Module 7... Continued

Component Assembly, materials for assembly and joining methods in electronics

- Failures library, thermal profile (continued..)
- Solders: lead-free

Reflow process

5% PTH
5% Connectors
EM device
90% SMD

CBGA Components
SMT Failures Library-
Non or Insufficient solder

- Insufficient solder will induce risk of joint failure during mechanical and thermal stress on PCB.
- Poor dispensing process of solder paste; stencil block could be a reason.
- Ensure stencil/screen printing process is perfect. Do not allow to dry the solder paste on the stencil printer.
- Rework has to be done to re-mount the component, manually.

Figures courtesy: www.smt.cn
SMT Failures Library-
Tomb stoning and Skewing

- Also called Manhattan effect, can be observed during reflow process where the chip components are lifted and stand on one end terminal.
- A variation of this is skewing.
- Caused by unequal soldering conditions on the two solder joints, either due to different melting temperatures and times, or due to volume of solder paste dispensed. Reflow in nitrogen atmosphere has seen an increase in this phenomenon.
- Ensure that the pad layout is correct and check the thermal profile for reflow soldering. Check the dispensing volume of solder paste on the pads.
- $T4 > T1 + T2 + T3$ (force)

Figures courtesy: www.smt.cn
SMT Failures Library - Excess Solder

- There is too much solder alloy than required for the joint. This condition can also cause mechanical stress on the component as well as the joint.
- Study the pad layout and use suitable pad sizes.
- Seen mostly in wave soldering process. Excess drag could be the reason.
- Design the layout for acceptable manufacturing and assembly.

Figures courtesy: www.smt.cn
SMT Failures Library-
Solder Bridging

- This is a defect seen as a mechanical bridge formed by the solder alloy after soldering resulting in an electrical short.
- Can be seen in wave and reflow soldering.
- Seen in fine pitch components.
- Check screen printing process during reflow soldering process—smearing of paste, solder paste lump formation, excessive solder paste etc. Dirt, hair and other foreign particles can cause bridge.
- In wave soldering, component orientation could be the reason for wave drag. Use solder thieves at corners to avoid this problem.

Figures courtesy: www.smt.cn
SMT Failures Library-
Missing Component

• Reasons could be the following:
  – High speed of pick and place machine used for placement
  – Poor adhesion of component on solder paste
  – Component blown out during reflow soldering

• Check the following:
  – Placement speed
  – PCB support
  – Solder paste condition - tackiness

Figures courtesy: www.smt.cn
SMT Failures Library-
Solder Beads

- Beads are seen on the board after reflow soldering process.
- Caused by the presence of too much solder paste at PCB pads, inaccurate solder paste print process and also due to outgassing of solvents from solder paste material during reflow.
- Work on the thermal profile, check the solder paste content, and check on the screen printing process (aperture type).

Figures courtesy: www.smt.cn
Voiding is caused by out-gassing of solvents in the solder.

Flux material could be the reason for this defect. Flux is present in the solder paste.

Poor thermal profile for such solder pastes can cause voids. Soaking zone time has to be increased.

Increase pre-heat time in the reflow profile and short peak reflow times. In this case, nitrogen atmosphere could be useful.
Wrong placement - upside down.
- Mostly seen in resistor placement process.
- A flipped chip capacitor may not be recognizable and may not cause problems.
- Less insulation between the component and track could be an issue if placed upside down in the case of resistors.
- Rework on the component by manual desoldering and removal.
- Place a new component.

Figures courtesy: www.smt.cn
- The component is shifted due to outgassing from a nearby large capacitor (tantalum material).
- Caused by absorption of moisture in the component which outbursts during heating. The force is so powerful enough to displace the nearby component.
- Low temperature baking of certain components is advised to rectify this situation.
- Plastic casings of packages do absorb moisture.

Figures courtesy: www.smt.cn
This is an X-ray picture of a soldered BGA device.

Some areas are soldered well, some (centre) are poorly soldered, some are shorted too.

Causes could be insufficient solder paste dispensing for all the BGA pads.

If additional solder paste is not used, then the defect could be due to BGA "popcorning" effect (delamination) caused by trapped moisture in BGA outgassing during reflow process.

Check solder paste material, BGA ball material, remove moisture by pre-baking, or setting your thermal profile for more pre-heating times.

Figures courtesy: www.smt.cn
Inspection and Testing

This is a key area in board assembly cycle. Special test pads for different nets should be provided while designing for in-circuit testing.

Bed of nails is not often useful.

Flying probe testers are required

• Double sided probing
• Unrestricted use of probes
• Multiple guards
• High fault coverage
• High speed and productivity
• Fast, automated programming

Design for Testability

ATE
Soldering Process using Sn-Pb, Sn and Pb-free materials

- Sn only
  - Protects Cu
  - Good wetting
  - Reasonable solderability
  - Poor finish (dull appearance)
  - Can do immersion plating
  - Needs alloying when used as Pb-free main ingredient
  - Used along with Ag, Cu, In, Bi, Sb etc
  - Sn is a major component in electronics soldering today
  - **Forms whiskers when used alone without any alloying**
Tin Whiskers

- Different from ‘Dendrites’ seen in Zn based plated surfaces
- Length up to 10mm, but typically 1-2mm
- Diameter from 10-150 microns
- Caused by
  - residual stresses within the tin plating
  - Intermetallic growth
  - Scratches or nicks in the plating surface induce whiskers
  - CTE mismatches between the plated surface and base
- Growth rate 0.03 to 0.9mm/yr
- High risk of component solder joint shorting
- Avoid pure TIN plating finishes or lessen the % of Tin in alloy
- Contamination from water used in plating/finishing; atmosphere

Figure reproduced from: Kurt Jacobsen; The Guardian, Thursday 3 April 2008
Soldering Process using Sn-Pb, Sn and Pb-free materials

• Sn-Pb and others
  – Sn-Pb eutectic melts at 183°C
  – As per phase diagram of Sn-Pb the eutectic composition of Sn-Pb is 62:38 but impurities like Bi and Sb are always present in small quantities; hence the common practice is to say 63:37 but there is hardly much difference with 62:38
  – At the eutectic temperature, the alloy has the maximum tensile strength, shear strength, impact strength and resistance to fatigue and creep/crack

  \[ \times \text{Sn-Pb} \]
  \[ \times \text{Sn} \]

• Sn-Pb-Ag eutectic is used for SMD assembly to improve wetting of the solder joint. Ag also provides a grain boundary barrier for intermetallic growth to some extent. Its MP is only 179°C

• In lead-free solders; Sn-Ag-Cu is used where the copper percentage is only about 0.5-1.0% and the Ag% will be ~5%

• In the above ternary mixture, Ag and Cu do not react much; they react with Sn separately to form intermetallics like Ag\(_3\)Sn and Cu\(_6\)Sn\(_5\) which are found to strengthen the alloy by building resistance to stress induced crack or creep.

\[ \text{Sn-Ag-Cu} \]
\[ \text{Other alloys} \]