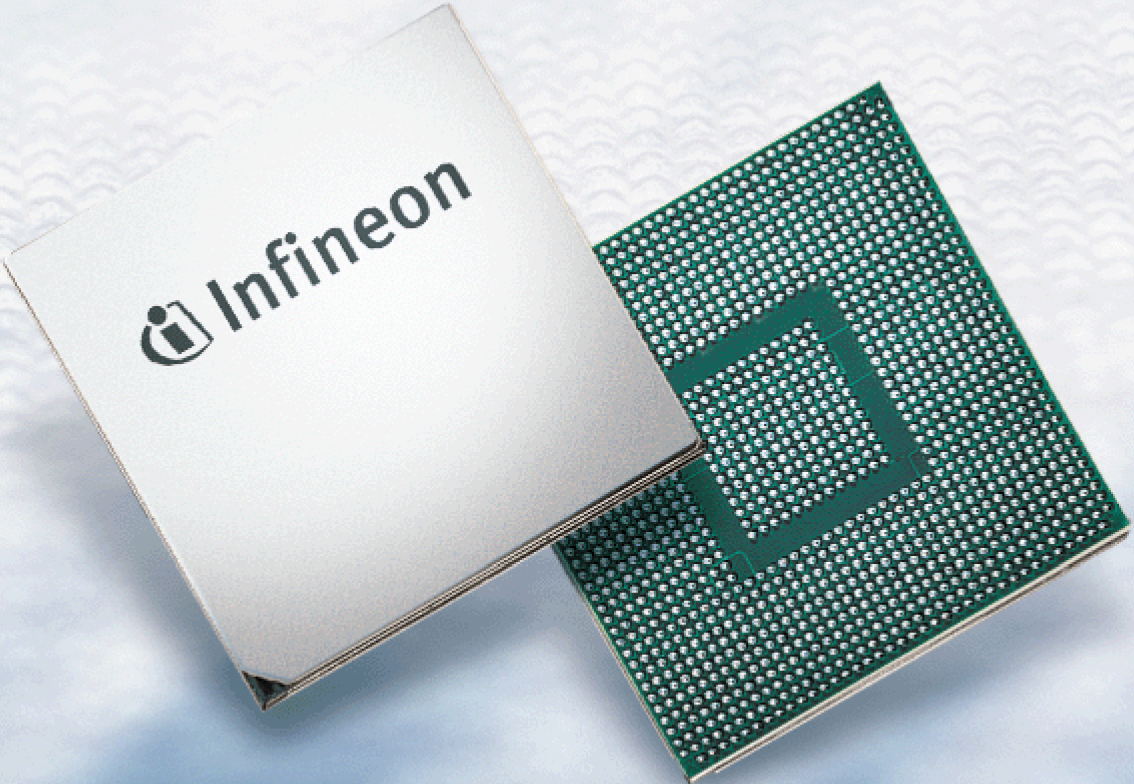
Additional Information, DS1, June 2004

Recommendations for Printed Circuit Board Assembly of Infineon P-HBGA/P-FCHBGA Packages



Never stop thinking

Edition 2004-06-21

Published by
Infineon Technologies AG
81726 München, Germany

Legal Disclaimer

The information given in this document shall in no event be regarded as a guarantee of conditions or characteristics ("Beschaffenheitsgarantie"). With respect to any examples or hints given herein, any typical values stated herein and/or any information regarding the application of the device, Infineon Technologies 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.

Information

For further information on technology, delivery terms and conditions and prices please contact your nearest Infineon Technologies Office (**www.infineon.com**).

Warnings

Due to technical requirements components may contain dangerous substances. For information on the types in question please contact your nearest Infineon Technologies Office.

Infineon Technologies Components may only be used in life-support devices or systems with the express written approval of Infineon Technologies, if a failure of such components can reasonably be expected to cause the failure of that life-support device or system, or to affect the safety or effectiveness of that device or system. Life support devices or systems are 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.

Table of Contents

1 Package Description

Infineon's P(G)-HBGA (Plastic (Green) High Performance Ball Grid Array Package, **Figure 1**) is a organic substrate based wide area cavity down wire bond package with solder ball grid array. It is a suitable package for high end applications like Logic IC's for networking devices.

Features

- Wide offer of package body sizes (medium to large)
- Applicable for large die sizes
- Wide offer of BGA ball counts (low to high)
- Wire bond interconnect technology
- Enhanced electrical performance (> 1 GHz)
- Superior thermal performance
- Same processing as for standard P(G)-BGA packages

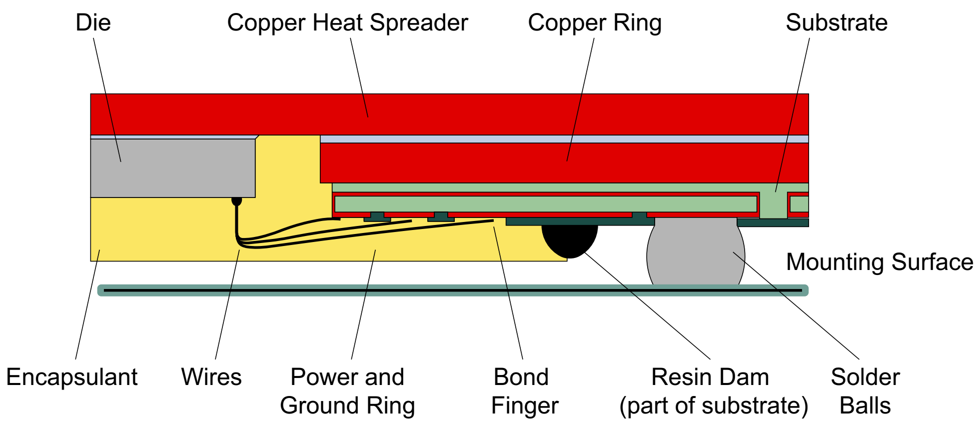


Figure 1 P(G)-HBGA Cross Section

Infineon's P(G)-FCHBGA (Plastic (Green) Flip Chip High Performance Ball Grid Array Package, **Figure 2**) is a build up substrate based wide area package with flip chip interconnects and a solder ball grid array. It is a preferred package for high end applications like Logic IC's for networking devices.

Features

- Wide offer of (large) package body sizes
- Applicable for large die sizes
- Wide offer of BGA ball counts (medium to high)
- Flip chip interconnect technology
- Superior electrical performance (> 1 GHz)
- Superior thermal performance
- Integrated passive components possible
- Option for multi chip package
- Same processing as for standard P(G)-BGA packages

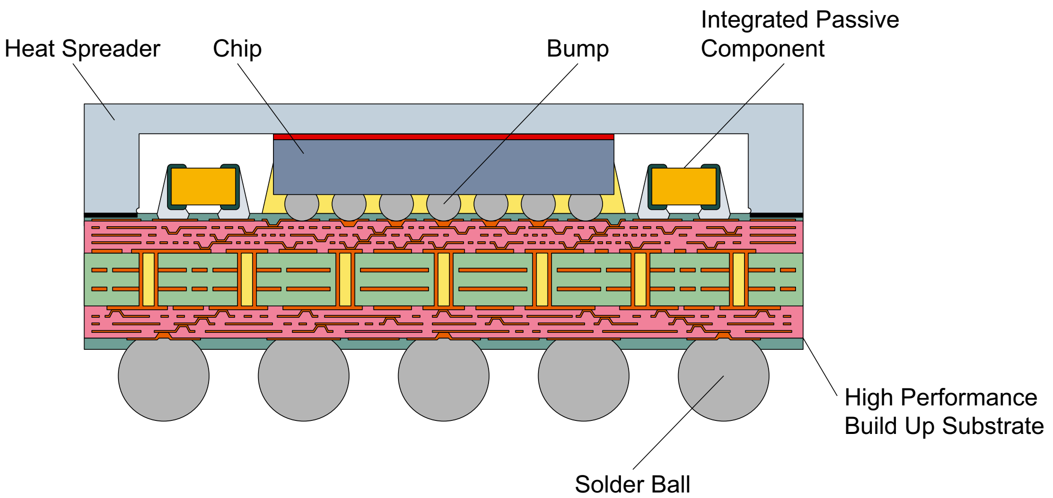


Figure 2 P(G)-FCHBGA Cross Section

2 Package Handling

2.1 ESD Protective Measures

Semiconductor devices are normally electrostatic discharge sensitive devices (ESDS) requiring specific precautionary measures in respect of handling and processing. Only in this way is it possible to insure that the components can be inserted into assemblies without becoming damaged. Discharging of electrostatic charged objects over an IC, caused by human touch or by processing tools may cause high current respectively high voltage pulses, which may damage or even destroy sensitive semiconductor structures. On the other hand ICs may also be charged during processing. If discharging takes place too quickly ("hard" discharge), it may cause load pulses and damages, either. ESD protective measures must therefore prevent a contact with charged parts as well as a charging of the ICs. Protective measures against ESD include both the handling and processing and the packing of ESDS. A few hints are provided below on handling and processing.

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
- Earth 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 Personal

- 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 having 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 which should be considered:

IFX packs according to the IEC 60286-* series (The IEC 60286-3 is similar to the EIA 481-*)

IEC 60286-1	Packaging of components for automatic handling - Part 1: Tape packaging of components with axial leads on continuous tapes
IEC 60286-2	Packaging of components for automatic handling - Part 2: Tape packaging of components with unidirectional leads on continuous tapes
IEC 60286-3	Packaging of components for automatic handling - Part 3: Packaging of surface mount components on continuous tapes
IEC 60286-4	Packaging of components for automatic handling - Part 4: Stick magazines for dual-in-line packages
IEC 60286-5	Packaging of components for automatic handling - Part 5: Matrix trays
IEC 60286-6	Packaging of components for automatic handling - Part 6: Bulk case packaging for surface mounting components

Moisture Sensitive Surface Mount Devices are packed according to IPC/JEDEC J-STD-033A July 2002: Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices

Detailed packing drawings: Packing Information (Internet)

Other references:

ANSI/EIA-481-*	Standards Proposal No. 5048, Proposed Revision of ANSI/EIA-481-B 8 mm through 200 mm Embossed Carrier Taping and 8 mm & 12 mm Punched Carrier Taping of Surface Mount Components for Automatic Handling (if approved, to be published as ANSI/EIA-481-C)
EIA-726	8 mm Punched & Embossed Carrier Taping of Surface Mount Components for Automatic Handling of Devices Generally Smaller than 2.0 mm x 1.2 mm
EIA-747	Adhesive Backed Punched Plastic Carrier Taping of Singulated Bare Die and Other Surface Mount Components for Automatic Handling of Devices Generally Less than 1.0 mm Thicl
EIA/IS-763	Bare Die and Chip Scale Packages Taped in 8 mm & 12 mm Carrier Tape for Automatic Handling
EIA-783	Guideline Orientation Standard for Multi-Connection Package (Design Rules for Tape and Reel Orientation)

2.3 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 popcorn effects.

List of relevant standards which should be considered:

IEC 60721-3-0	Classification of environmental conditions: Part 3: Classification of groups of environmental parameters and their severities; introduction
IEC 60721-3-1	Classification of environmental conditions: Part 3: Classification of groups of environmental parameters and their severities; Section 1: Storage
IEC 60721-3-2	Classification of environmental conditions: Part 3: Classification of groups of environmental parameters and their severities; Section 2: Transportation
IEC 61760-2	Surface mounting technology - Part 2: Transportation and storage conditions of surface mounting devices (SMD) - Application guide
IEC 62258-3	Semiconductor Die Products - Part 3: Recommendations for good practise in handling, packing and storage
ISO 14644-1	Clean rooms and associated controlled environments Part 1: Classification of airborne particulates

Table 1 General Storing Conditions - Overview

Product	Condition for Storing
Wafer/Die	N2 or MBB (IEC 62258-3)
Component - moisture sensitive	MBB[1)] (JEDEC J-STD-033*)
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 Institute (ANSI)

Electronics Industries Alliance (EIA)

Association Connecting Electronics Industries (IPC)

3 Printed Circuit Board, PCB

3.1 Routing

The main difference between the P(G)-HBGA / P(G)-FCHBGA and conventional SMT leadframe packages is the array configuration of solder spheres on the package, which implies different concepts for routing the signal, power and ground pins on the PCB.

Generally the printed circuit board design and construction is a key factor for achieving a high reliability of the solder joints. For example, it is recommended not to place P(G)-HBGA / P(G)-FCHBGA packages at the same opposite locations on the PCB (if double sided mounting is used), because this results in a stiffening of the assembly with earlier solder joint fatigue compared to a design where the component locations are shifted against each other. Furthermore it is known, that the lower bending stiffness of thinner boards (e.g. 1.0 mm) are favourable for solder joint reliability (temperature cycling) compared to thick boards (e.g. 2.35 mm).

Typically fineline PCBs with conductor width/spacing of 100 µm are necessary for routing. The details of the used PCB design strongly depends on the board technology (conventional technology with drilled vias, build-up technology with microvias), the conductor width/spacing, number of metal layers, and electrical restrictions.

3.2 PCB Pad Design

The solder pads have to be designed to assure optimum manufacturability and reliability. Generally two basic types of solder pads are commonly used:

- "Solder mask defined" (SMD) pad (**Figure 3**): The copper pad is larger than the solder mask opening above this pad. Thus the land area is defined by the opening in the solder mask.

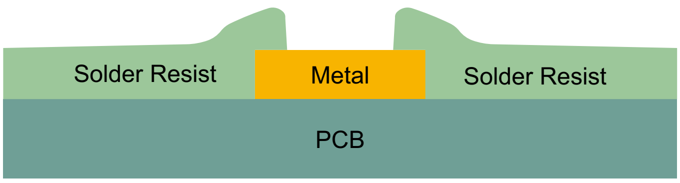


Figure 3 SMD Pad

- "Non solder mask defined" (NSMD) pad (**Figure 4**): Around each copper pad there is solder mask clearance. It is necessary to specify the dimensions and tolerances of the solder mask clearance in this way, that no overlapping of the solder pad by solder mask occurs (depending on PCB manufacturers tolerances, 75 µm is a widely used value).

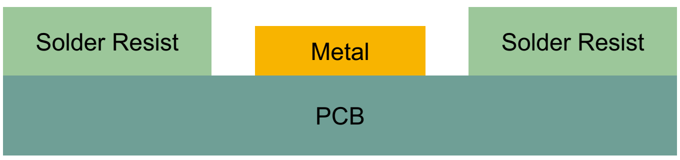


Figure 4 NSMD Pad

Because NSMD pads 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), NSMD type is recommended for the solder pads on the PCB.

Figure 5 shows the recommended PCB Pad design. Refer to **Table 2** for the appropriate dimensions, which are ball pitch specific. The values in the table are typical dimensions. The details depend on the used PCB technology, capability of the suppliers and the planned routing. If drilled via holes are placed between the pads, they should be plugged or plated closed to prevent solder flow into this via. If microvias are placed inside the pads, it is recommended to specify a good flatness of the vias. Deep dips inside the pads may cause solder joint voiding.

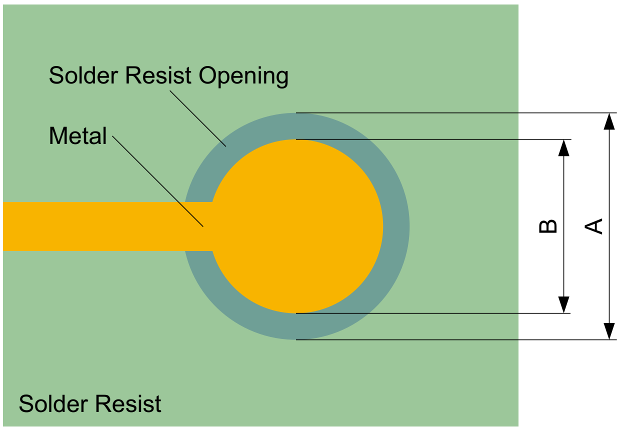


Figure 5 PCB Pad Design Recommendation (NSMD Pad)

Table 2 PCB Pad Dimensions

Ball Pitch [mm]	**1.00**	**1.27**
Solder resist opening diameter A [µm]	600	800
Metal pad diameter B [µm]	400	600

3.3 Pad Surfaces

The solder pads must have good wettability to the soldering material (solder paste). All finishes listed here are well proven for SMT assembly, but especially for fine pitch applications the quality of the plating gets more important. Therefore Hot Air Solder Leveling finish can be used, but the flatness of the single pads has to specified for high quality, not to bring on soldering problems. The properties of some preservatives are listed in **Table 3**.

Table 3 Properties of some Solderability Preservative Layers on PCBs

Finish	Typ. Layer Thickness [µm]	Properties	Concerns
HASL (SnAg) *(Hot Air Solder Leveling)*	> 5	cheap, widely usage, know how in fabrication	uneven surface, formation of humps, flatness of single pads has to be good for fine pitch applications
Electroless Tin	0.3 - 1.2	solder joint consists only 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 consists only 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)*	typical 1	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. In literature the test results of solderability, wetting force and wetting time for several preservative layers are not always coincident.

Special notes for green ("G") packages:

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. Generally, the wettability of tin-lead solder paste on the described surface platings is better compared to lead-free solder paste.

4 PCB Assembly

4.1 Solder Stencil

The solder paste is applied onto the PCB metal pads 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 a stencil has to be matched to the needs of all components on the PCB. For P(G)-HBGA and P(G)-FCHBGA packages it is recommended to use 100 - 150 µm thick stencils. Stencil apertures should be of circular shape. The aperture diameter should be the same as the metal pad diameter on the PCB or slightly above.

To ensure a uniform and high solder paste transfer to the PCB, lasercut (mostly made from stainless steel) or electroformed stencils (Nickel) should be preferred.

Table 4 Recommendations for Stencil Dimensions

Package	Stencil Aperture Diameter [µm]	Stencil Thickness [µm]
P(G)-HBGA, P(G)-FCHBGA, Ball pitch 1.00 mm	400 - 450	100 - 150
P(G)-HBGA, P(G)-FCHBGA, Ball pitch 1.27 mm	600 - 650	100 - 150

4.2 Solder Paste

Solder paste consists of solder alloy and a flux system. Normally the volume is split into about 50% alloy and 50% flux. In term of mass this means approx. 90 wt% alloy and 10 wt% flux system. The flux system has the function to remove the contaminations from the solder joints during the soldering process. The capability of removing contaminations is given by the respective activation level. The solder paste metal alloy has to be of leaded eutectic or near-eutectic composition (SnPb or SnPbAg) or lead-free composition (SnAgCu whereas Ag 3 - 4%, Cu 0.5 - 1%). A “no-clean” solder paste is preferred, because cleaning under the soldered P(G)-HBGA / P(G)-FCHBGA 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 age, temperature and humidity. Please notice the handling recommendations of the paste manufacturer.

4.3 Component Placement

P(G)-HBGA and P(G)-FCHBGA packages have to be placed accurately according to their geometry. Positioning the packages manually is not recommended.

Component placement accuracies of ±50 µm are obtained with modern automatic component placement machines using vision systems. With these systems both the PCB and the components are optically measured and the components are placed on the PCB at their programmed positions. The fiducials on the PCB are located either on the edge of the PCB for the entire PCB or additionally on individual mounting positions (local fiducials). They are detected by a vision system immediately before the mounting process. Recognition of the packages is performed by a special vision system, enabling a correct centering of the complete package.

If they are slightly displaced, packages with solder balls as the P(G)-HBGA / P(G)-FCHBGA have the favourable property of self alignment during the reflow process due to the solder’s high surface tension. As a rule of thumb a maximum tolerable displacement of the components is 30% of the metal pad diameter on the PCB (for non solder mask defined pads). In consequence, for P(G)-HBGA / P(G)-FCHBGA packages the solder ball to PCB pad misalignment has to be better than 150 µm to assure a robust mounting process. Generally this is achievable with a wide range of placement systems.

The following remarks are important:

- Especially on large boards local fiducials close to the device can compensate a large amount of PCB tolerances.
- It is recommended to use the ball recognition capabilities of the placement system, not the outline centering. This eliminates the solder ball to package edge tolerances of the package (please refer to the respective package outline drawing for details).
- To ensure the identification of the packages by the vision system, an adequate lighting as well as the correct choice of the measuring modes are necessary. The accurate settings can be taken from the equipments manuals.
- Too much placement force can lead to squeezed out solder paste and cause solder joint shorts. On the other hand too low placement force can lead to insufficient contact between package and solder paste and this can lead to open solder joints or badly centered packages.

4.4 Soldering

Soldering determines the yield and quality of assembly fabrication to a very large extent. Generally all standard reflow soldering processes

- forced convection
- vapour phase
- infrared (with restrictions)

and typical temperature profiles are suitable for board assembly of the P(G)-HBGA / P(G)-FCHBGA. Wave soldering is 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 PCB with its components has to be avoided. Please refer to the bar code label on the packing for the peak package body temperature. When 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 P(G)-HBGA / P(G)-FCHBGA. The most recommended type is forced convection reflow. Nitrogen atmosphere can generally improve solder joint quality, but is normally not necessary for soldering tin-lead metal alloys.

The temperature profile of a reflow process is one of the most important factors of the soldering process. The temporal progression of the temperature profile is divided into several phases, each with a special function. **Figure 6** shows a general forced convection reflow profile for soldering P(G)-HBGA / P(G)-FCHBGA packages, **Table 5** shows an example of the key data of such a solder profile for Tin-lead and for lead-free alloys. The single parameters are influenced by various facts, not only by the package. It is essential to follow the solder paste manufactures application notes, too. Additionally, most PCBs contain more than one package type and therefore the reflow profile has to be matched to all components' and materials' demands. We recommend to measure the solder joints' temperatures by thermocouples beneath the respective packages. It has to be considered, that components with large thermal masses don't heat up in the same speed as lightweight components, and also the position and the surrounding of the package on the PCB, as well as the PCB thickness can influence the solder joint temperature significantly. Therefore no concrete temperature profile can be given.

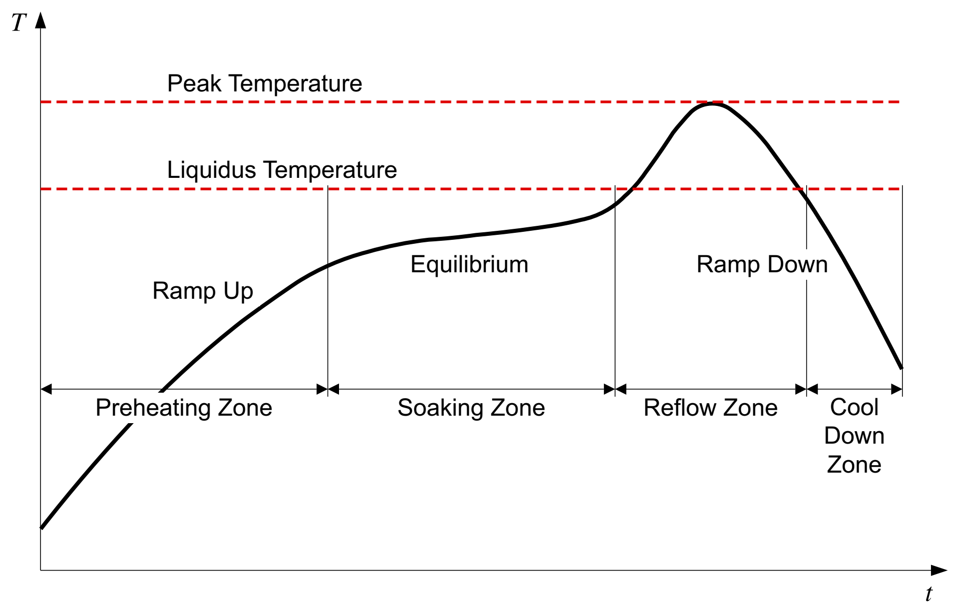


Figure 6 General Forced Convection Reflow Solder Profile

The following table is an example, not a recommendation (for reference only).

Table 5 EXAMPLE for the Key Data of a Forced Convection Reflow Solder Profile

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)
Reflow time over Liquidus	60 s	60 s	Alloy (Solder paste)
Cool down rate	2.5 K/s	2.5 K/s	

4.4.1 Double-Sided Assembly

P(G)-HBGA / P(G)-FCHBGA packages are generally suitable for mounting on double-sided PCBs. That means that in a first step one side of the PCB is fitted with components and soldered. Afterwards the second side of the PCB is fitted with components and soldered again. Be aware that especially packages with a high mass could fall of during the second reflow process, while heading face down. In such cases this packages have to be assembled with the last (= second) reflow process. As a rule of thumb a weight limit of 0.2 g/mm^2 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.

4.4.2 Processing of Moisture-Sensitive Components

For moisture-sensitive packages like P(G)-HBGA and P(G)-FCHBGA it is necessary to control the moisture content of the components. The penetration of moisture into the package molding compound is generally caused through exposure to the ambient air for a few days. Moisture absorption leads in many cases to moisture concentrations in the component which are high enough to destroy the package during the solder cycle (“popcorn effect” due to the evaporation of the humidity). Hence the necessity to dry moisture-sensitive components, to seal

them in a moisture-resistant bag and only to remove them immediately prior to processing (soldering onto the PCB). The permissible time (from opening the moisture barrier bag until the final soldering process) that a component can remain unprotected in an environment with a level of humidity approximating to real-world conditions (e.g. 30°C/60% RH) is a measure of the sensitivity of the component to ambient humidity (Moisture Sensitivity Level, MSL). The most commonly applied standard IPC/JEDEC J-STD-033* thus defines eight different MSLs (see **Table 6**). Please refer to the "Moisture Sensitivity Caution Label" on the packing material, which contains informations about the moisture sensitivity level of your packages.

Table 6 Moisture Sensitivity Levels (acc. 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

Internet Link to **Association Connecting Electronics Industries (IPC)**

If moisture-sensitive components have been exposed to ambient air longer than the specified time according to their MSL, or the humidity indicator card dot 10% is wet (read 1 minute after bag opening), the packages have to be baked prior to the assembly process. Please refer to IPC/JEDEC J-STD-033* for bake procedure. Baking a package too often can cause solderability problems due to oxidation and/or intermetallic growth. Notice that packing material possibly can not withstand the baking temperature. See imprints/labels on the respective packing for maximum temperature.

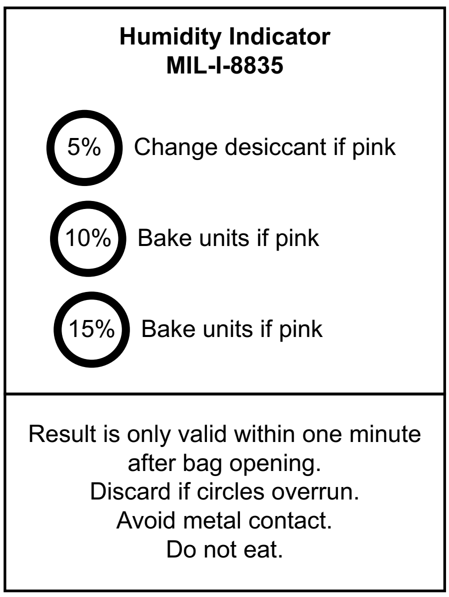


Figure 7 Humidity Indicator

4.4.3 Special Notes for Green (“G”) Packages

Like in Tin-lead-process, generally all commonly used reflow soldering processes (vapour phase, forced convection, infrared) are suitable for board assembly. But due to a higher peak temperatures at the lead-free process, some equipment can reach the limits of their technical capability. Normally reflow with forced convection is chosen. Due to the aggravated conditions at the lead-free process, nitrogen atmosphere may reduce oxidation and improve the solder joint quality. Be aware, that the increasing temperature stress may worsen the MSL. Therefore the data for lead-free packages names 2 MSLs: One for a lower reflow peak temperature (tin-lead) and one for a higher reflow peak temperature (lead-free). Each one is valid for the respective application.

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 table descirbes the possible combinations.

Table 7 Possible Combinations of Lead-Based and Lead-Free Solder Balls and Solder Paste

	SnPb(Ag) Solder Paste (= low peak temperature)	**SnAgCu Solder Paste (= high peak temperature)**
SnPb(Ag) solder ball	ok	ok
SnAgCu solder ball	with restrictions (see the following notes)	ok

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

Table 8 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 - 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 exactly on the solder paste. With peak temperature over 230°C suitable solder joints are formed.

4.5 Cleaning and Subsequent Processing

Cleaning

After the reflow soldering process some flux residues can be found around the solder joints. If a “no-clean” solder paste has been used for solder paste printing, the flux residues usually don’t have to be removed after the soldering process. Be aware, that cleaning beneath a P(G)-HBGA / P(G)-FCHBGA package is difficult because of the small gap between package substrate and PCB and is therefore not recommended. Whether the solder joints have to be cleaned, however, the cleaning method (e.g. ultrasonic, spray or vapor cleaning) and solution has to be selected with consideration of the packages to be cleaned, the used flux in the solder paste (rosin-based, water-soluble, etc.), environmental and safety aspects. Removing/Drying even of small residues of the cleaning solution should also be done very thorough. Contact the solder paste manufacturer for recommended cleaning solutions.

Underfilling (optional)

The application of an underfill material under the package is an efficient method to improve the board level reliability regarding mechanical and thermomechanical stress, especially for smaller ball pitches.

A liquid epoxy resin is dispensed on the PCB next to the package outline at a distance that allows the material to wet the gap between package and board. Through capillary action the underfiller is then driven into the gap and fills it completely.

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

Typically liquid, filled or unfilled epoxy resins are used for package underfilling. Ultra fine-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).

4.6 Inspection

A visual inspection of the solder joints with conventional AOI (automatic optical inspection) systems is not possible. The only reasonable method to realize an efficient inline control is the implementation of AXI (automatic X-ray inspection) systems. AXI systems are available as 2D and 3D solutions. They usually consist of an X-ray camera and the hard- and software needed for inspection, controlling, analysing and data transfer routines. These systems enable the user to detect soldering defects like poor soldering, bridging, voiding and missing parts quite reliable. But other defects like broken solder joints are not easily detectable by X-ray. For the acceptability of electronic assemblies please refer also to the IPC-A-610C standard.

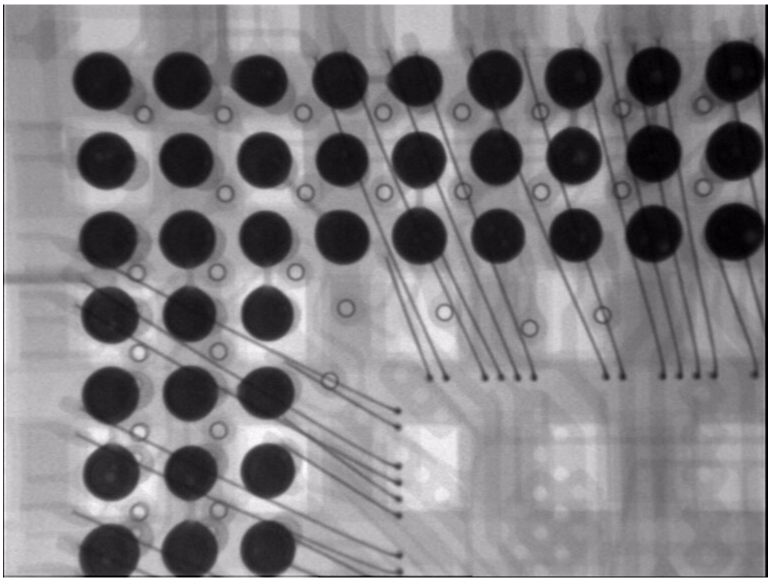


Figure 8 Typical X-ray Image of Soldered BGA Package

The newest developments in inspection equipment are endoscopes. With them a lot of failures can be detected beneath BGAs. The optical head of the system moves around the package near the PCB area. The user can look along the solder rows by adjusting the focus. The pictures from such a endoscope system are much more easy to interpret than X-ray images.

Cross sectioning of a soldered package as well as dye penetrant analysis can serve as tools for sample monitoring only, because of their destructive character.

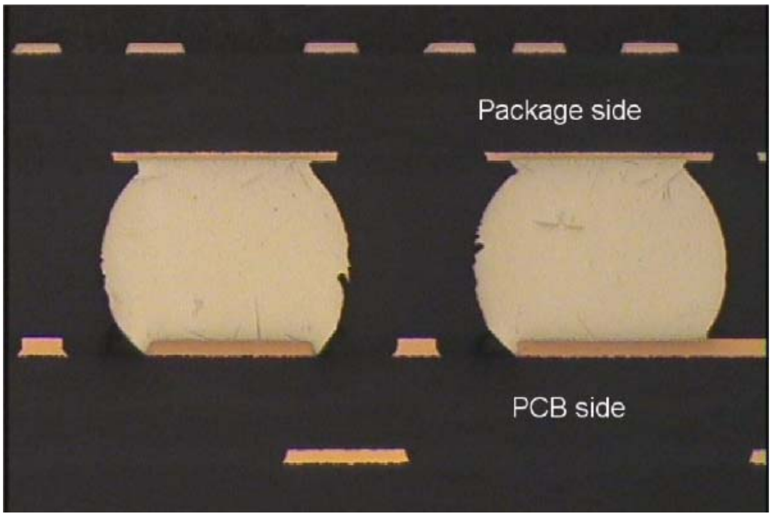
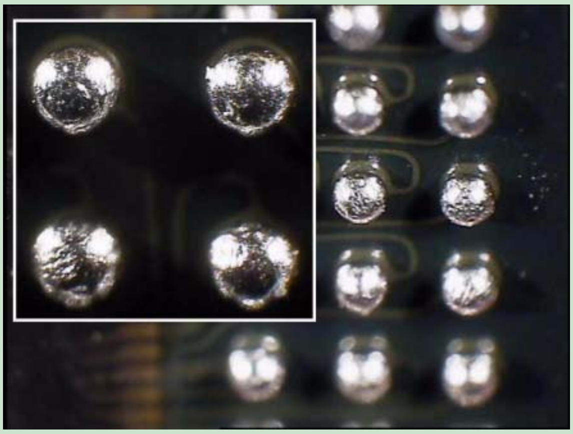


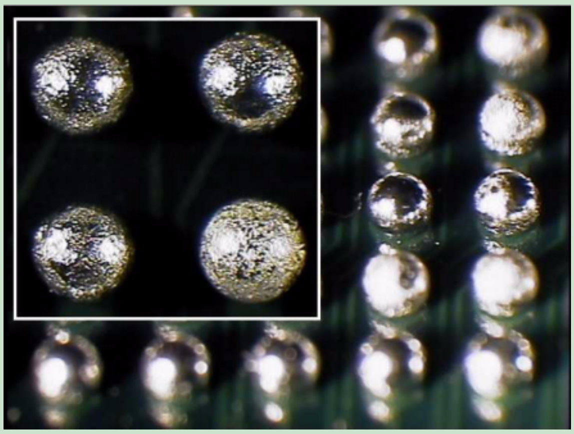
Figure 9 Sample Cross Section of BGA Solder Balls

Special notes for green (“G”) packages:

Lead-free solder joints look different than tin-lead (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 (see Figure 10). 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.



SnPb Solder Balls: Shiny



SnAgCu Solder Balls: Dull

Figure 10 Comparison of Solder Ball Appearance

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.

5 Rework

If a defect component is observed after board assembly the device can be removed and replaced by a new one. Repair of components' single solder joints is not possible.

5.1 Tooling

The rework process is commonly done on special rework equipment. There are a lot of systems available on the market, and for processing these packages the equipment should fulfill the following requirements:

- *Heating*: Hot air heat transfer to the package and PCB is strongly recommended. Temperature and air flow for heating the device should be controlled. With free-programmable temperature profiles (e.g. by PC controller) it is possible to adapt the profiles to different package sizes and masses. PCB preheating from underside is recommended. Infrared heating can be applied, especially for preheating the PCB from underside, but it should be only supporting the hot air flow from the upside. Instead of air also nitrogen can be used.
- *Vision system*: The bottom side of the package as well as the site on the PCB should be observable. For precise alignment of package to PCB a split optic should be implemented. Microscope magnification and resolution should be appropriate for the pitch of the device.
- *Moving and additional tools*: The device should be relocatable on the whole PCB area. Placement accuracy is recommended to be better than ±100 µm. The system should have the capability of removing solder residues from PCB pads (special vacuum tools).

5.2 Device Removal

If it is intended to send a defect component back to the supplier, please note that during the removal of this component no further defects must be introduced to the device, because this may hinder the failure analysis at the supplier. This includes the following recommendations:

- *Moisture*: According to his moisture sensitivity level, possibly the package has to be dried before removal. If the maximum storage time out of the dry pack (see label on packing material) is exceeded after board assembly, the PCB has to be dried according to the recommendations (see **Chapter 4.4**), otherwise too much moisture may have been accumulated and damage may occur (popcorn effect).
- *Temperature profile*: During soldering process it should be assured that the package peak temperature is not higher and temperature ramps are not steeper than for the standard assembly reflow process (see **Chapter 4.4**).
- *Mechanics*: Be aware not to apply high mechanical forces for removal. Otherwise failure analysis of the package can be impossible or PCB can be damaged. For large packages pipettes can be used (implemented on most rework systems), for small packages tweezers may be more practical.

5.3 Site Redressing

After removing the defect component the pads on the PCB have to be cleaned from solder residues. Don't use steel brushes because steel residues can lead to bad solder joints. Before placing a new component it is recommended to apply flux by dispensing it on each PCB pad or by oblitering it over the whole site (e.g. by a brush). It is recommended to use only no-clean flux, if possible the same one that is used in your solder paste for stencil printing.

5.4 Reassembly and Reflow

After preparing the site, the new package can be placed onto the PCB. The package is positioned exactly above the PCB pads, in height just that there is no contact between the package and the PCB and the package is then dropped into the printed or dispensed flux or solder paste depot (Zero-force-placement). During soldering process it should be assured that the package peak temperature is not higher and temperature ramps are not steeper than for the standard assembly reflow process (see **Chapter 4.4**).

Special notes for green ("G") packages:

Packages with lead-free balls can be reworked like leaded packages, but peak temperatures have to be adjusted according to the used solder (see also **Chapter 4.4**).

www.infineon.com

Published by Infineon Technologies AG