Technical Note

General recommendations for board assembly of lead-free BGA

Creation	Status:	Released
	Date / Version:	May 2003 / 2.0
	Author:	Gerhard Haubner
	Approver:	Jürgen Winterer / Dr. Raimund Schwarz

Contact	Department:	CAT AIT PGP
	Address:	infineon.goesgreen@infineon.com
		http://www.infineon.com/greenproduct

Summary Recommendations are given for the lead-free board assembly process of ball grid array packages (BGA)

Key Words
BGA
board assembly
lead-free soldering
SnAgCu

Content

1 Target

Infineon has chosen the lead-free alloy consisting of tin (balance), silver (3.0 wt% - 4.0 wt%) and copper (0.5 wt% - 1.0 wt%) as replacement for the eutectic SnPb or SnPbAg alloy at BGA packages. Due to this change, there will be some modifications in the board assembly process of BGAs. This document should supply instructions for our customers for a successful change-over. As this document deals mainly with modified process steps, please refer also to the existing product specific recommendations.

2 Design of Printed Circuit Board

The recommendations for design of Printed Circuit Board (PCB) for Pb-free processing doesn't differ from today's process for eutectic SnPb (resp. SnPbAg) soldering. The details of the used PCB design strongly depends on the board technology, number of metal layers and electrical restrictions. Generally two basic types of solder pads are commonly used: "solder mask defined" (SMD) and "non solder mask defined" (NSMD). Because NSMD pads provide more space for routing and result in a higher solder joint reliability, this type is recommended.[1] As in conventional soldering, the solder pads must have good wettability and solderability at the soldering process. To protect the metal surface, several different preservative layers are used. The properties of some preservatives are listed:

Table 1: properties of some solderability preservative layers at PCBs (Ref.: [2], [3], [4], [5], [6])

finish	typ. layer thickness *in µm*	properties	concerns
HAL / HASL *(Hot Air Solder Level)*	> 5	cheap, widely usage, know how in fabrication	uneven surface, formation of humps, normally not suitable for BGA and fine pitch in general
Electroless Tin	0.3 - 1.2	solder joint is formed only by copper and solder, no extra metal is needed for solder joint	long-term stability of protection may be a concern
Electroless Ni / Immersion Au *(ENIG)*	1-3 / 0.1 - 0.5	very good solderability protection, high shear force values	expensive, concerns about brittle solder joints
galvanic Ni/Au	>3 / 0.1 – 2	only for thicker layers	expensive
OSP *(Organic Solderability Preservatives)*	typical ca. 1 µm	cheap, simple, fast and automated fabrication	must be handled carefully to avoid damaging the OSP; not as good long-term stability as other coatings; at double-sided reflow only suitable with inert gas reflow

The question about the "best" preservative surface can not be answered generally. It depends strongly on board design, pad geometry, components on board or process conditions. The best choice for one application needn't be the best one for another application.

In literature the test results of solderability, wetting force and wetting time for several preservative layers are not always coincident, but in general electroless Ni/immersion Au and OSP turned out to be suitable for BGA assembly. In-house tests showed the suitability (please refer to "Reliability / Compatibility").

The PCB must be able to resist the higher temperatures which are occurring at the lead-free process. This question should be discussed with the PCB-supplier.

3 Solder Paste Printing

The printing process is nearly the same as for standard SnPb-alloys. Lead-free solder pastes have in general the same handling conditions as the used SnPb pastes. The paste must be prevented from extreme heat or humidity, the paste must be at room temperature before using. Old and new paste should not be mixed together in one jar. If you have good results with proper paste handling today, there should be no major problems with lead-free pastes. But you should mind the topic of metal content of a paste:

Solder pastes consist principally of the solder alloy and the flux system. Normally the volume is divided in about 50 % alloy and 50 % flux. In term of mass this means approx. 90 wt% alloy and 10 wt% flux system. If there is an important difference in printing of a lead-free solder paste in contrary to the equivalent tin-lead paste, it may depend on the metal content or the flux system of a paste. In this case the paste vendor should be involved. The paste has to be a "no-clean" solder paste, because cleaning under the BGA is hardly possible. The paste must have the printing capability for fine pitch printing, typically a type 3 paste should be adequate (a).

4 Placement of Components

Components with solder balls have the favourable property of selfalignment As a rule of thumb a maximum tolerable misplacement of the components is ≤ 50% of the PCB pad diameter.

The following remarks are important:

- Especially on large boards local fiducials close to the device can compensate a large amount of PCB tolerances
- Typically for placement of BGA packages are vision systems with ball recognition and alignment with regard to ball locations. This eliminates the solder ball to package edge tolerances (please refer to the respective package outline drawing for details) and is therefore recommended
- To ensure the identification of the packages trough the vision system, an adequate lightning as well as the correct choice of the measuring modes are necessary
- The self alignment effect only takes place, if the ball is above its melting temperature. For mixed assembly (lead-free ball, lead containing solder paste) please refer to chapter 7

(a) Acc. to J-STD-006A [10] a paste of type 3 is defined as follows: at least 80% of the solder particle have a size between 45-25 µm, less than 1% are larger than 45 µm and less than 10% are smaller than 20 µm

5 Reflow Soldering & Solderability

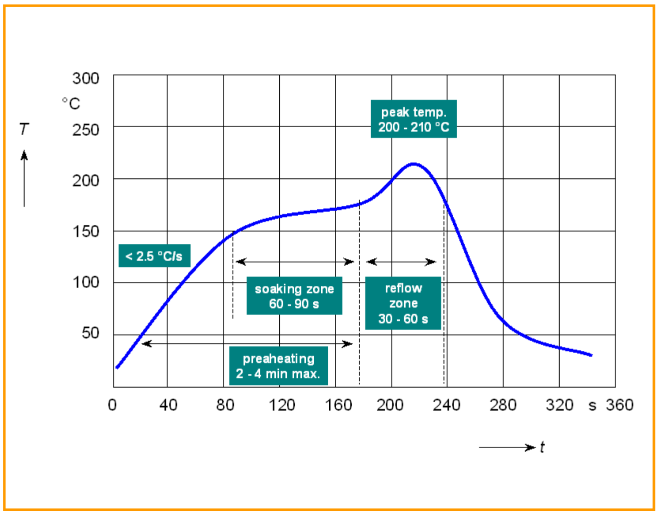


Figure 1: Typical forced convection reflow profile for soldering 62Sn36Pb2Ag

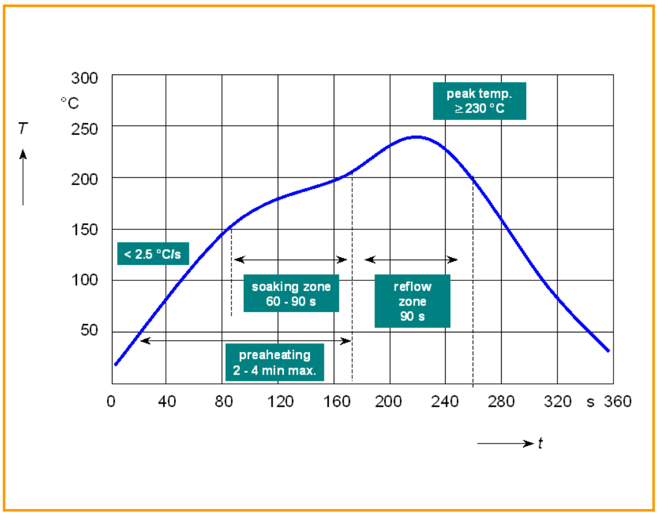


Figure 2: Typical forced convection reflow profile suitable for soldering SnAgCu

Like in SnPb-process, generally all commonly used reflow soldering processes (vapour phase, forced convection, infrared) are suitable for board assembly. But due to a higher peak temperature at the lead-free process, some equipment can reach the borders of their technical capability. Normally reflow with forced convection is chosen.

Wave soldering is for BGA-packages not possible.

At the reflow process each solder joint has to be exposed to temperatures above solder liquidus for a sufficient time to get the optimum solder joint quality, whereas overheating the board with its components has to be avoided.

Figure 1 shows a typical forced convection reflow profile for soldering BGA with conventional eutectic tin-lead solder. This temperature profile also has to be within the specification of the used solder paste and depends on the board and oven.

Figure 2 shows a typical forced convection reflow profile for Pb-free soldering, e.g. with SnAgCu.

Like in conventional soldering measurement of reflow profile has to be done with the temperature sensor(s) beneath the BGA, directly at the solder ball to get the genuine temperature at this position. You don't obtain the correct temperature, if measuring the temperature on top of the BGA or somewhere outside.

Using infrared ovens without convection special care may be necessary to assure a sufficiently homogeneous temperature profile for all solder joints on the PCB (especially on large, complex boards with different thermal masses of the components) including those under the BGA. The most recommended types are therefore forced convection. Due to the demanding conditions at the lead-free process, using nitrogen may reduce oxidation and improve the solder joint quality.

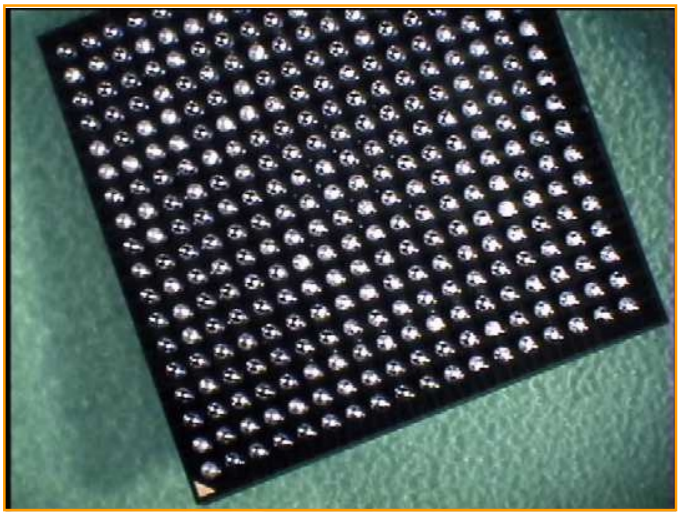
In case of using a vapour phase reflow a pre-heating zone is necessary to prevent components from thermal shock.

Please keep to the barcode label on the reel, tube or cartonage, which serves as a guideline for proper handling and storing to prevent excessive moisture content at the time of board assembly.

6 Inspection & Rework

Inspection

Lead-free solder joints look different than SnPb solder joints. Tin/lead joints have a bright and shiny surface, a dull surface is an indicator for a insufficient solder joint. Lead-free solder joints don't have this bright surface. Lead-free solder joints are dull and grainy. This surface properties are caused by the irregular solidification of the solder, as the used solder alloys are not exactly eutectic (like the 63Sn37Pb solder alloy). This means this SnAgCu-solders don't have a melting point but a melting range of some degrees. Although lead-free solder joints have this dull surface, this does not mean that lead-free joints are of lower quality or weaker than the SnPb joints. This characteristic makes it necessary to instruct the inspection personnel how these new lead-free joints look like, and/or to adjust optical inspection systems to lead-free solder joints. In section "Typical Material Data" there are pictures comparing the appearance of SnAgCu and SnPbAg balls.



Lead-free solder balls can look dull and grainy, and they may be not uniform in their appearance.

Figure 3: appearance of lead-free solder balls

Rework

Defect BGA's with lead-free balls can be reworked like standard SnPb components. Single solder joints cannot be repaired, if a defect solder joint at a component is observed, this component has to be removed and replaced by a new one. The therefore used handling equipment can be used further on. You may have a close look at the rework station to apply the proper amount and type of solder paste before the new component is placed to ensure a good rework result. At the following (rework)- reflow process, the proper melting temperature for the lead-free alloy must be reached at the BGA solder joint.

If a complete reflow of the whole PCB is chosen, be sure that the remaining components on board are handled according to their moisture sensitivity classification. If they are sensitive against humidity and are exposed to room atmosphere, drying of the PCB (plus the components on it) may be necessary (24 h at 125 °C)[16].

If there is only local heating for soldering the replaced BGA, take care not to overheat the PCB or heat up components near to the reworked BGA.

7 Board Level Reliability & Compatibility

The board level reliability is topic of a number of publications and papers (for example Ref. [11] [12] [13] [14]). There the SnAgCu alloy is stated as a suitable replacement for conventional solders and the use of SnAgCu is regarded as successful in terms of reliability.

The reliability of BGAs was also investigated in several studies at Infineon, two investigations are described below:

Investigation 1:

package type: P-LFBGA with SnAgCu balls

testboard: FR4 with electroless Ni immersion Au surface finish

result: TCoB (temperature cycling on board)

-40/+125 °C, 2000x: pass

vibration + mechanical shock: pass

P-LFBGA-208 packages with daisy chain chip were assembled to test boards with SnAgCu paste and SnPbAg paste. All material combinations showed good solder wetting during assembly. Independent from solder paste all assemblies passed these tests. The P-LFBGA-208 with SnAgCu balls is more reliable with regard to solder joint cracking as the standard lead containing LFBGA [15].

Investigation 2:

package: P-LBGA with SnAgCu balls

testboard: FR4 with Cu-OSP surface finish

result: TCoB -40/+125 °C, 1000x: pass

P-LBGA-260 packages were assembled to test boards with SnAgCu and SnPb paste. Independent from solder paste all assemblies passed this test.

One of the most asked questions during the change to lead-free, deals with the compatibility of lead-free packages with lead containing solder paste. The following figure should describe the possible combinations.

Table 2: possible combinations of lead-based and lead-free solder balls and solder paste

paste / ball	SnPb (= low peak temperature) (b)	SnAgCu (= high peak temperature) (c)
SnPb	**ok**	**ok**
SnAgCu	**ok with restrictions**	**ok**

(b) "low peak temperature" = peak temp. for reflow process with SnPb solder pastes (about 200 °C – 210 °C)

(c) "high peak temperature" = peak temp. for reflow process with lead-free solder pastes (about 225 °C – 230 °C)

In respect of material combination it is possible to mix SnPb(Ag) paste with SnAgCu solder balls. Of particular interest is the required minimum peak temperature at reflow for the combination lead-free ball and SnPb solder paste. The following table is the result of an in-house evaluation about this topic:

Table 3: correlation between processability and peak temperature/coplanarity

peak temperature (at solder joint)	**regular coplanarity**
200 °C – 215 °C	no
220 °C – 230 °C	yes with restrictions
235 °C – 245 °C	yes

Below a peak temperature of 215 °C no suitable solder joints are formed.

With peak temperature around 225 °C suitable solder joints are only possible, if the packages have good coplanarity values. Due to reduced self-alignment of the BGA packages at this temperature, the package must be placed well on the solder paste.

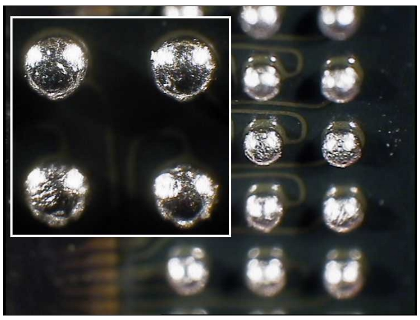
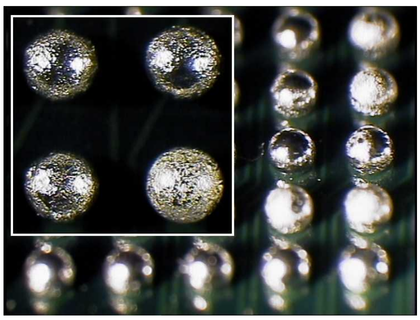
With peak temperatures over 230 °C suitable solder joints are formed.

<u>In one sentence:</u>

The combination lead-free BGA / SnPb paste can be processed with a minimum peak temperature of 220 °C, if the BGA has good coplanarity. Due to the reduced self-alignment effect, this solution should only be chosen in exceptional cases. Peak temperatures over 230 °C at the ball are recommended.

8 Typical Material Data for Tin-Lead and Lead-Free Solder Balls

Table 2: material properties of SnPbAg in contrary to SnAgCu (Ref.: [7], [8], [9])

		62Sn36Pb2Ag	Sn3.5Ag0.75Cu
melting temperature in °C		179	217 - 219
tensile strength in MPa		43 - 48	48 - 53
density in $\frac{g}{cm^3}$		8.4 - 8.5	7.3 - 7.5
shear strength *normalised: 62Sn36Pb2Ag = 1*		1	1.3
electrical resistivity in µΩ·cm		14 - 14.5	11 - 13.8
creep strength in $\frac{N}{mm^2}$ at 0.1 $\frac{mm}{min}$	***20 °C***	3	13
	100 °C	1	5
CTE in ppm/K (Coefficient of Thermal Expansion)		19 – 27	22 – 30
optical appearance		The standard SnPbAg-balls have uniform gloss and reflection. Normally all balls have the same appearance. 	The new lead-free, especially the SnAgCu-balls have different gloss and reflection. They often have a varying roughness on the surface. Generally this balls have a different optical appearance, without influence on processability or reliability. 

9 References

[1] Reliability of BGA Packages in an Automotive Environment; Roger Rörgren et al., IVF - The Swedish Institute of Production Engineering Research

[2] Baugruppentechnologie der Elektronik / Hrsg. Wolfgang Scheel. - Berlin : Verl. Technik, 1999

[3] Going beneath the Surface of Surface Finishes; Don Cullen, circuitree2002

[4] Evaluation of Solderability of Pb-free PWB Finishes with Pb-free Solders; Auburn University, Alabama, 2001

[5] Lead-Free Solderability Preservative Coatings of PCBs; Zofia Morawska, et al., Advancing Microelectronics Volume 28, No. 3, 2001

[6] Brittle Interfacial Fracture of PBGA Packages Soldered on Electroless Nickel / Immersion Gold; Zequn Mei, Matt Kaufmann, et al., IEEE, 1998

[7] vendor information

[8] several in-house experiments

[9] Database for Solder Properties with Emphasis on New Lead-free Solders - Release 4.0; Siewert Thomas, Liu Stephen, Smith David, Madeni Juan Carlos, National Institute of Standards and Technology & Colorado School of Mines, 2002

[10] Joint Industry Standard IPC/EIA J-STD-006A "Requirements for Electronic Grade Solder Alloys and Fluxed and Non-Fluxed Solid Solders for Electronic Soldering Applications", May 2001

[11] Board level reliability of lead-free soldered interconnections; H.-J. Albrecht, K. Wilke, D.S. Brodie; SMTA International Conference, Rosemont, Illinois USA, 2002

[12] Impact of component terminal finish on the reliability of Pb-free solder joints; Gregory Henshall, Patrick Roubaud, Geary Chew; SMTA International Conference, Rosemont, Illinois USA, 2002

[13] Assembly and solder joint reliability of Plastic Ball Grid Array with lead-free versus lead-tin interconnect; Kim-Marie Levis, Andrew Mawer; ECTC 2000

[14] Microstructural Evolution and Damage Mechanisms in Pb-free Solder Joints During Extended –40 °C to 125 °C Thermal Cycles; Dunford, Primavera, Meilunas, IPC 2002; New Orleans, LA, Nov. 2-3, 2002

[15] Board level reliability of lead-free packages; Christian Birzer, Infineon Technologies, 2001

[16] Recommendations for printed circuit board assembly on Infineon P-LFBGA / P-TFBGA packages; customer information, Infineon Technologies, 2002