLCCs and QFPs, often coplanarity is the problem.

Solder paste not active enough.

Poor reflow profile does not allow all surfaces to come up to reflow temperature.

Not enough paste present.

## 6. SHORT CIRCUITS (BRIDGES BETWEEN PADS)

Possible Reasons:      Excessive solder paste slump.

Reflow profile not suitable i.e. Initial ramp rates too steep.

Soldering pads too big relative to gap between pads.

Too much solder on the pads due to incorrect stencil specification.

Solder paste not active enough.

Bad seal between the stencil and the board during printing.

Mismatch between the stencil and PCB.

Poor component placement or poor component leg to PCB pad size relationship.

## 7. INSUFFICIENT SOLDER ON SOLDER PADS

- Possible Reasons:
- Excessive solder paste slump.
  - Reflow profile not suitable i.e. Initial ramp too steep.
  - Solder paste not active enough.
  - Stencil too thin.
  - Printing speed/pressure incorrect.
  - All solderable surfaces such as legs and pads not up to reflow temperature.
  - Metal % of solder paste too low.

## 8. COMPONENT MOVEMENT/INSUFFICIENT TACKINESS

- Possible Reasons:
- Too long a period between printing and placement.
  - Incorrect vehicle selection.
  - Incorrect placement pressure.

## 9. TOMB STONING (MANHATTAN) EFFECT

Instead of being soldered, one end of the component is pulled away from the surface of the PCB either as a result of the mechanical displacement of the component itself or else by the surface tension of the solder that has properly wetted the other end of the component. The problem is only seen with the chip resistors and capacitors.

- Possible Reasons:
- Asymmetry of:
    - geometry's (PCB and/or chips)
    - Heat capacity
    - Solderability of the opposite soldering surfaces

The component was incorrectly placed, or has been displaced.

Reflow profile not suitable i.e. Initial ramp too steep.

Uneven application of solder paste.

Solder paste not active enough.

Poor profile results in uneven temperature distribution.

## 10. WICKING

Wicking is the term to describe the phenomenon in which the solder travels up the component lead.

The metallic surfaces (component leads) warm up faster than FR4 or solder paste.

The component lead is then used like a wick by the solder (hence the name).

This phenomena occurs mostly with J-Leads (PLCCs).

To avoid this problem it is necessary to minimise the temperature differences between all solderable surfaces on the board.