Component Assembly, materials for assembly and joining methods in electronics

- Soldering-continued
- Failures library, thermal profile

Soldering methods
1. Wave
2. Reflow (hot air/IR)
   - Vapor phase reflow
   - Laser reflow

Solder Types
- Sn/Pb
- Sn/Ag
- Sn/Ag/Cu

SMD/PTH
- Design Issues
Vapour phase reflow soldering

Very good technique to avoid heat shocks

Every component, depending on its size and nature will have different “heat capacity”

Exposing all components to a heat source does not assure “uniform” heating..... Smaller components with lesser heat capacity will experience more effects

In vapour phase heating, all components pick up heat corresponding to their heat capacities, and therefore, no heat shocks- no “pop corning” effect
Vapor Phase Soldering
Used only for SMDs

Perfluoropolyether (PFPE)

Ideal for solder alloys with MP of around 220°C since PFPE boils at 230°C
Ideal for BGAs to avoid “popcorn effect” (delamination)

Ideal for lead-free alloys since they melt at higher temperatures
No overheating is possible
VPS is now becoming popular for BGAs.

Fig: CEDT doc library
Cleaning after assembly……why?

* Flux is corrosive; long term contact will lead to corrosion

* Cleaning after assembly is a major issue in SMD soldering due to small stand off height of components such as PGAs, BGAs and micro BGAs etc

* Ultrasonic cleaning may be used but can be detrimental to micro components and solder joints

* Pressure cleaning with solvent like IPA/acetone is adopted.
Design for Manufacturing Issues

Component orientation

Longitudinal axis of all components parallel or perpendicular

Placement machines loose speed at other angles
Clearance between components

Placement machines hold the components either by “vacuum pens” or “claws”....allow space for “multiple heads” to operate simultaneously.

100 mil minimum

Diagram showing clearance between components.
Considerations for good soldering

Good clearance between foot prints

Too close

0.60 mm minimum
1.2 mm preferable

Ideal foot print will cool evenly, centering the component
BGA soldering steps (typical)

• Pre-bake component if required (moisture)
• Dispense solder paste on PCB footprints
  – Sometimes no solder paste is used. The BGA solder balls reflow and join to substrate.
• Place BGA package and align carefully
• RT tacky cure (if recommended)
• Reflow solder (by IR or hot air or VPS)
• Clean and Test
Rework/ Desoldering BGA

• Apply liquid solder flux on the sides of the package.
• Preheat the package from both top and bottom. Heat can be given from the bottom using a heater while heat from the top can be given using a Hot Air Rework System.
• Now using the correct BGA Nozzle, provide appropriate heating on top of the BGA Package.
• The solder balls underneath the BGA package will melt. Pick up the package with a tweezer or using vacuum pick-up.
BGA Ball Desoldering
BGA rebailing methodology

- Once the balls from the BGA Package is removed, clean the pad and remove any excess solder from the board.
- Apply flux paste or cream on to the pad. Paste or cream flux will help solder balls to stick so that they do not fall or change position.
- Place the new solder balls very carefully on the pad using some kind of stencil that matches the ball dia etc.
- Apply hot air or use your rework system with a suitable thermal profile defined. Temperature and time are key parameters for a good reflow process.
- Solder balls will melt and get soldered.
- Remove the stencil and your BGA is reballed now.
BGA Reballing
BGA assembly or soldering
<table>
<thead>
<tr>
<th></th>
<th>Defect Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Spattering</td>
</tr>
<tr>
<td>2</td>
<td>Large solder balls next to pad</td>
</tr>
<tr>
<td>3</td>
<td>Tomb stoning</td>
</tr>
<tr>
<td>4</td>
<td>Bridging</td>
</tr>
<tr>
<td>5</td>
<td>Component cracks</td>
</tr>
<tr>
<td>6</td>
<td>De-wetting</td>
</tr>
</tbody>
</table>
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.
- $T_4 > T_1 + T_2 + T_3$ (force)
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