Lantiq's Overmold P(G)-BGA Packages

PCB Assembly Recommendations

Solution Guide

Revision 1.1, 2013-12-10

Edition 2013-12-10

Published by
Lantiq Deutschland GmbH
Lilienthalstraße 15
85579 Neubiberg
Germany

© 2013 Lantiq Deutschland GmbH
All Rights Reserved.

Legal Disclaimer

The information given in this document is confidential.

SUCH INFORMATION SHALL IN NO EVENT BE REGARDED AS A GUARANTEE OF CONDITIONS OR CHARACTERISTICS. WITH RESPECT TO ANY VALUES STATED AS "TYPICAL" AS WELL AS EXAMPLES OR HINTS PROVIDED HEREIN, INCLUDING THOSE RELATED TO USE AND/OR IMPLEMENTATION OR APPLICATION OF COMPONENTS, LANTIQ DEUTSCHLAND GMBH ("LANTIQ") HEREBY DISCLAIMS ANY AND ALL WARRANTIES AND LIABILITIES OF ANY KIND, INCLUDING WITHOUT LIMITATION, WARRANTIES OF NON-INFRINGEMENT OF INTELLECTUAL PROPERTY RIGHTS OF ANY THIRD PARTY. IN PARTICULAR, THOSE VALUES, EXAMPLES OR HINTS ARE NO WARRANTIES OF A CERTAIN FUNCTIONALITY, CONDITION OR QUALITY OF THE LANTIQ COMPONENT AND ALWAYS HAVE TO BE VERIFIED IN THE TARGET APPLICATION.

The information in this document is subject to regular updates and/or corrections via errata sheet(s) or new revisions. It is only valid as updated and/or corrected. Once a new revision or errata sheet is available, it can be downloaded from Lantiq's Competence Center:

www.lantiq.com/contact-lantiq

Any user of this document must ensure that they only use the latest update- and/or corrected revision.

LANTIQ HEREBY DISCLAIMS ANY AND ALL WARRANTIES AND LIABILITIES OF ANY KIND, INCLUDING WITHOUT LIMITATION, WARRANTIES OF NON-INFRINGEMENT OF INTELLECTUAL PROPERTY RIGHTS OF ANY THIRD PARTY, WITH RESPECT TO PREVIOUS REVISIONS OF THIS DOCUMENT ONCE AN UPDATE OR ERRATA SHEET IS AVAILABLE.

Information

For further information on technology, delivery terms and conditions and prices, please contact the nearest Lantiq office (**www.lantiq.com**).

Warning

Lantiq components must not be used in life-support devices or systems.

Life-support devices or systems are those systems intended to be implanted in the human body or to support and/or maintain and sustain and/or protect human life. If they fail, it is reasonable to assume that the health of the user or other persons may be endangered.

Lantiq's Overmold P(G)-BGA Packages, PCB Assembly Recommendations

Revision History: Revision 1.1, 2013-12-10

Previous Revision: Revision 1.0, 2013-06-24	
Page	**Subjects (major changes since last revision)**
All	Language and layout improvements.

Trademarks of Lantiq

CONVERGATE™, COSIC™, DUALFALC™, DUSLIC™, ELIC™, EPIC™, FALC™, GEMINAX™, ISAC™, IWORX™, OCTALFALC™, OCTAT™, QUADFALC™, SCOUT™, SEROCCO™, SICOFI™, SLIC™, SMINT™, SOCRATES™, VINAX™, VINETIC™, XWAY™

Other Trademarks

ARM™, Bluetooth™ of Bluetooth SIG Inc. CAT-iq™ of DECT Forum. eCosCentric®, eCos® and eCosPro® of eCosCentric Limited. EPCOS™ of Epcos AG. HYPERTERMINAL™ of Hilgraeve Incorporated. IEC™ of Commission Electrotechnique Internationale. IQfact™ and IQmax™ of LitePoint Corporation. IrDA™ of Infrared Data Association Corporation. ISO™ of INTERNATIONAL ORGANIZATION FOR STANDARDIZATION. MATLAB™ of MathWorks, Inc. NUCLEUS™ of Mentor Graphics Corporation. Linux® of Linus Torvalds. MIPS® of MIPS Technologies, Inc., USA. muRata™ of MURATA MANUFACTURING CO. SOLARIS™ of Sun Microsystems, Inc. Samtec® of Samtec Inc. TEAKLITE™ of CEVA, Inc. TEKTRONIX™ of Tektronix Inc. UNIX™ of X/Open Company Limited. VERILOG™, PALLADIUM™ of Cadence Design Systems, Inc. VxWorks™, WIND RIVER™ of WIND RIVER SYSTEMS.

All other trademarks are the property of their respective owners.

Last Trademarks Update 2013-09-20

Table of Contents

List of Figures

List of Tables

Preface

This solution guide provides **P**rinted **C**ircuit **B**oard (PCB) assembly recommendations for Lantiq's overmold P(G)-BGA (**P**lastic (**G**reen) **B**all **G**rid **A**rray) packages.

Organization of this Document

This document is organized as follows:

- **Chapter 1**, **Package Description**
- **Chapter 2**, **Package Handling**
- **Chapter 3**, **Printed Circuit Board (PCB)**
- **Chapter 4**, **Board Assembly**
- **Chapter 5**, **Inspection**
- **Chapter 6**, **Rework**
- **Standards References**

1 Package Description

Table 1 shows the types of overmolded laminate ball grid array (BGA) packages offered by Lantiq:

Table 1 Ball Grid Array Packages

Package	Description	Comment
P(G)-BGA	Plastic (Green) Ball Grid Array Package	**Figure 1**
P(G)-LBGA	Plastic (Green) Low Profile Ball Grid Array Package	**Figure 2**
P(G)-LFBGA	Plastic (Green) Low Profile Fine Pitch Ball Grid Array Package	**Figure 2**
P(G)-TFBGA	Plastic (Green) Thin Profile Fine Pitch Ball Grid Array Package	**Figure 2**
P(G)-FCLBGA	Plastic (Green) Flip Chip Low Profile Ball Grid Array Package	**Figure 3**
P(G)-LF^{2}BGA	Plastic (Green) Low Profile Flip Chip Fine Pitch Ball Grid Array Package	**Figure 3**

Features

- Small package outline
- Area array solder terminals
- Flip chip or wire bond interconnection
- Various ball pitches available
- Possibility of multi-chip packages
- Better thermal performance than with QFPs
- Reference JEDEC Design Standards: Design Guide 4.5 (FBGA) and Design Guide 4.14 (BGA)
- For package height profile codes, please refer to JEDEC JESD30: Descriptive Designation System for Semiconductor-device Packages.

Figure 1 shows the cross section through a wire-bonded plastic overmolded, rigid laminate substrate, ball grid array IC package. The devices are molded as individual units and singulated by punching. The P(G)-BGA package is typically used for larger body sizes (> 17 mm) and higher IO counts (> 208). There is a substrate flange around the mold cap.

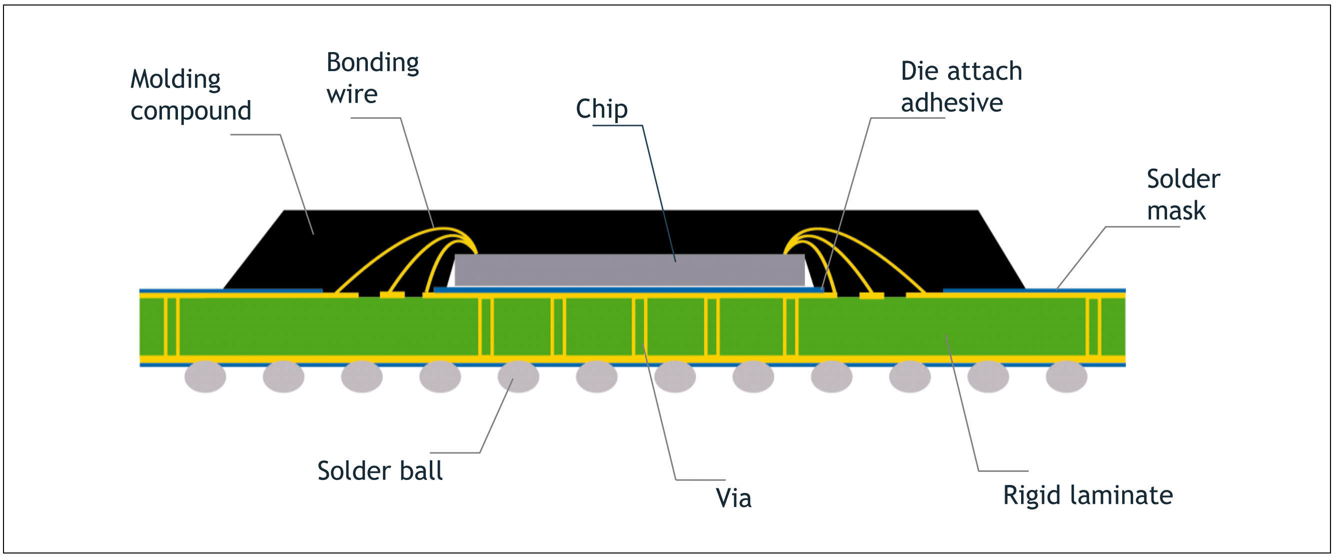


Figure 1 P(G)-BGA Cross Section

Figure 2 shows the cross section through P(G)-LFBGA, P(G)-LBGA and P(G)-TFBGA wire-bonded plastic overmolded, rigid laminate substrate, ball grid array IC packages. These are molded in an array format and singulated by sawing. The mold cap is extended to the substrate edge. The P(G)-TFBGA package only differs in thickness (for details please refer to the package outline drawing).

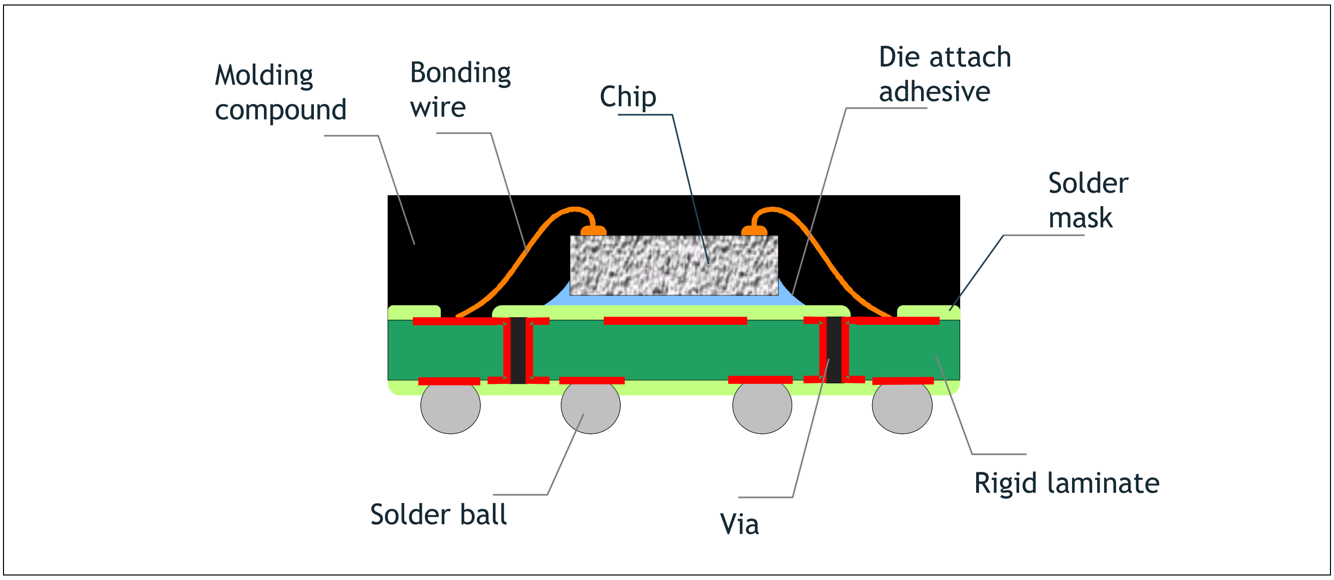


Figure 2 P(G)-LFBGA, P(G)-LBGA and P(G)-TFBGA Cross Section

Figure 3 shows the cross section through P(G)-FCLBGA and P(G)-LF^{2}BGA flip chip plastic overmolded, rigid laminate substrate, ball grid array IC packages. These are molded in an array format and singulated by sawing. The mold cap is extended to the substrate edge. The P(G)-LF^{2}BGA package only differs in the ball pitch (for details please refer to the package outline drawing).

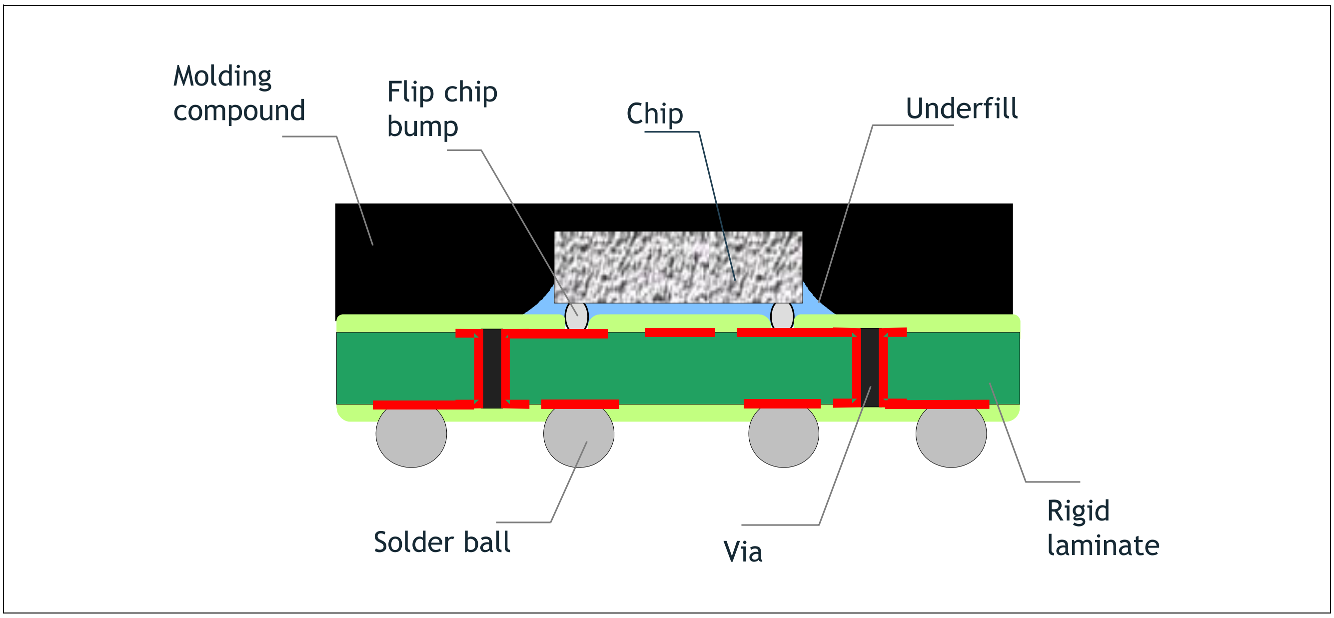


Figure 3 P(G)-FCLBGA & P(G)-LF^{2}BGA Cross Section

2 Package Handling

This chapter is organized as follows:

- **Chapter 2.1**, **ESD Protective Measures**
- **Chapter 2.2**, **Packing of Components**
- **Chapter 2.3**, **Moisture Sensitive Components (MSL Classification)**
- **Chapter 2.4**, **Storage and Transportation Conditions**
- **Chapter 2.5**, **Handling Damage and Contamination**
- **Chapter 2.6**, **Component Solderability**

2.1 ESD Protective Measures

Semiconductor devices are normally electrostatic discharge sensitive devices (ESDS) requiring specific precautionary measures regarding handling and processing. An electrostatic charge may accidentally be discharged via an IC when it is touched by a person or processing tools. This can cause high current or voltage pulses which can damage or even destroy sensitive semiconductor structures. ICs may also be charged during processing. If the discharge takes place too quickly ("hard" discharge), it can cause load pulses and, worst case, electrical damage. Therefore ESD protective measures must ensure that there is no contact with charged objects and that the IC is not charged during processing. These measures cover the handling, processing and packing of ESDS devices. Recommendations regarding handling and processing are provided below.

2.1.1 Workplace ESD Protective Measures

- Standard marking of ESD protected areas
- Access controls, with wrist strap and footwear testers
- Air conditioning
- Dissipative and grounded floor
- Dissipative and grounded working and storage areas
- Dissipative chairs
- Ground bonding point for wrist strap
- Trolleys with dissipative surfaces and wheels
- Suitable shipping and storage containers
- No sources of electrostatic fields

2.1.2 Equipment for Personnel

- Dissipative/conductive footwear or heel straps
- Suitable smocks
- Wrist strap with safety resistor
- Volume conductive gloves or finger cots
- Regular training of staff

2.1.3 Production Installations and Processing Tools

- Machine and tool parts made of dissipative or metallic materials
- No materials with thin insulating layers for sliding tracks
- All parts reliably connected to ground potential
- No potential difference between individual machine and tool parts
- No sources of electrostatic fields

Detailed information on ESD protective measures may be obtained from the ESD Specialist through Area Sales Offices. Our recommendations are based on the internationally applicable standards IEC 61340-5-1 and ANSI/ESD S2020.

2.2 Packing of Components

List of relevant standards that should be taken into consideration

Lantiq packs according to the IEC 60286-* series of standards:

- **IEC 60286-3** [1]
  Packaging of components for automatic handling –
  Part 3: Packaging of surface mount components on continuous tapes
- **IEC 60286-5** [2]
  Packaging of components for automatic handling –
  Part 5: Matrix trays
- **IEC 60286-6** [3]
  Packaging of components for automatic handling –
  Part 6: Bulk case packaging for surface mounting components

Moisture Sensitive Surface Mount Devices

Lantiq packs according to:

- **IPC/JEDEC J-STD-033C** [4]
  Handling, packing, shipping and use of moisture/reflow sensitive surface mount devices

For the package body dimensions, refer to the detailed package outline drawings which can be requested from your Lantiq representative.

Other references

- **ANSI/EIA-481-D** [5]
  8 mm through 200 mm embossed carrier taping and 8 mm & 12 mm punched carrier taping of surface mount components for automatic handling
- **EIA-783** [6]
  Guideline orientation standard for multi-connection package (design rules for tape and reel orientation)

2.3 Moisture Sensitive Components (MSL Classification)

It is necessary to control the moisture content of components that are housed in moisture-sensitive packages. The penetration of moisture into the package molding compound is generally caused by exposure to the ambient air. In many cases, moisture absorption leads to moisture concentrations in the component that are high enough to damage the package during the reflow process. Thus it is necessary to dry moisture-sensitive components, seal them in a moisture-resistant bag and only remove them immediately prior to mounting them on the PCB.

The time for which a component may remain outside the moisture-resistant bag (from opening the moisture-resistant bag until the final soldering process) is a measure of the sensitivity of the component to ambient humidity (Moisture Sensitivity Level, MSL). The most commonly applied standard IPC 1) /JEDEC J-STD-033* defines eight different MSLs (see **Table 2**). Please refer to the "Moisture Sensitivity Caution Label" on the packing material, which provides information on the moisture sensitivity level of the product. IPC/JEDEC-J-STD-20 specifies the maximum reflow temperature that must not be exceeded during board assembly by the customer.

Table 2 Moisture Sensitivity Levels (according to IPC/JEDEC J-STD-033*)

Level	Floor Life (Out of Bag)	
	Time	**Conditions**
1	Unlimited	≤30°C/85% RH
2	1 year	≤30°C/60% RH
2a	4 weeks	≤30°C/60% RH
3	168 hours	≤30°C/60% RH
4	72 hours	≤30°C/60% RH
5	48 hours	≤30°C/60% RH
5a	24 hours	≤30°C/60% RH
6	Mandatory bake before use. After bake, must be reflowed within the time limit specified on the label.	≤30°C/60% RH

If moisture-sensitive components have been exposed to the ambient air for longer than the time specified by their MSL, or the humidity indicator card indicates too much moisture when the package is opened, then the packages have to be baked prior to being assembled. Please refer to IPC/JEDEC J-STD-033* for further details.

Baking a package too often can lead to solderability problems due to oxidation and/or intermetallic growth. Please note that the packing material might not be able to withstand the baking temperature (see the labeling on the packing for the maximum allowed temperature).

2.4 Storage and Transportation Conditions

Improper transportation and unsuitable storage of components can lead to a number of problems during subsequent processing, such as poor solderability, delamination and package cracking effects.

List of relevant standards that should be taken into consideration:

- **IEC 60721-3-0** [7]
  Classification of environmental conditions –
  Part 3: Classification of groups of environmental parameters and their severities –
  Introduction
- **IEC 60721-3-1** [8]
  Classification of environmental conditions –
  Part 3 Classification of groups of environmental parameters and their severities –
  Section 1: Storage
- **IEC 60721-3-2** [9]
  Classification of environmental conditions –
  Part 3: Classification of groups of environmental parameters and their severities –
  Section 2: Transportation
- **IEC 61760-2** [10]
  Surface mounting technology –
  Part 2: Transportation and storage conditions of surface mounting devices (SMD) –
  Application guide
- **IEC 62258-3** [11]
  Semiconductor die products –
  Part 3: Recommendations for good practice in handling, packing and storage
- **ISO 14644-1** [12]
  Cleanrooms and associated controlled environments –
  Part 1: Classification of air cleanliness

Table 3 Storing Conditions

Product	Storage Condition
Wafer/die	N2 or MBB[1] (IEC 62258-3)
Component - moisture sensitive	MBB (JEDEC J-STD-033C)
Component - not moisture sensitive	1K2 (IEC 60721-3-1)

1) MBB = Moisture Barrier Bag

Maximum Storage Time

The conditions to be complied with in order to ensure problem-free processing of active and passive components are described in standard IEC 61760-2.

Internet Links to Standards Institutes

- **American National Standards Institutes (ANSI)**
- **Electronics Industries Alliance (EIA)**
- **Association Connecting Electronics Industries (IPC)**
- **JEDEC Solid State Technology Association (JEDEC)**

2.5 Handling Damage and Contamination

Care must be taken to avoid mechanical damage to the package balls and/or body when components are being placed in the component packing or removed from it, regardless of whether the handling is automatic or manual.

The components in the packing are ready for use. Any contamination of the components or packing may cause or induce (together with other factors) processes which may lead to a damaged device. The most critical types of damage are:

- Solderability problems
- Corrosion
- Electrical shorts (due to conductive particles)

2.6 Component Solderability

The solder balls of the BGA package types listed in **Chapter 1** ensure good solderability, even after a long period of storage. Suitable methods for the assessment of solderabiltiy can be derived from JESD22B 102 or IEC 6068-2-58.

Please also refer to **Chapter 4.5.2** regarding the compatibility of different solder alloys.

3 Printed Circuit Board (PCB)

This chapter is organized as follows:

- **Chapter 3.1**, **Routing**
- **Chapter 3.2**, **PCB Pad Design**
- **Chapter 3.3**, **Pad Surfaces**

3.1 Routing

The main difference between the BGA package types listed in **Chapter 1** and conventional SMT lead frame packages is the array configuration of solder balls on the package. This implies that different concepts are necessary for the routing of the signal, power and ground pins on the PCB.

The design and construction of a printed circuit board is a key factor in achieving high solder joint reliability. For example, it is recommended that BGA packages are **not** placed at the same location on the top and bottom surfaces of the PCB (if double-sided mounting is used) as this can result in a stiffening of the assembly with earlier solder joint fatigue than in a design where the component locations are offset from one another. In addition, the lower bending stiffness of thinner boards (for example, 1.0 mm), compared to thicker boards (for example, 2.35 mm), is better for solder joint reliability (temperature cycling).

Fine line PCBs with a conductor width/spacing of 100 µm are typically necessary for routing. The exact details of the PCB design are highly dependent on the following factors:

- Board technology (conventional technology with drilled vias, build-up technology with microvias)
- Conductor width/spacing
- Number of metal layers
- Any electrical restrictions

3.2 PCB Pad Design

The solder pad design must ensure optimum manufacturability and reliability. Two basic types of solder pads are commonly used:

Solder Mask Defined (SMD) Pad

Figure 4 shows the SMD pad. The copper metal pad is larger than the solder mask opening above it. Thus the pad area is defined by the opening in the solder mask.

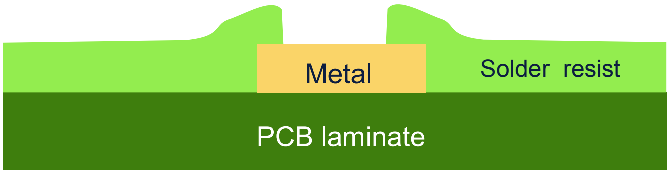


Figure 4 **SMD Pad**

Non Solder Mask Defined (NSMD) Pad

Figure 5 shows the NSMD pad. There is solder mask clearance around each copper metal pad. The dimensions and tolerances of this clearance must ensure that the solder mask does not overlap the solder pad (dependent on the PCB manufacturer's tolerances, 75 µm is a widely used value).

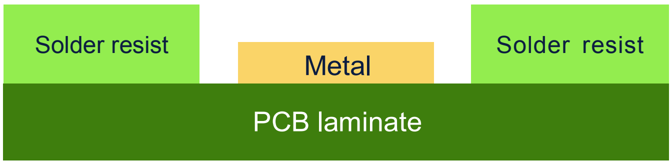


Figure 5 NSMD Pad

We recommend NSMD type pads for the solder pads on the PCB as they provide more space for routing and result in a higher solder joint reliability (also the side walls of the lands are wetted by the solder, which results in less stress concentration).

Figure 6 shows the recommended PCB pad design. Refer to **Table 4** and **Table 5** for the appropriate dimensions, which are ball pitch specific. The values in the tables are typical dimensions. The details depend on the specific PCB technology used, on the capability of the suppliers and on the planned routing. If drilled via holes are placed between the pads, they should be plugged or plated closed to prevent solder flowing into them. If microvias are placed inside the pads, it is recommended that good flatness is specified for them as deep dips within the pads could cause solder joint voiding.

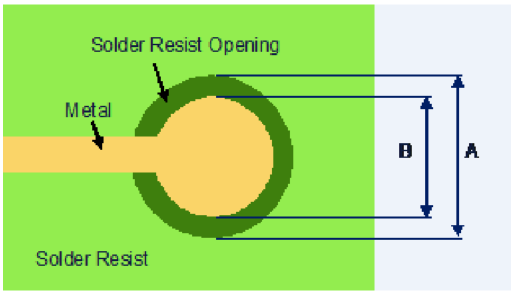


Figure 6 Recommended PCB Pad Design (NSMD Pad)

Table 4 PCB Pad Dimensions (Standard Ball Pitch)

Ball Pitch [mm]/Ball Diameter [mm]	1.00/0.40	1.00/0.50	1.00/0.60	1.27/0.76
Solder resist opening diameter A [µm]	450	500	600	800
Metal pad diameter B [µm]	300	350	400	600

Table 5 PCB Pad Dimensions (Fine Ball Pitch)

Ball Pitch [mm]/Ball Diameter [mm]	0.50/0.30	0.65/0.30	0.80/0.40	0.80/0.50
Solder resist opening diameter A [µm]	430	430	450	500
Metal pad diameter B [µm]	280	280	300	350

3.3 Pad Surfaces

The solder pads must be easy for the solder paste to wet. In general, all finishes are well proven for SMT assembly, but the quality of the plating/finish is especially important for fine-pitch applications. Due to the uneven surface of the Hot Air Solder Leveling (HASL) finish, Pb-free or Pb-containing HASL is less preferable (especially for pitches <0.65 mm) than completely "flat" platings such as Cu-OSP (OSP: Organic Solderability Preservative) or electroless Sn or NiAu.

From a package point of view, it is difficult to recommend one particular PCB pad finish that will always meet all requirements. The choice of finish also depends strongly on the board design, pad geometry, the specific components on the board, and the process conditions. The choice of finish must be made according to the specific needs of the customer.

Internal tests performed by Lantiq have shown that Cu-OSP is a suitably reliable plating.

Table 6 Typical PCB Pad Finishes

Finish	Typical Layer Thickness [µm]	Properties	Concerns
HASL (SnAg) (Hot Air Leveling)	> 5	Low cost, widely used, know-how in fabrication	Uneven surface, formation of humps, flatness of single pads must be good for fine pitch applications
Electroless Tin	0.3 – 1.2	Solder joint only consists of copper and solder, no further metal is added to the solder joint	Long-term stability of protection may be a concern, baking of PCB may be critical
Electroless Silver	0.2 - 0.5	Solder joint only consists of copper and solder, no further metal is added to the solder joint	Long-term stability of protection may be a concern, baking of PCB may be critical
Electroless Ni / Immersion Au (ENIG)	3 – 7 / 0.05 – 0.15	Good solderability protection, high shear force values	Expensive, concerns about brittle solder joints
Galvanic Ni / Au	> 3 / 0.1 – 2	Only for thicker layers, typically used for connectors	Expensive, not recommended for solder pads
OSP (Organic Solderability Preservatives)	1	Low cost, simple, fast automated fabrication	Must be handled carefully to avoid damaging the OSP; not as good long-term stability as other coatings; in the case of double-sided assembly, only suitable with inert gas during reflow

4 Board Assembly

This chapter is organized as follows:

- **Chapter 4.1**, **General Remarks**
- **Chapter 4.2**, **Solder Stencil**
- **Chapter 4.3**, **Solder Paste**
- **Chapter 4.4**, **Component Placement**
- **Chapter 4.5**, **Soldering**
- **Chapter 4.6**, **Cleaning and Subsequent Processing**

4.1 General Remarks

Many factors within the board assembly process influence the assembly yield and board level reliability, for example, the design and material of the stencil, the solder paste material, the solder paste printing process, component placement and the reflow process.

It should be emphasized that this document only provides a guideline aimed at supporting customers in selecting the appropriate processes and materials. Further optimization that takes into account the actual PCB, the customer's SMT equipment and product specific requirements must be carried out by the customer.

4.2 Solder Stencil

The solder paste is applied to the metal pads on the PCB by screen printing. The volume of the printed solder paste is determined by the stencil aperture and the stencil thickness. In most cases, the thickness of the stencil has to be matched to the needs of all the components on the PCB.

A stencil thickness of 100-150 µm is recommended for the BGA packages listed in **Chapter 1**. The stencil apertures should be circular, and the aperture diameter should be the same as the metal pad diameter on the PCB or slightly larger.

In order to ensure uniform, high solder paste transfer to the PCB, laser cut (mostly made from stainless steel) or electroformed stencils (nickel) are preferable.

Table 7 Recommendations for Stencil Dimensions

Package	Stencil Aperture Diameter [µm]	Stencil Thickness [µm]
P(G)-BGA, Ball pitch 1.27 mm, ball diameter 0.76 mm	600-650	100-150
P(G)-BGA, P(G)-LBGA, Ball pitch 1.00 mm, ball diameter 0.60 mm	400-450	100-150
P(G)-BGA, P(G)-LBGA, P(G)-FCLBGA, Ball pitch 1.00 mm, ball diameter 0.50 mm	350-400	100-150
P(G)-TBGA, P(G)-FCTBGA Ball pitch 1.00 mm, ball diameter 0.40 mm	300-350	100-125
P(G)-LFBGA, P(G)-LF^{2}BGA, Ball pitch 0.80 mm, ball diameter 0.50 mm	350	100-150
P(G)-LFBGA, P(G)-LF^{2}BGA, P(G)-TFBGA, Ball pitch 0.80 mm, ball diameter 0.40 mm	300-350	100-125
P(G)-LFBGA, P(G)-LF^{2}BGA, P(G)-TFBGA, Ball pitch 0.65 mm, ball diameter 0.30 mm	300	100-120
P(G)-LFBGA, P(G)-LF^{2}BGA, P(G)-TFBGA, Ball pitch 0.50 mm, ball diameter 0.30 mm	300	100-120

4.3 Solder Paste

Solder paste consists of a solder alloy combined with a flux system, with a typical volume split of 50% alloy and 50% flux. In terms of mass, this means approx. 90 wt% alloy and 10 wt% flux system. One of the functions of the flux system is to remove oxidation (contamination) from the surfaces to be soldered. The ability to remove contaminants is given by the activation level. A lead based solder paste metal alloy has to be of leaded eutectic or near-eutectic composition (SnPb or SnPbAg). A lead-free solder paste metal alloy composition (typically SnAgCu with Ag 3 - 4%, Cu 0.5 - 1%) can also be applied. A "no-clean" solder paste is preferable, because cleaning under the soldered BGA may be difficult. The paste must be suitable for printing the solder stencil aperture dimensions. Type 3 paste is recommended. Solder paste is sensitive to storage time, temperature and humidity. Please adhere to the handling recommendations of the paste manufacturer.

4.4 Component Placement

Manual component positioning is not recommended as BGA packages have to be placed very accurately.

Component placement accuracies of ±50 µm are obtainable with modern automatic component placement machines using vision systems. These systems measure both the PCB and the components optically. The components are then placed on the PCB at their programmed positions. The fiducials on the PCB are either located at the edge of the PCB (for the entire PCB) or at individual mounting positions (local fiducials). They are detected by the vision system immediately before the mounting process. Package recognition is performed by a special vision system, which enables correct centering of the complete package.

One of the advantages of packages with solder balls, as is the case with BGA packages, is that if they are slightly displaced, they self-align during the reflow process due to the high surface tension of the solder. The maximum tolerable displacement of the components is 30% of the metal pad width on the PCB (for non solder mask defined pads). Consequently for BGA packages, the solder ball to PCB pad misalignment must be better than 150 µm to ensure a sufficiently accurate mounting process. A wide range of placement systems can achieve this.

The following points are important:

- Especially on large boards, local fiducials close to the device can compensate for PCB inaccuracies.
- It is recommended that the ball recognition capabilities of the placement system are used, not the outline centering. This eliminates the solder ball to package edge tolerances of the package (please refer to the corresponding package outline drawing for details).
- To ensure that the packages are identified correctly by the vision system, adequate lighting and the correct choice of measuring modes are necessary. The accurate settings can be taken from the equipment manuals.
- If too much force is applied when a component is placed, solder paste may be squeezed out and cause solder joint shorts. On the other hand if too little force is applied, there may be insufficient contact between the package and solder paste, which can lead to open solder joints or badly centered packages.

4.5 Soldering

To a large extent, the yield and quality of the assembly process are determined by the soldering. All the standard reflow soldering processes:

- Forced convection
- Vapor phase
- Infrared (with restrictions)

and typical temperature profiles are suitable for BGA board assembly. However, wave soldering cannot be used. During the reflow process, the time each solder joint is exposed to temperatures above the solder liquidus temperature must be long enough to achieve the optimum solder joint quality, but overheating of the PCB and its components must be avoided. Please refer to the bar code label on the device packing for the peak package body temperature. It is important that the maximum temperature of the BGA package during the reflow process does not exceed the specified peak temperature.

When using infrared ovens without convection, special care may be necessary to ensure a sufficiently homogeneous temperature profile for all the solder joints on the PCB, especially on large complex boards where the components, including those under the BGA packages, have different thermal masses. The most recommended type of reflow process is forced convection reflow. A nitrogen atmosphere can improve the solder joint quality, but is normally not necessary when soldering tin-lead metal alloys.

The temperature profile of a reflow process is one of the most important factors in the soldering process. It is divided into several phases, each with a special function. **Figure 7** shows the temperature profile for a forced convection reflow process used to solder BGA packages. **Table 8** gives examples of the key parameters for such a solder profile, for both tin-lead and lead-free alloys. The individual parameters are influenced by various factors, not only by the package. It is essential to follow the solder paste manufacturer's application notes.

In addition, most PCBs contain devices of more than one package type and, therefore, the reflow profile has to be matched to the requirements of all the components and materials. We recommend that the temperatures of the solder joints are measured using thermocouples beneath the respective packages. The fact that components with large thermal masses do not heat up at the same speed as lightweight components must be taken into account. The position and surroundings of the package on the PCB, as well as the PCB thickness can also influence the solder joint temperature significantly. Therefore no concrete temperature profile can be given.

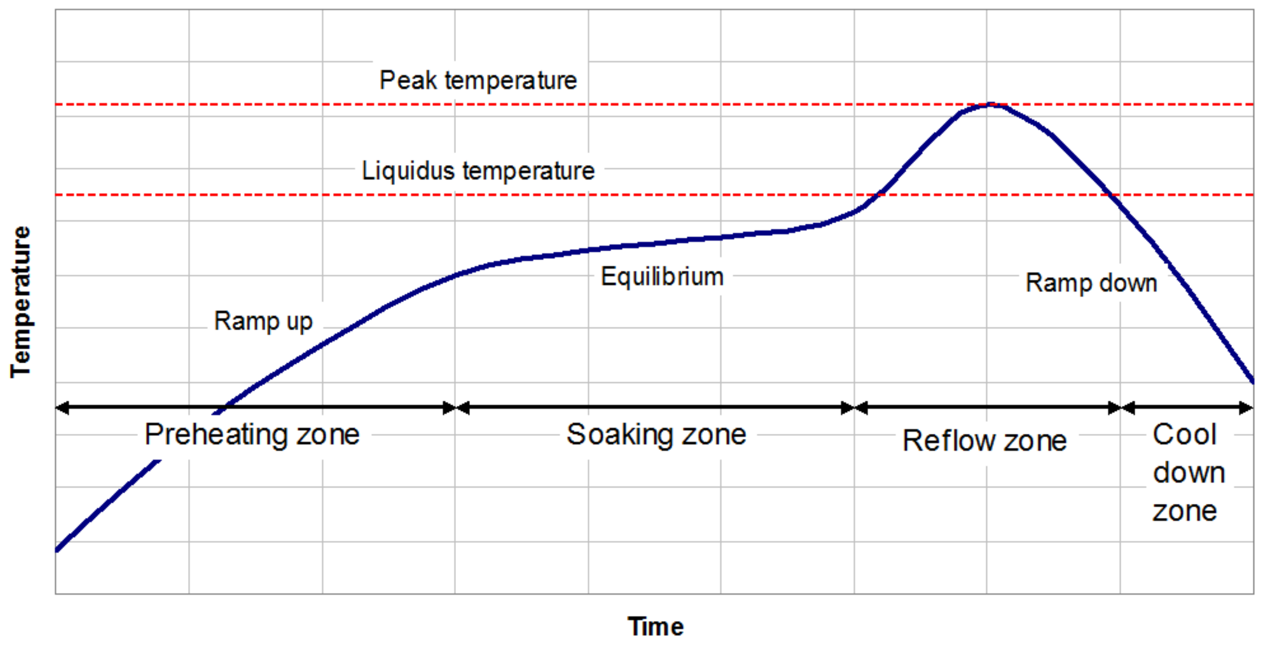


Figure 7 Temperature Profile of Forced Convection Reflow Solder Profile

The details given in **Table 8** are examples, not recommendations (for reference only):

Table 8 Examples of Key Parameters for a Forced Convection Reflow Solder Process

Parameter	Tin-lead Alloy (SnPb or SnPbAg)	Lead-free Alloy (SnAgCu)	Main Requirements From ...
Preheating rate	2.5 K/s	2.5 K/s	Flux system (solder paste)
Soaking temperature	140 - 170°C	140 - 170°C	Flux system (solder paste)
Soaking time	80 s	80 s	Flux system (solder paste)
Peak temperature	225°C	245°C	Alloy (solder paste)

Table 8 Examples of Key Parameters for a Forced Convection Reflow Solder Process (cont'd)

Parameter	Tin-lead Alloy (SnPb or SnPbAg)	Lead-free Alloy (SnAgCu)	Main Requirements From ...
Reflow time over liquidus	60 s	60 s	Alloy (solder paste)
Cool down rate	2.5 K/s	2.5 K/s	–

4.5.1 Double-Sided Assembly

The BGA packages listed in **Chapter 1** are suitable for mounting on double-sided PCBs. In such cases, the PCB is assembled on one side first (including soldering) and then the other side is assembled.

It should be noted that heavy packages that are facing downwards could fall off during the second reflow process,. In such cases these packages have to be assembled during the last (second) reflow process. As a rule of thumb, a weight limit of 0.2 g/mm² soldered area (NSMD pad) can be assumed. Which packages are affected depends not only on the mass, but also on the vibrations and the air draft in the reflow oven.

4.5.2 Compatibility of Solder Alloys

As different solder alloys may be used for the package balls and solder paste printed on the PCB, the compatibility of the alloys has to be taken into account. **Table 9** describes the possible combinations and our recommendations as to which of them provide good processability and reliability.

Table 9 Possible Combinations of Lead-Based and Lead-Free Solder Balls and Solder Paste

	SnPb(Ag) Solder Paste (= low peak temperature, 220 - 230°C)	SnAgCu Solder Paste (= high peak temperature, 235 - 260°C)
SnPb(Ag) Solder Ball	OK	OK
SnAgCu Solder Ball	Reliability concerns / with restrictions (see the following notes)	OK

In terms of combining materials, it is possible to mix SnPb(Ag) solder paste with SnAgCu solder balls. However, the required minimum peak temperature at reflow for the combination of lead-free solder balls and SnPb(Ag) solder paste is of particular interest. **Table 10** is the result of an in-house evaluation on this topic.

Table 10 Correlation Between Processability and Peak Temperature/Coplanarity

Peak Temperature (at Solder Joint)	Regular Coplanarity
200 – 215°C	No
220 – 230°C	Yes with restrictions
235 – 260°C	Yes

No suitable solder joints are formed below a peak temperature of 215°C. Suitable solder joints are only possible with a peak temperature around 225°C if the packages have good coplanarity values. As the ability of BGA packages to self-align is reduced at this temperature, the package must be placed exactly on the solder paste. Suitable solder joints are formed with a peak temperature over 230°C.

4.6 Cleaning and Subsequent Processing

4.6.1 Cleaning

Flux residue may be found around the solder joints after the reflow soldering process. If a "no-clean" solder paste has been used for the solder paste printing, the flux residue does not usually have to be removed after the soldering process.

Please note that it is difficult to clean beneath a BGA package because of the small gap between the package and the PCB, and therefore it is not recommended. However if the solder joints have to be cleaned, the following points need to be taken into account when selecting the cleaning method and solution (for example, ultrasonic, spray or vapor cleaning):

- The type of the packages to be cleaned.
- The flux used in the solder paste (resin-based, water-soluble, etc.).
- Environmental and safety aspects.

Any cleaning solution residue should be removed/dried very thoroughly. Please contact the solder paste manufacturer for recommendations on which cleaning solutions to use.

4.6.2 Underfilling

Applying an underfill material beneath the package is an efficient method of improving the board level reliability with regard to mechanical and thermomechanical stresses, especially in the case of smaller ball pitches.

A liquid epoxy resin is dispensed onto the PCB next to the package outline at a distance that allows the material to wet the gap between the package and board. The underfiller then flows into the gap by means of capillary action and fills it completely.

The process parameters are dependent on the underfill material used, the package size, the ball matrix and the design of the PCB. To achieve shorter flow times the substrate is usually heated up to 50°C - 90°C during dispensing to reduce the fluid viscosity. In the case of small package sizes (approx. < 40 mm^2), it is sufficient to only deposit the underfiller in a line along one side of the package (length approx. 75% of package side). In the case of larger packages, it is recommended that the underfiller is dispensed in an L-shaped pattern along two adjacent sides of the package. In the case of very large packages, it may be necessary to dispense an additional L-shape, after the first dispensed material has flowed under the package, in order to achieve complete underfilling. U-shape patterns are an alternative for large packages, but the risk here is that air might be trapped between colliding flow fronts.

Typically liquid, filled or unfilled epoxy resins are used for package underfilling. Ultrafine fillers are not needed and it is possible to use low cost materials with larger filler particles (max. filler size should be ≤ 1/3 of the gap).

5 Inspection

Visual inspection of the solder joints using conventional AOI (automatic optical inspection) systems is not possible. The only reasonable method of realizing efficient inline control is the implementation of AXI (automatic X-ray inspection) systems. These are available as 2D and 3D solutions and usually consist of an X-ray camera and the hardware and software required for inspection, controlling, analysis and data transfer. AXI systems offer the user a fairly reliable way of detecting soldering defects such as poor soldering, bridging, voiding and missing parts. However, other defects such as broken solder joints cannot easily be detected via X-rays. Please refer to the IPC-A-610C standard for further details on the acceptability of such electronic assemblies.

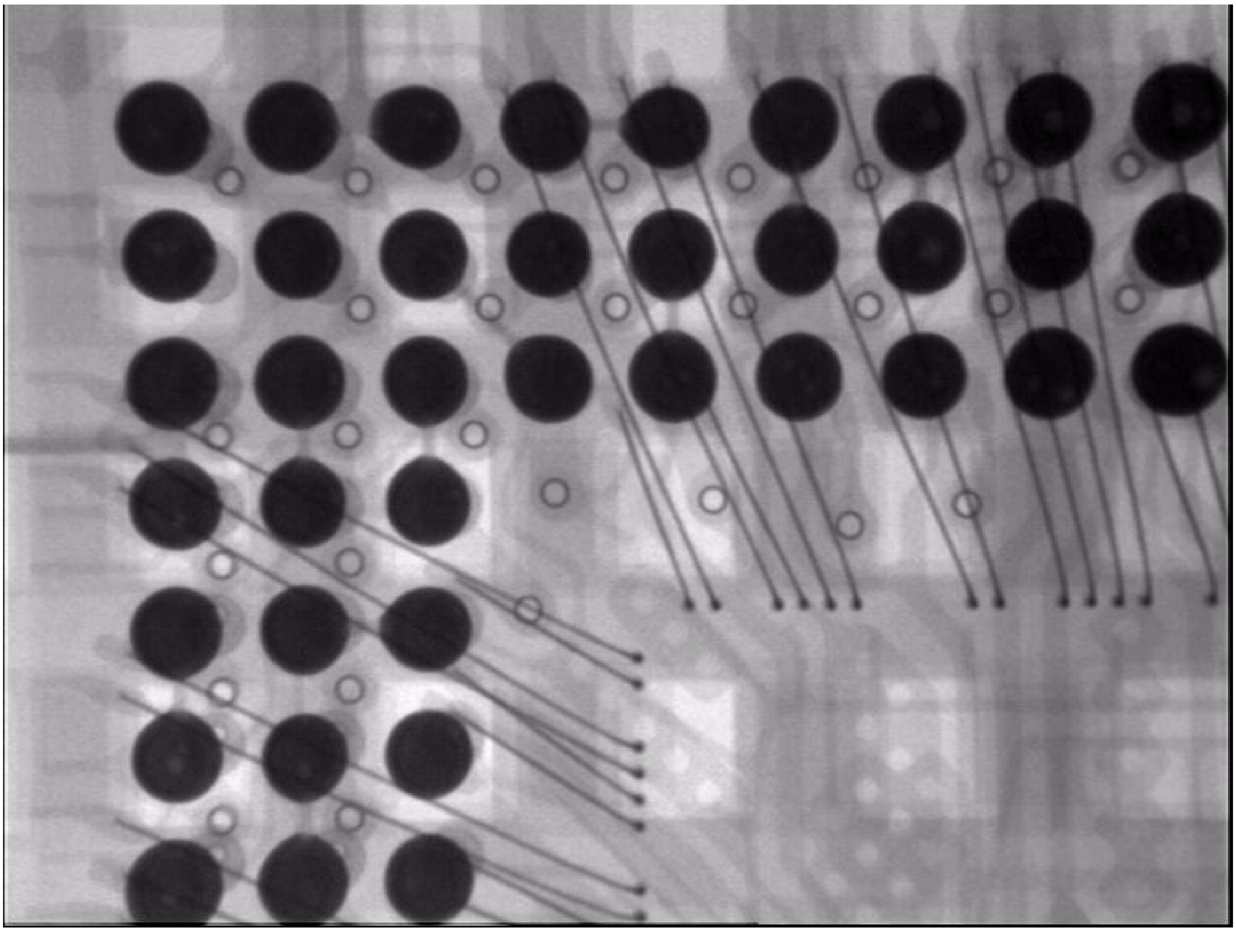


Figure 8 Typical X-ray Image of Soldered Wire-Bond BGA Package

A further type of inspection equipment that can be used is endoscopes. These can be used to detect numerous types of failures beneath the BGAs. The optical head of the system moves around the package near the PCB area, allowing the user to look along the solder rows by adjusting the focus. The pictures from such an endoscope system are far easier to interpret than X-ray images.

The cross-sectioning of a soldered package or dye penetrant analysis can only be used for sample monitoring, as both of these methods destroy the device.

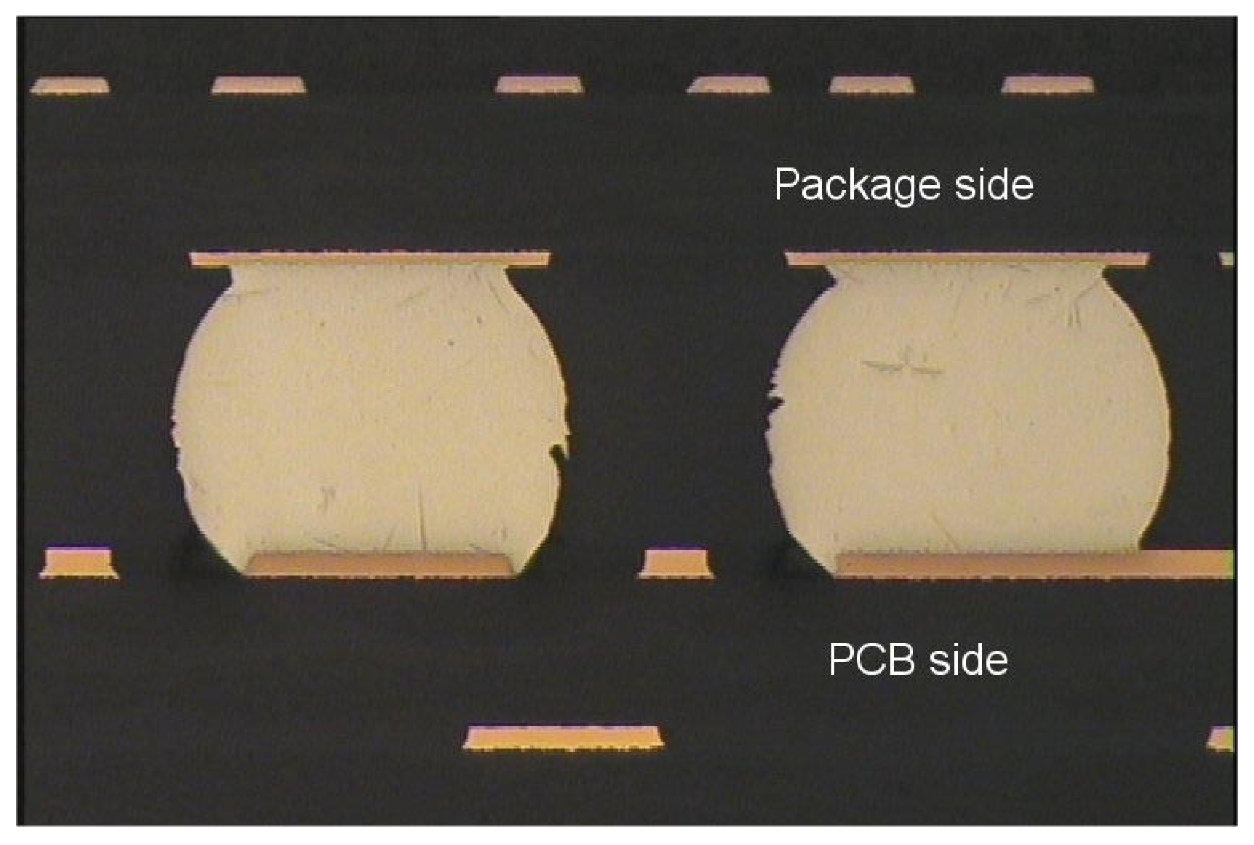


Figure 9 Sample Cross Section of BGA Solder Balls

Please note that lead-free solder joints look different to tin-lead solder joints. The surface properties are caused by the irregular solidification of the solder. As the solder alloys used are not exactly eutectic (for example, 63Sn37Pb solder alloy), they do not have a melting point but a melting range over several degrees temperature difference.

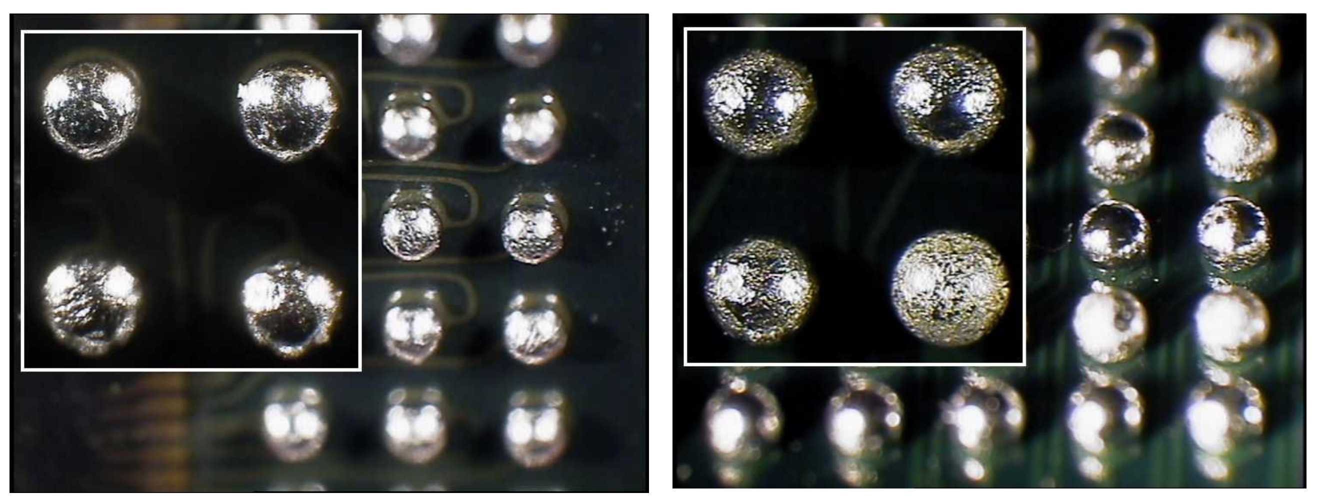


Figure 10 Shiny SnPb Solder Balls (Left) versus Dull SnAgCu Solder Balls (Right)

6 Rework

This chapter is organized as follows:

- **Chapter 6.1**, **Tooling**
- **Chapter 6.2**, **Device Removal**
- **Chapter 6.3**, **Site Redressing**
- **Chapter 6.4**, **Reassembly and Reflow**

If a defect component is found after board assembly, the device can be removed and replaced with a new one. It is not possible to repair individual solder joints on a component.

6.1 Tooling

The rework process is commonly performed using special rework equipment. Numerous systems are available on the market, however, the equipment should fulfill the following requirements if used to process these packages:

Heating

Hot air heat transfer to the package and PCB is highly recommended. The temperature and air flow used to heat the device is controlled. Programmable temperature profiles can be used to adapt to different package sizes and masses (for example, using a PC controller).

The preheating of the PCB from the underside is recommended. In particular, infrared heating can be used for this, but it should only support the hot air flow from the top side. Nitrogen can also be used instead of air.

Vision System

The bottom surface of the package, as well as the site on the PCB are observable. A split vision optic is implemented to superimpose the component and board images. This enables precise alignment of the package with the PCB. The microscope magnification and resolution are appropriate for the pitch of the device.

Moving and Additional Tools

The tool is able to move across the whole area of the PCB. The recommended placement accuracy is better than ±100 µm. The system is able to remove solder residue from the PCB pads (special vacuum tools).

6.2 Device Removal

If a defect component is to be sent back to the supplier, it must be ensured that no further defects are introduced when removing the component as this would interfere with failure analysis to be carried out by the supplier. This includes the following recommendations:

Moisture

Depending on the moisture sensitivity level, the package may have to be dried before removal. If the maximum storage time out of the moisture-resistant bag (see label on packing material) is exceeded after board assembly, the PCB must be dried according to the recommendations (see **Chapter 2.3**), otherwise too much moisture may have accumulated and damage may have occurred (popcorn effect).

Temperature Profile

During the soldering process, it should be ensured that the package peak temperature is not higher nor the temperature ramps steeper than for the standard assembly reflow process (see **Chapter 4.5**).

Mechanics

Do not apply excessive mechanical force to remove the device, otherwise failure analysis of the package may be impossible or the PCB might be damaged. Pipettes can be used for larger packages (implemented in most rework systems), whereas tweezers may be more practical for small packages.

When defect components are underfilled, it is necessary to clarify whether they can be tested by Lantiq directly after desoldering or whether they have to be re-balled first. The underfill may prevent adequate contact with the desoldered component being established or may even hinder re-balling. In this case, or when removal of the suspect component its too difficult or risky, it may be necessary to send back the whole (or part of the) PCB containing the defect component to Lantiq.

6.3 Site Redressing

After the defect component has been removed, the pads on the PCB have to be cleaned of solder residue. Do not use steel brushes as steel residue can lead to poor solder joints. Before placing a new component on the PCB, it is recommended that flux is applied by dispensing it onto each PCB pad or by covering the whole site with it (for example, using a brush). It is recommended that only no-clean flux is used, if possible the same one that is used in the solder paste for stencil printing.

6.4 Reassembly and Reflow

After the site has been prepared, the new package can be placed on the PCB. It is positioned exactly above the PCB pads, just high enough to ensure that there is no contact between the package and the PCB. The package is then dropped into the printed or dispensed flux or solder paste deposit (zero-force-placement). During the soldering process, it should be ensured that the package peak temperature is not higher nor the temperature ramps steeper than for the standard assembly reflow process (see **Chapter 4.5**).

Packages with lead-free balls can be reworked in the same way as leaded packages, but peak temperatures have to be adjusted according to the solder used (see **Chapter 4.5**). In addition, overheating must be avoided.

Standards References

[1] IEC 60286-3 Edition 5.0 2013-05
Packaging of components for automatic handling –
Part 3: Packaging of surface mount components on continuous tapes

[2] IEC 60286-5 Edition 2.1 2009-05
Packaging of components for automatic handling –
Part 5: Matrix trays

[3] IEC 60286-6 Edition 2.0 2004-02
Packaging of components for automatic handling –
Part 6: Bulk case packaging for surface mounting components

[4] IPC/JEDEC J-STD-033C February 2012
Handling, packing, shipping and use of moisture/reflow sensitive surface mount devices

[5] ANSI/EIA-481-D 2008-07
8 mm through 200 mm embossed carrier taping and 8 mm & 12 mm punched carrier taping of surface mount components for automatic handling

[6] EIA-783 Edition 98
Guideline orientation standard for multi-connection package (design rules for tape and reel orientation)

[7] IEC 60721-3-0 Edition 1.1 2002-10
Classification of environmental conditions –
Part 3: Classification of groups of environmental parameters and their severities –
Introduction

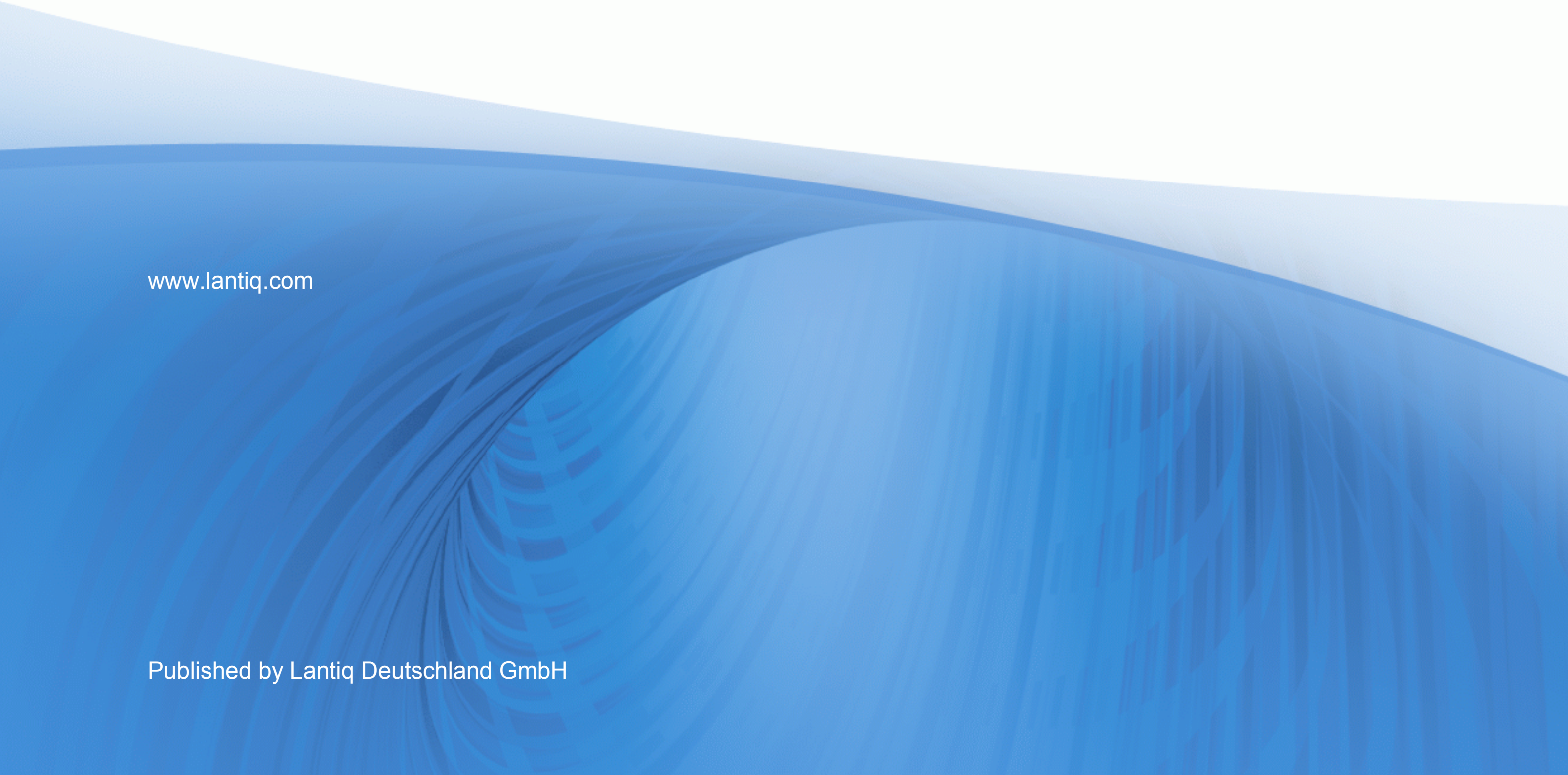
[8] IEC 60721-3-1 Edition 2.0 1997-02
Classification of environmental conditions –
Part 3 Classification of groups of environmental parameters and their severities –
Section 1: Storage

[9] IEC 60721-3-2 Edition 2.0 1997-03
Classification of environmental conditions –
Part 3: Classification of groups of environmental parameters and their severities –
Section 2: Transportation

[10] IEC 61760-2 Edition 2.0 2007-04
Surface mounting technology –
Part 2: Transportation and storage conditions of surface mounting devices (SMD) –
Application guide

[11] IEC/TR 62258-3 Edition 2.0 2010-08
Semiconductor die products –
Part 3: Recommendations for good practice in handling, packing and storage

[12] ISO 14644-1:1999
Cleanrooms and associated controlled environments –
Part 1: Classification of air cleanliness



www.lantiq.com

Published by Lantiq Deutschland GmbH