Recommendations for board assembly of Infineon optical sensor packages with ball grid array

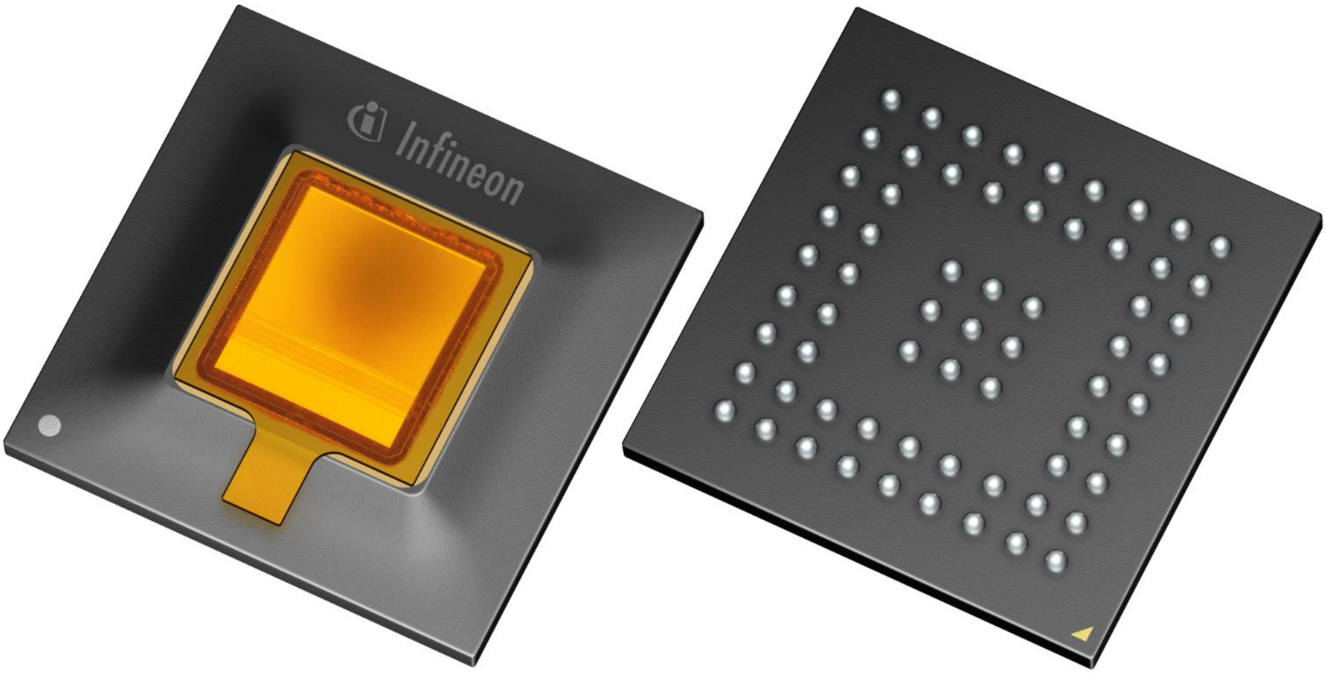

Table of Contents

Acronyms and Abbreviations

AOI	Automated Optical Inspection
BGA	Ball Grid Array
ESD	Electrostatic Discharge
IC	Integrated Circuit
IR	Infrared
LFBGA	Low-profile Fine-pitch Ball Grid Array
NSMD	Non-Solder Mask Defined
PG	Plastic Green
PCB	Printed Circuit Board
SAC	Tin Silver Copper (SnAgCu)
SMD	Solder Mask Defined
SMD	Surface-Mount Device
SMT	Surface-Mount Technology

1 Package Description

This document provides information about the board assembly of packages with optical sensor window. The termination arrangement is of Ball Grid Array (BGA) configuration. The lead-free solder balls allow for assembly by Surface Mount Technology (SMT).

This document does not discuss BGA packages in general or MEMS packages. These package families are described in a separate document.

1.1 Package Type

The Infineon optical sensor Low-profile Fine-pitch Ball Grid Array (LFBGA) package is square and provides a top side cavity for the sensor chip. The sensor area and imager chip are covered by a glass plate. Figure 1 shows two available types of optical sensor LFBGA package – with and without overmold.

- PG-LFBGA packages

PG = Plastic Green
LF = Low-profile Fine-pitch
BGA = Ball Grid Array

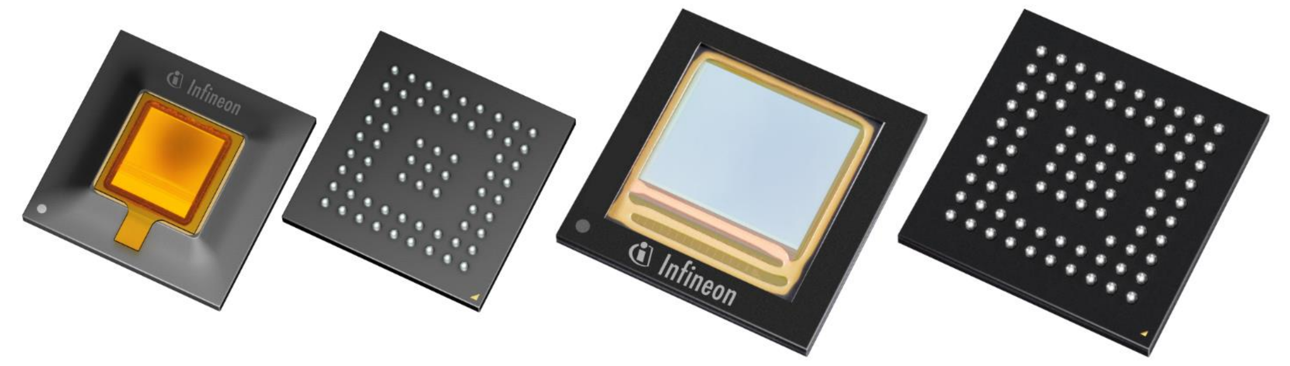

Figure 1 **Types of optical sensor LFBGA package.**

1.2 Package Features and General Handling Guidelines

Infineon optical sensor packages can be handled using standard industry SMT equipment and processes. However, care should be taken to avoid affecting the optical path by damaging or contaminating the sensor or its package. The following items outline the most sensible topics.

ESD and Radiation Precautions

- Notwithstanding the potential presence of protection circuitry, damage may occur on devices subjected to high-energy electrostatic discharge. Since charged devices and circuit boards can discharge without detection, proper Electrostatic Discharge (ESD) precautions should be taken during transport, storage, handling, and processing to avoid performance degradation or loss of functionality.
- The optical sensors are sensitive to electromagnetic radiation. The optical sensor device in the LFBGA package must not be exposed to X-ray radiation higher than 0.5 Gray, as it can deteriorate its performance.

For further information about ESD protective measures, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon Technologies web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

Package Handling and Assembly Precautions

- Do not use excessive force to place the component on the PCB. The use of standard industry pick-and-place tools is recommended in order to limit the mechanical force exerted on the package.
- The sensor glass shall be handled with utmost care during the entire assembly procedure to prevent scratching, chipping, or contamination by e.g. particles or else.
- During assembly the polyimide protection cover tape shall be kept on the sensor window during the entire assembly process including potential PCB sawing or system assembly. Please be aware, that a residual free adhesion of the polyimide tape can only be guaranteed for 12 months of storage time.
- Semiconductor devices are generally sensitive to moisture, mechanical handling, and contamination. Therefore, they require specific precautionary measures to ensure that they are not damaged during transport, storage, handling, and processing.
- The reflow profile should be optimized to avoid excessive flux evaporation or spattering.
- If necessary, cleaning the sensor glass shall be executed as described in the cleaning section of this document.

For further information about component handling, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

Internal Construction

The optical sensor LFBGA technology combines the BGA configuration of a multi-layer glass woven epoxy substrate with a cavity package approach. The Integrated Circuit (IC) is attached to the substrate and then additionally fixed and protected by liquid epoxy.
In case of the package type with overmolding the optical sensor LFBGA package is map-molded, meaning it is molded in an array prior to its singulation by sawing. Consequently in this case the mold cap extends to the substrate edge and levels the package top surface to the sensor glass.
Figure 2 shows a schematic drawing of the inner setup of the two different types of optical sensor LFBGA type. First the type without overmolding, which results in a package topside, which is only flat at the sensor glass area. Secondly the type with overmolding, which results in a flat package topside.

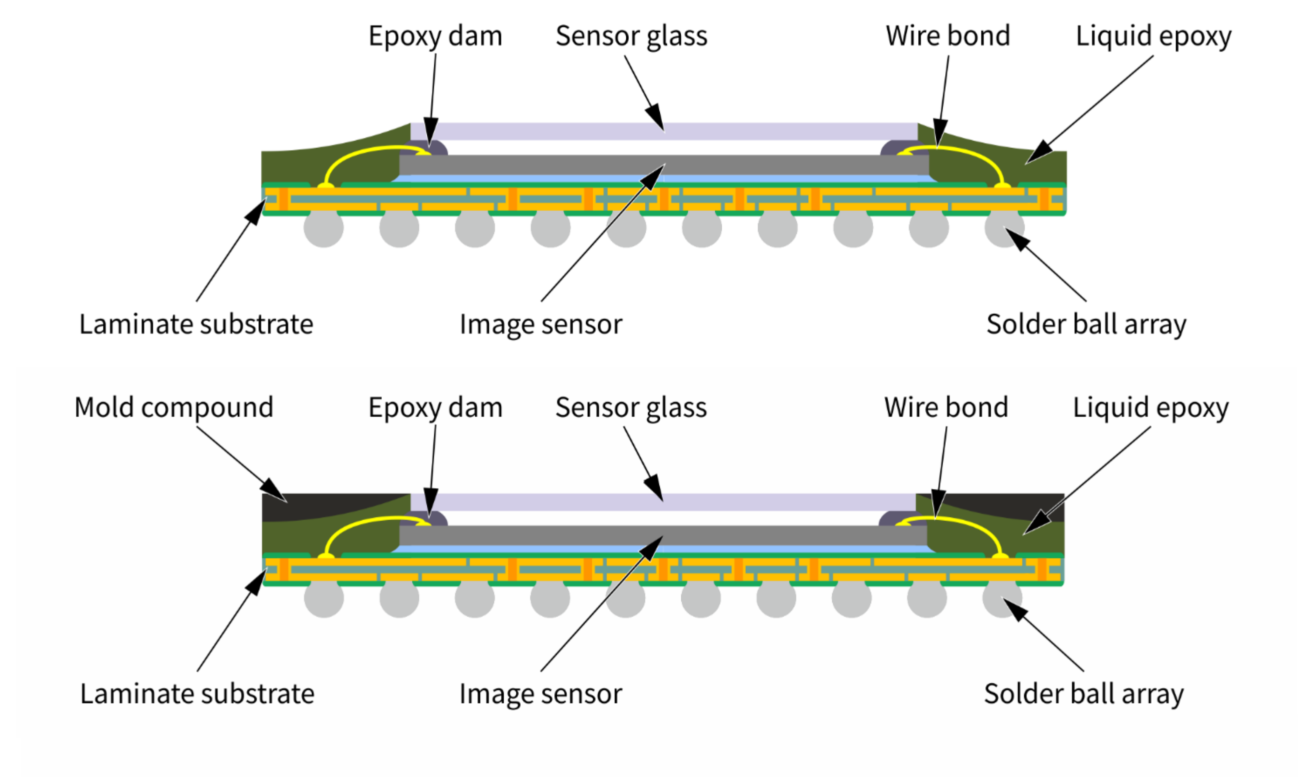

Figure 2 **Schematic of the inner setup of the optical sensor LFBGA package without (top) and with overmolding (below).**

Termination Design

Generally, the solder-ball of a BGA package is formed on a Solder-Mask-Defined pad (SMD) of the package substrate. Figure 3 shows a schematic of such a solder ball.

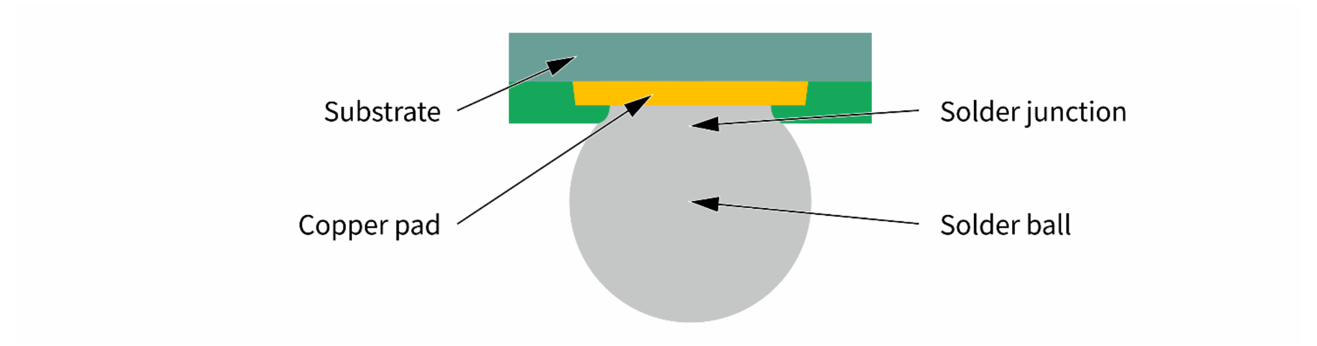

Figure 3 **Schematic of a SMD ball pad on an optical sensor LFBGA package.**

Sensor Glass

The optical signal is delivered to the sensor through a cover glass with an anti-reflective coating. The glass is covered by a polyimide tape with silicone adhesive for protection during handling and assembly from reel to soldering cool-down. Figure 4 shows photographs of the two types of optical sensor LFBGA package with such a protective tape.

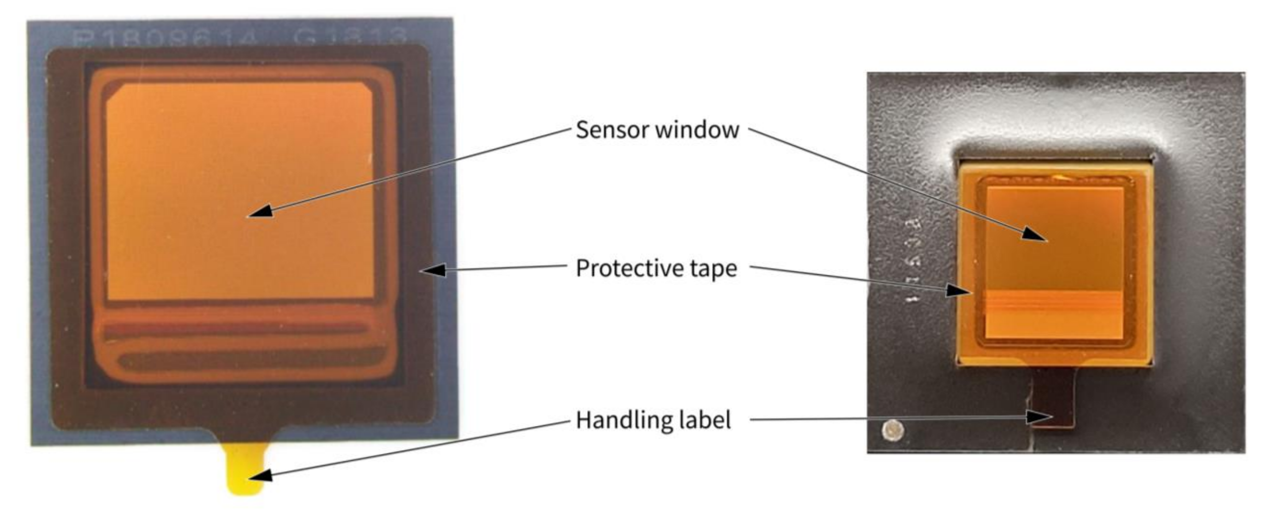

Figure 4 **Photographs of the two types of optical sensor LFBGA package with protective polyimide tape for sensor glass protection during handling and board assembly.**

The film on the sensor glass is for protective purposes only and can be expected to have a negative impact on the optical performance of the sensor. Therefore, the protective tape should be removed prior to testing of the final product. The protective tape can be peeled using tweezers and the handling label (see Figure 4) while complying with standard ESD precautions.

Note: Please be aware, that a residual free adhesion of the polyimide tape can only be guaranteed for 12 months of storage time.

For more detailed information about cleaning the sensor component before and after the tape removal, please consider the cleaning section in this document

2 Printed Circuit Board

2.1 Routing

The main difference between components with solder balls such as BGA and conventional SMD such as leadframe-based components is the array configuration of solder spheres on the package. This implies different concepts for routing the signal, power, and ground pins on the Printed Circuit Board (PCB). Typically fine-line PCBs with conductor width/spacing of 100 µm are necessary for routing. The specifics of the PCB design to be used strongly depend 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.

The PCB design and construction are key factors for achieving highly reliable solder joints. BGA packages should not be placed opposite one another on the PCB (if double sided mounting is used), because this stiffens the assembly and results in earlier solder-joint fatigue compared to a design where the components are offset. Furthermore, the board stiffness itself has a significant effect on the reliability (temperature cycling) of the solder-joint interconnect if the system is used in critical temperature-cycling conditions. The lower bending stiffness of thinner boards (e.g. 1.0 mm) improves solder-joint reliability (temperature cycling) compared to thick boards (e.g. 2.35 mm).

2.2 Pad Design

The quality and reliability of interconnect solder joints to the board are affected by:

- Pad type (Solder Mask Defined, SMD or Non-Solder Mask Defined, NSMD)
- Specific pad dimensions
- Pad finish (also called metallization or final finish)
- Via layout and technology

The NSMD pad type on PCB is preferred since it allows for appropriate pad precision and good board reliability. Figure 5 shows the recommended NSMD PCB pad design for BGA packages. The dimensions recommended for individual pad diameter and solder mask clearance depend on the specific component, and can be found in the package data base that is available on the Infineon web page [1]. Please choose a specific package when searching the data base, which will then show an example of the stencil aperture layout for each package. Design details will depend on the PCB technology used, the capability of the suppliers, and the target routing.

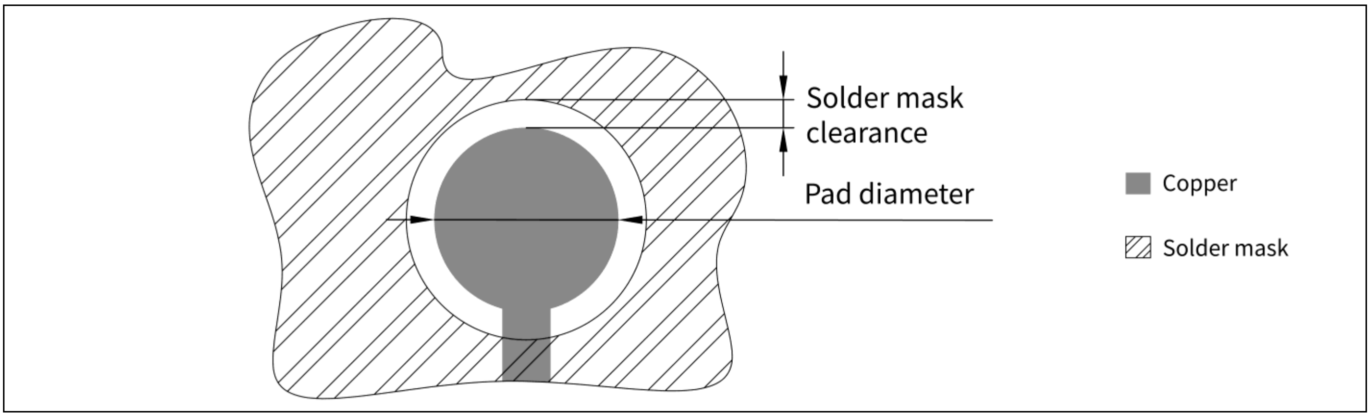

Figure 5 **Recommended NSMD PCB pad design for BGA components. Pad diameter and solder-mask clearance depend on the individual components, and can be found in the Infineon package data base that is available on the Infineon web page [1].**

Figure 6 shows an example of a BGA solder ball with SMD pad design on the component side and with NSMD pad design on the PCB side. Investigations on board-level reliability within Infineon have shown that in most cases this configuration results in the highest solder-joint reliability. The NSMD pad on the PCB allows the solder to grip the PCB pad sidewalls, improving the reliability of the solder joint.

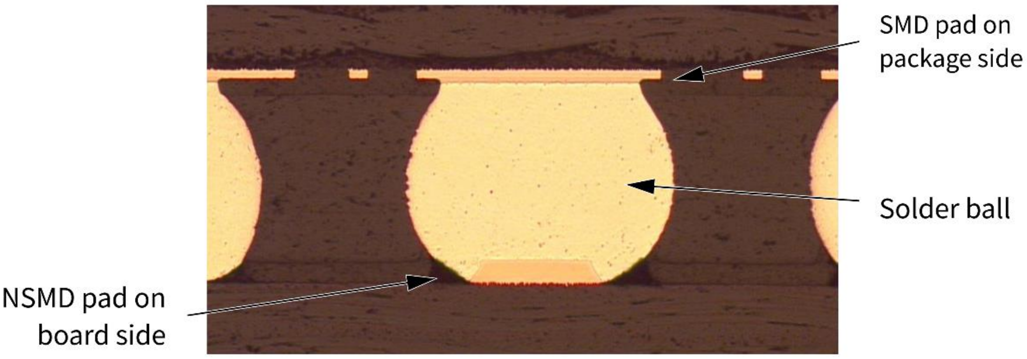

Figure 6 **Cross-section of a BGA solder ball with SMD pad on the component side and NSMD pad on the board side. The latter allows the solder to "grip" the pad sidewalls increasing the board level reliability of the component.**

If any kind of via hole is placed inside the pad area, its opening should be closed by plugging or plating in order to prevent solder flow into it. If closed microvias are placed inside the pads, their intersection with the pad should be as flat as possible. Deep dips inside the pads may cause voiding in the solder joint even if the microvia opening is closed.

For further information about PCB pad and via design, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3 PCB Assembly

3.1 Solder Paste Stencil

In SMT, the solder paste is applied onto the PCB metal pads by stencil printing. The volume of the printed solder paste is determined by the stencil aperture and the stencil thickness.

The stencil aperture for BGA packages should be circular. The stencil thickness can vary between 100 and 150 µm depending on aperture area ratio, the stencil material, and the aperture diameter. Specific stencil aperture recommendations can be found in the package data base that is available on the Infineon web page [1]. Please choose a specific package when searching the data base, which will then show an example of the stencil aperture layout for each package.

To ensure a uniform and sufficient solder paste transfer to the PCB, laser-cut stencils (mostly made from stainless steel) or electroformed stencils (nickel) are preferred. For individual design adaptations to use the optimum amount of solder, the stencil thickness, the PCB pad finish, quality and solder masking, the via layout, and the solder paste type should be considered. In every case, application-specific experiments are recommended.

For further information about solder stencil design, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3.2 Solder Paste

Pb-free solder pastes typically contain some type of SnAgCu alloy (SAC solder with typically 1-4% Ag and <1% Cu). The most common alloy is SAC305 (3.0% Ag and 0.5% Cu). The average alloy particle size must be suitable for printing the solder stencil aperture dimensions. Using Type 3 or Type 4 paste is recommended for the assembly of BGA components, depending on the specific stencil transfer.

The solder alloy particles are dispersed in a blend of liquid flux and chemical additives (approx. 50% by volume or 10% by weight), forming a creamy paste. The flux and chemical solvents have various functions such as adjusting the viscosity of the paste for stencil printing or removing contamination and oxides from the surface.

The solder paste solvents have to evaporate during reflow soldering, while residues of the flux will remain on the solder joint. The capacity of the flux additive for removing oxides is given by its activation level, which also affects the potential need for removing the flux residuals after the assembly. For area-array packages such as BGA, in which the solder joint is formed mainly on the bottom side, a “no clean” paste is recommended to avoid subsequent cleaning underneath the package. The small gaps make cleaning highly difficult if not impossible. Certain precautions have to be taken if any kinds of flux residues remain on the board prior to any kind of coating.

Generally, solder paste is sensitive to ageing, temperature, and humidity. Please follow the handling recommendations of the paste manufacturer.

3.3 Component Placement

Although the self-alignment effect due to the surface tension of the liquid solder will support the formation of reliable solder joints, the components have to be placed accurately depending on their geometry. Positioning the packages manually is not recommended, especially for packages with small terminations and pitch. An automated pick-and-place machine is recommended to obtain reliable solder joints.

Component placement accuracies of +/-50 µm and less 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 at additional individual mounting positions (local fiducials). These fiducials are detected by a vision system immediately prior to the mounting process. Most such vision systems provide special lighting and algorithms for area array packages. For BGA components it is recommended to use the ball recognition instead of the component outline recognition for centering. This approach eliminates potential tolerances between the solder ball and the package.

The polyimide tape shown in Figure 4 protects the sensor glass from reel to soldering cool-down. However, when placing the package, the applied force shall be sufficiently low not to damage the cover glass. Excessive placement force can also squeeze the solder paste out of its intended location and cause solder joint shorts.

Especially for the non-overmolded optical LFBGA type (please refer to Figure 2) the nozzle design has to be considerd more thoroughly. If the nozzle area is larger than the sensor glass the following topics have to be taken in account. The higher topology of this package does not allow vacuum to act on the outer, uneven area and the smaller size of the polyimide tape could allow vacuum to act near to the edge of the tape and may result in peel-off. On the other hand a too small nozzle may not be able to hold the component during handling (too less suction power).

For further information about component placement, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3.4 Reflow Soldering

For printed PCB assembly of the BGA components, the widely used method of reflow soldering in a forced convection oven is recommended. Soldering in a nitrogen atmosphere can generally improve the solder joint quality but is not necessary to create a reliable joint. Vapor phase or infrared (IR) soldering is not recommended.

The soldering profile should be in accordance with the recommendations of the solder paste manufacturer to achieve optimal solder-joint quality. The position and the surrounding of the component on the PCB, as well as the PCB thickness, can influence the solder joint temperature significantly.

Minimum Reflow Conditions

The lower temperatures and durations of an optimal reflow profile shall stay above those of the wettability test profile of a solderability qualification. The solderability of the terminations of Infineon components is tested according to the standards IEC 60068-2-58 and J-STD-002 [2][3].

Maximum Reflow Conditions and Cycles

Components that are Moisture-Sensitivity Level (MSL) classified by Infineon have been tested by three reflow runs in accordance with the J-STD-020 standard, including a double-sided reflow and one rework cycle. The maximum temperatures must not be exceeded during board assembly. Please refer to the product barcode label on the packing material that states this maximum reflow temperature according to the J-STD-020 [4] standard as well as the MSL according to the J-STD-033 standard [5].

Note: The protective tape is designed to withstand the same reflow conditions as the component.

BGA packages are generally suited for double-sided PCB mounting. That means that both sides of the PCB are fitted and reflowed one after another. As a consequence, the side that was initially assembled experiences two reflow cycles. During the second cycle, the components are hanging upside-down. Therefore, during the peak zone of the reflow profile (where the solder is liquid), the components are only held by wetting forces of the molten solder. Gravity acting in the opposite direction will elongate the solder joints, unlike joints on the top side, where gravity forces the components nearer to the PCB surface. This shape will be frozen during cooling and therefore will result in a higher stand-off on the bottom side after the reflow process. Any mechanical impacts on components that are soldered upside down should be avoided.

Although BGA components have a natural tendency for warpage due to their layered construction. Investigations have shown, that the optical sensor LFBGA package shows a low warpage compared with standard BGA packages.

For further information about reflow soldering, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

4 Cleaning

The protective tape shall be kept on the sensor glass during the entire assembly procedure. It consists of a polyimide film with silicone adhesive. Cleaning prior to tape removal should not use chemicals which would affect either of these materials. Especially chemicals such as methanol or acetone shall be avoided due to their toxicity and corrosive nature.

After the soldering process, some flux residues will remain on the board, especially near the solder joints. Generally, cleaning beneath a component with bottom-only terminations is difficult due to the small gap between the component body and the PCB. Therefore, a “no-clean” flux is recommended whose residues usually do not have to be removed after the soldering process.

In case the solder joints have to be cleaned, the cleaning method (e.g. ultrasonic, spray, or vapor cleaning) and cleaning solution have to be selected while taking into account the type of package, the flux used in the solder paste (rosin/resin-based, water-soluble, etc.) as well as the environmental and safety aspects. Even small residues of the cleaning solution should be removed or dried out very thoroughly. For recommended cleaning solutions, please contact the solder paste or flux manufacturer.

In case the sensor glass has to be cleaned after protective tape removal, special focus should be on the following items:

- The cover glass shall not be touched with bare fingers since the grease can etch optical coatings and cause permanent damage.
- Gloves should be static dissipative and powder free.
- Use only high-grade IPA (100% pure) and ESD protective lint-free wipes for wet cleaning. Other solvents such as acetone are attacking parts of the package or posing the risk of contaminating the glass. Never use an already used area of the wipe twice.
- Use clean compressed nitrogen for removing loose particles or for drying after wet cleaning.
- In case the glass does not look clean even after several wipe strokes inspect the surface in optical microscope for permanent damage.

5 Inspection

5.1 Optical Solder Joint Inspection

A visual inspection of the BGA solder joints with conventional AOI (Automated Optical Inspection) systems is not possible. As can be seen in Figure 7, even a side-view can only reveal a small portion of the solder balls in their area array configuration.

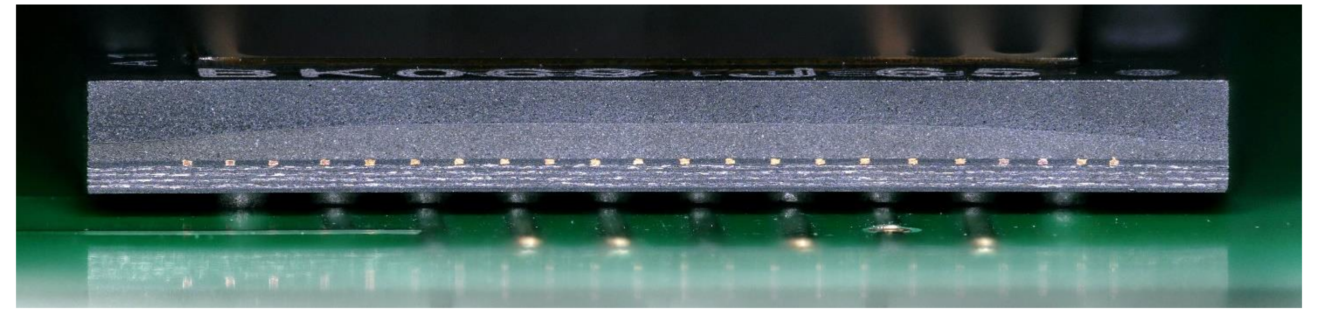

Figure 7 Side view of a mounted optical sensor LFBGA package. The package mold, the package substrate and the solder balls can be seen from top to bottom.

For engineering tasks, cross-sectioning can offer detailed information about the solder joint quality. Due to its destructive character, cross-sectioning during monitoring is naturally not practical.

5.2 X-Ray Solder Joint Inspection

X-ray inspection systems are appropriate for control of components such as LFBGA whose solder joints cannot be inspected properly by optical systems. The optical sensor device in the LFBGA package must not be exposed to X-ray radiation higher than 0.5 Gray. In order to avoid sensor damage a dose measurement prior to inspection is required. X-Ray systems are available as 2D and 3D solutions. They usually consist of an X-ray camera and the hardware and software needed for inspection, controlling, analyzing, and data transferring routines. These reliable systems enable the user to detect soldering defects such as poor soldering, bridging, voiding, and missing parts. However, other defects such as broken solder joints are not easily detectable by X-ray.

Note: The optical sensor device in the LFBGA package must not be exposed to X-ray radiation higher than 0.5 Gray, as it can deteriorate its performance.

Figure 8 shows a section of an X-ray photograph of a properly soldered optical sensor LFBGA component. Balls and bond wires as well as vias are visible. X-ray images might also reveal a certain amount of voiding in the solder balls. The extent of voiding depends on the solder paste, the reflow profile, and the presence of microvias-in-pad.

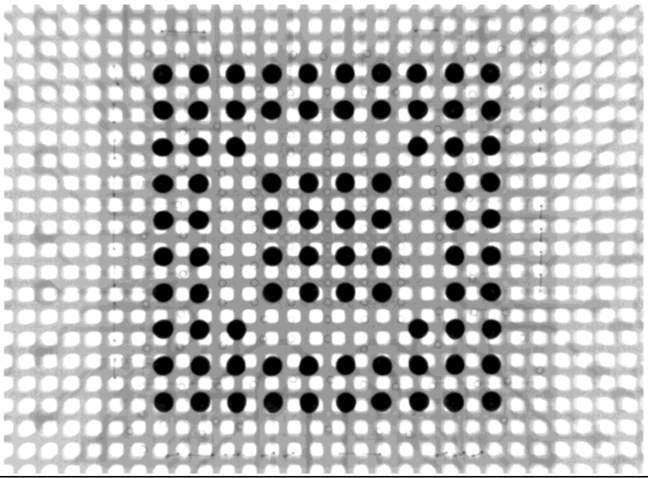

Figure 8 **X-ray photograph of a properly soldered optical sensor LFBGA package. Balls and bond wires as well as vias are visible.**

For the acceptability of electronic assemblies, please also refer to the IPC-A-610 standard [6].

For further information about the on-board inspection of Infineon MEMS packages with open sensor ports please contact your local sales, application, or quality engineer.

6 Rework

Single solder joint repair of bottom-only terminated packages is highly difficult, if not impossible, and is therefore generally not recommended. Furthermore, the reuse of de-soldered components is not recommended. The de-soldered components should be replaced by new ones.

A rework process is commonly done on special rework equipment. There are various systems available that meet the requirements for reworking SMD packages. All handling guidelines discussed in this document have to be respected. Special focus should be on the following items:

- Due to the decreased automation level given by the general rework approach, even higher care compared to standard assembly must be taken. Tools that do not damage the component mechanically have to be chosen. Mechanical forces that do not necessarily cause visible external damage can still cause internal damage that reduces the component’s reliability. A proper handling system with vacuum nozzle may be the gentlest process and is therefore recommended. However, the impact of rework tools has to be assessed properly. In general, more manual handling increases the effort for documentation, training, and monitoring of the rework process(es).
- During rework, special care must be taken concerning the proper moisture level of the component according to the J-STD-033. Drying the PCB and the component prior to rework might be necessary. A proper drying procedure for SMD packages is described in the international J-STD-033 standard [5]. Please also refer to the recommendations of your PCB manufacturer and take all specific needs of components, PCB, and other materials into account.
- Whatever heating system is used (hot air, infrared, hot plate, etc.), the applied temperature profile at the component must never exceed the maximum temperature according to the J-STD-020 standard. Depending on the specific heating profile used during rework, components adjacent to the mounting location might also experience a further “reflow run” in terms of the J-STD-020 standard [4]. Internal investigations have shown that the temperature profile must be recorded.

If a device is suspected to be defective and a failure analysis is planned, Infineon usually expects customers to desolder the component prior to return to Infineon. The component shall be returned in a proper condition according to the original package outlines.

In some special cases such as solder joint inspection Infineon may request that the PCB or part of the PCB with the component still attached should be sent to Infineon.

Note: Before returning a device for failure analysis at Infineon, please clarify the return condition of the suspected component (i.e. onboard or desoldered) with the Infineon Application Engineer or Customer Quality Manager who supports your company.

For further information about component rework on PCB, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

7 References

[1] Infineon: Packages. www.infineon.com\packages.

[2] International Electrotechnical Commission: IEC 60068-2-58. Environmental testing - Part 2-58: Tests - Test Td: Test methods for solderability, resistance to dissolution of metallization and to soldering heat of surface mounting devices (SMD).

[3] Electronic Components Industry Association, Assembly and Joining Processes and JEDEC Solid State Technology Association Committee: EIA/IPC/JEDEC J-STD-002. Solderability Tests for Component Leads, Terminations, Lugs, Terminals and Wires.

[4] JEDEC Solid State Technology Association: IPC/JEDEC J-STD-020. Moisture/Reflow Sensitivity Classification for Nonhermetic Surface Mount Devices.

[5] JEDEC Solid State Technology Association: IPC/JEDEC J-STD-033. Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices.

[6] Association Connecting Electronics Industries: IPC-A-610. Acceptability of Electronic Assemblies.

Revision History

Page or reference	Major changes since the last revision
Title package	New figure of optical sensor package with ball grid array
Section 1 “Package description”	New description of package type; internal construction, Fig.2 with non-overmold type; "sensor glass", Fig. 4 with smaller protective tape;
Section 3 “	3.3."component placement" with hints where nozzle/vacuum should act on protection tape
Entire document	Editorial review.

Trademarks

All referenced product or service names and trademarks are the property of their respective owners.

Edition 2023-03-13

Published by

Infineon Technologies AG

81726 Munich, Germany

© 2023 Infineon Technologies AG.
All Rights Reserved.

Do you have a question about this document?

Email: erratum@infineon.com

Document reference
B179-I1271-V2-7600-EU-EC

IMPORTANT NOTICE

The information contained in this application note is given as a hint for the implementation of the product only and shall in no event be regarded as a description or warranty of a certain functionality, condition or quality of the product. Before implementation of the product, the recipient of this application note must verify any function and other technical information given herein in the real application. 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) with respect to any and all information given in this application note.

The data contained in this document is exclusively intended for technically trained staff. It is the responsibility of customer's technical departments to evaluate the suitability of the product for the intended application and the completeness of the product information given in this document with respect to such application.

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

WARNINGS

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

Except as otherwise explicitly approved by Infineon Technologies in a written document signed by authorized representatives of Infineon Technologies, Infineon Technologies' products may not be used in any applications where a failure of the product or any consequences of the use thereof can reasonably be expected to result in personal injury.