Recommendations for board assembly of Infineon modules with land grid array terminations

About this document

Scope and purpose

This document provides additional information on the surface board mounting of Infineon devices with the package type LG-MLGA-14-1.

Intended audience

This document addresses all users that are handling the pad and stencil design, the board mounting or the rework of the herein discussed Infineon devices.

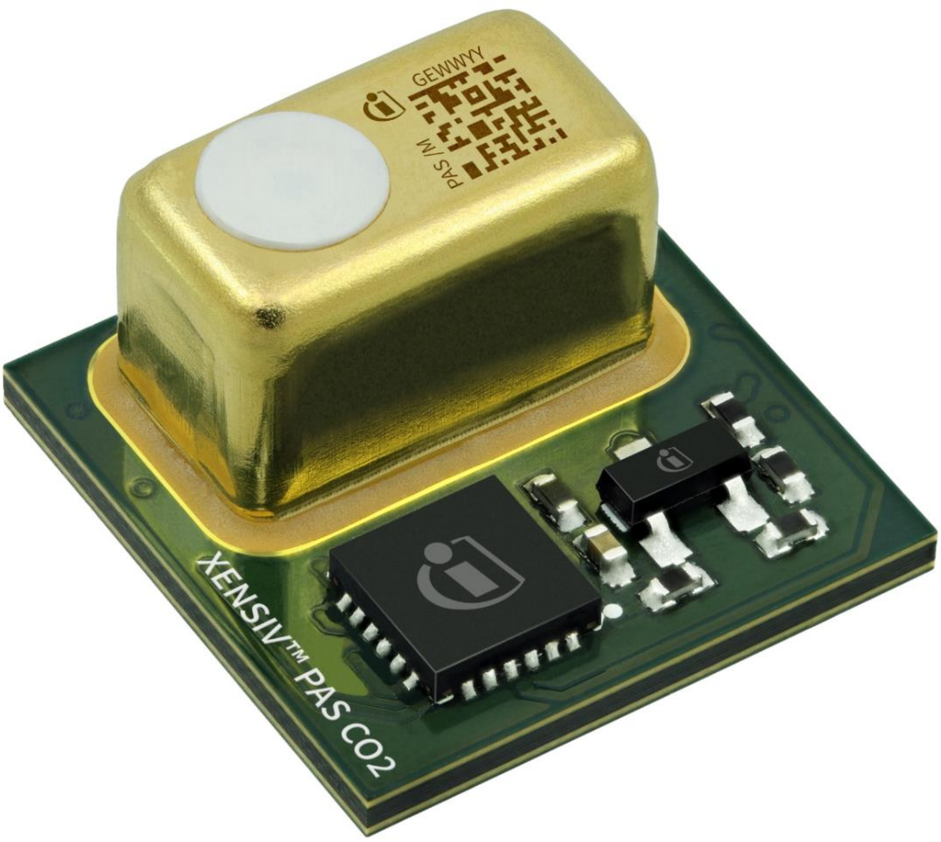

Table of contents

Acronyms and abbreviations

AOI	automated optical inspection
AXI	automated x-ray inspection
ESD	electrostatic discharge
I/O	input/output
IQFN	integrated quad-flat no-lead
LG	laminate green
LGA	land grid array
MC	microcontroller
MEMS	micro-electro-mechanical systems
MLGA	module land grid array
MSL	moisture-sensitivity level
Ni/Au	nickel/gold
NSMD	non-solder mask defined pad
PAS	photo-acoustic spectroscopy
PCB	printed circuit board
SAC	tin silver copper (SnAgCu)
SMD	solder mask defined
SMD	surface-mount device
SOM	system on module
SMT	surface-mount technology

1 Component description

This document provides information about the surface mount technology (SMT) board assembly of Infineon module land grid array (MLGA) components. A system on module (SOM) provides a specific function by system integration on board-level. The concerted devices allow for an efficient circuitry, a compact module design and a cost-effective application. Infineon modules can consist of multiple micro-electro-mechanical system (MEMS), micro-controllers (MC) as well as active and passive devices. An example are systems for indoor air quality monitoring whose working principle are based on photo-acoustic spectroscopy (PAS).

This document does not discuss land grid array (LGA) packages in general or integrated packages such as integrated quad-flat no-lead (IQFN). These package families are described in separate documents.

1.1 MLGA component type

MLGA type components are built on an advanced laminate substrate with multiple layers of circuitry. The terminations are of bottom-only type with no solderable area reaching the substrate outline. This LGA configuration allows for a small footprint. **Figure 1** shows an example of an MLGA type module.

- LG-MLGA

LG = Laminate Green
M = Module
LGA = Land Grid Array

Figure 1 Example of a MLGA module.

1.2 Component features and general handling guidelines

Infineon LGA type modules can be assembled using standard industry pick-and-place equipment and processes. Parts inside of and on the module may consist of sensitive features. Care should be taken to avoid damage to the MEMS devices by following the guidelines outlined below.

ESD 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.

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.

Component handling precautions

- Do not use excessive force to place the component on the base printed circuit board (PCB). The use of standard industry pick-and-place tools is recommended in order to limit the mechanical force exerted on the module components.
- The PAS port hole is protected by a dust membrane that is capable of preventing particle ingress. The contamination during base PCB assembly should, however, be kept as low as possible in order to prevent clogging of the dust membrane.
- There is no wet cleaning such as board washing allowed. This may clog the gas inlet of the sensor port hole.
- If an air-blowing cleaning process is used, do not blow air directly into the sensor port hole.
- The module should not be operated in a harsh environments (e.g. dust, salt) that might deteriorate the functionality of the dust membrane and the inner MEMS devices.
- The sensor port hole below the dust membrane shall not be exposed to vacuum. That includes not to pick up the module at the dust membrane using vacuum tools.

Internal construction

MLGA modules offer a compact outline and a small component footprint. The LGA substrate allows for high complexity inner circuitry by using multiple layers that are inter-connected by vias. The system function is achieved by use of components in various leaded and leadless packages, which are coordinated with each other. The core PAS device is protected underneath a metal lid. The power regulator and logic controller are placed outside the lid on the laminate substrate. **Figure 2** shows a schematic drawing of the inner setup of the Infineon PAS MLGA module.

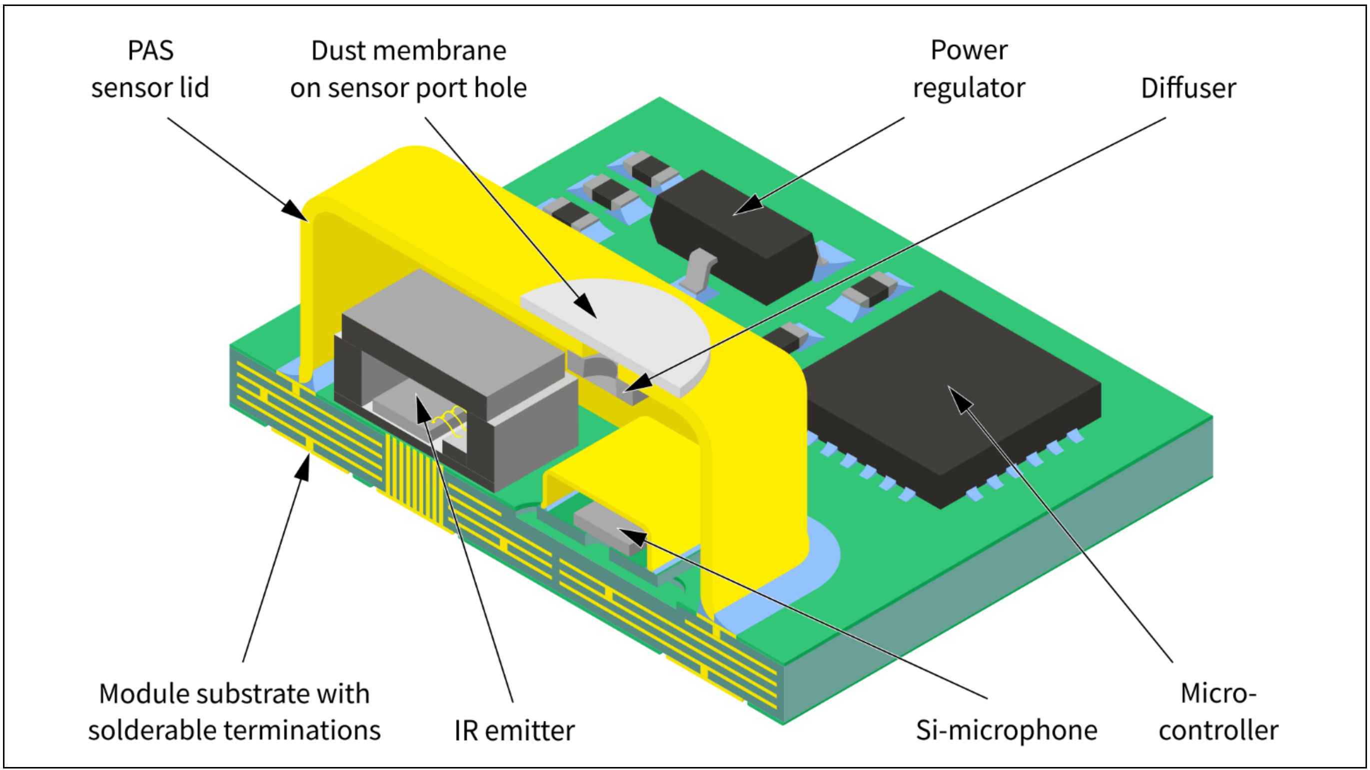

Figure 2 **Schematic showing the inner setup of a PAS MLGA module.**

Termination design

The MLGA modules have terminations with a land grid array configuration. The solder joint is consequently formed at the bottom only. **Figure 3** shows a schematic of the potential solder joint shape. The basically non-solder mask defined (NSMD) termination pads of the MLGA module allow the solder to grip the module pad outline when forming the joint. That can increase the board level reliability.

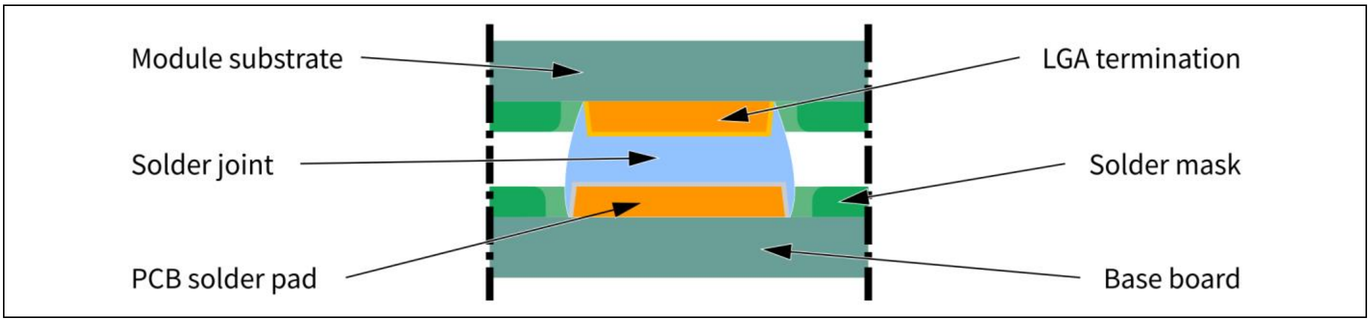

Figure 3 **Schematic showing the termination design of a LGA type module and the potential solder joint shape.**

Termination plating

The final finish of MLGA module terminations consist of nickel/gold (Ni/Au).

2 Printed circuit board

2.1 Routing

The printed circuit board layout and construction are key factors for achieving solder joints with high reliability. Packages with exposed pads should not be placed opposite to each on the PCB top and bottom side when using double-sided nounting. That will stiffen the assembly and cause solder joints to fatigue earlier than in a design in which the components are offset. Furthermore, the board stiffness itself has a significant influence on the reliability of the solder joint interconnect if the system is used in critical temperature-cycling conditions.

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

MLGA modules have a land grid array pad configuration as shown in **Figure 4**. The base PCB pads are typically designed by enlarging the component pad outline circumferentially (i.e. by adding 50 µm). It is also recommended to add a certain tip extension to smaller I/O PCB pads in the direction of the component outline. With such a design a reliable frustum shaped solder joint can form while also considering certain print and alignment tolerances. The NSMD pad registration is recommended based on board assembly tests. The SMD pad registration can be beneficial for packages with large ground pads. It allows for large conductor areas below the solder mask. However, the offset and usually higher tolerances of the solder mask compared with the copper must be taken into account. Generally, mixing both solder mask registration types in one footprint pad design is not recommended.

The modules may feature open copper areas on the landing area that are not intended for soldering. Such areas should be kept free from open metal structures on the PCB by defining “keep-out areas”.

An optimal base PCB design generally depends on the specific application as well as on the specific design guidelines of the chosen board manufacturer.

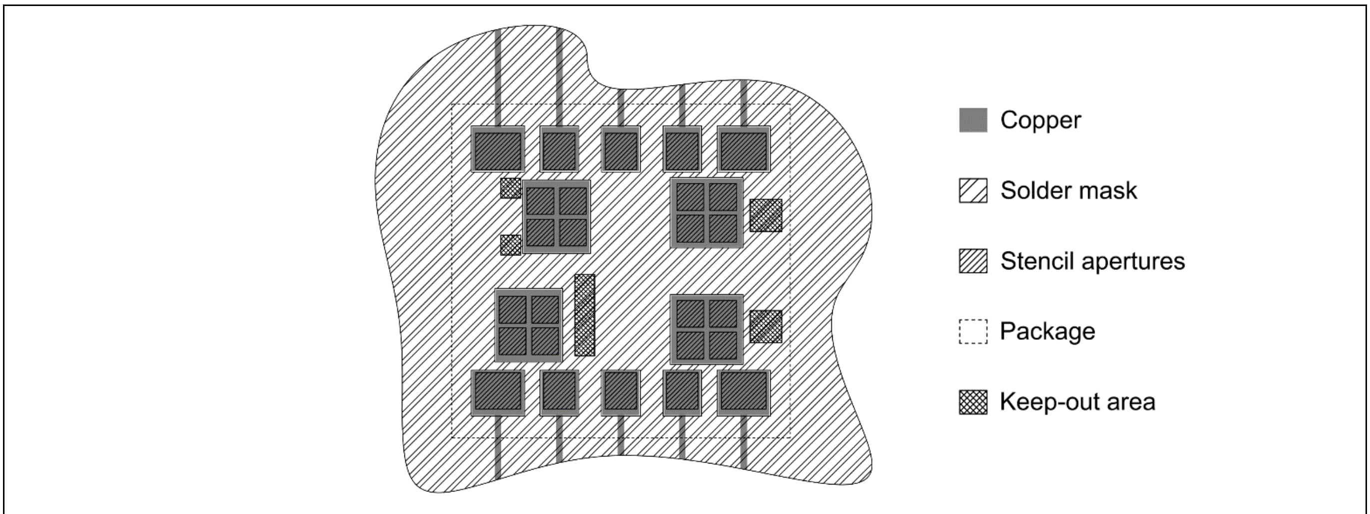

Figure 4 Example of a MLGA module pad and stencil design with NSMD pads.

Details on package specific base PCB pad 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 database, which will then show an example of the footprint layout for each package.

For further information about base PCB pad 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 base PCB landing pads by stencil printing. The volume of the printed solder paste is determined by the stencil aperture and the stencil thickness. While an excessive solder paste volume will cause solder bridging, an insufficient solder paste volume can lead to reduced solder spreading between all contact surfaces. To ensure a uniform and sufficiently high solder paste transfer to the base PCB, laser-cut (mostly made from stainless steel) or electroformed stencils (made from nickel) are preferred. The latter are used especially for fine-pitch component assembly.

The terminations of LGA type modules are of bottom-only style with no solderable area reaching the substrate outline. When applying a PCB pad design approach as shown in **Figure 4** a print reduction difference by area between larger center and smaller peripheral I/O package pads of approx. 10 to 15 percentage points is recommended. The thickness of a stencil usually has to be matched to the needs of all components on the base PCB. For typical module applications, a stencil thickness of 130 µm (5 mil) is recommended.

For individual design adaptations to reach the optimum amount of solder, the stencil thickness, the base 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.

Further details and 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 component when searching the data base, which will then show an example of the stencil aperture layout.

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% to 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 4 paste or higher (with lower grain size of the solder alloy powder) is recommended for the assembly of LGA type modules, depending on the specific stencil aperture size and therefore solder paste transfer efficiency.

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 contaminants and oxides on the surface.

The solder paste solvents have to evaporate during reflow soldering, while residues of the flux will remain on the 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 leadless packages in which the solder joint is formed mainly on the package bottom side, a “no-clean” paste is recommended to avoid subsequent cleaning steps underneath the package. The small gaps make cleaning highly difficult if not impossible.

Note: *A “no-clean” solder paste is recommended for the assembly of the LGA type modules to avoid subsequent cleaning steps.*

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

3.3 Component placement

The use of standard industry pick-and-place equipment is recommended in order to limit the mechanical force applied to the package as well as to allow for accurate placement. Such machinery typically uses vacuum tools to pick the components. Placing the packages manually is not recommended, especially not for packages with small terminations and pitch. Component placement accuracies of +/-50 µm and less are obtained with modern automatic machinery using vision systems. With such systems, both the base PCB and the components are optically measured and the components are placed on the base PCB at their programmed positions. The fiducials on the base PCB are located either on the edge of the whole base PCB, or at additional individual mounting positions (local fiducials).

The sensor port hole below the dust membrane shall not be exposed to vacuum. That includes not to pick up the module at the dust membrane using vacuum tools. In case of PAS MLGA modules, the flat surfaces of the sensor lid or the top of the microcontroller should be used for placing the nozzle of the pick-up tool as shown in **Figure 5**.

The off-centered nozzle placement position requires a fixed pick-up position when programming the component placement. Auto-correction routines may lead to misplaced components.

Note: The PAS MLGA module must not be picked up at the dust membrane using vacuum tools.

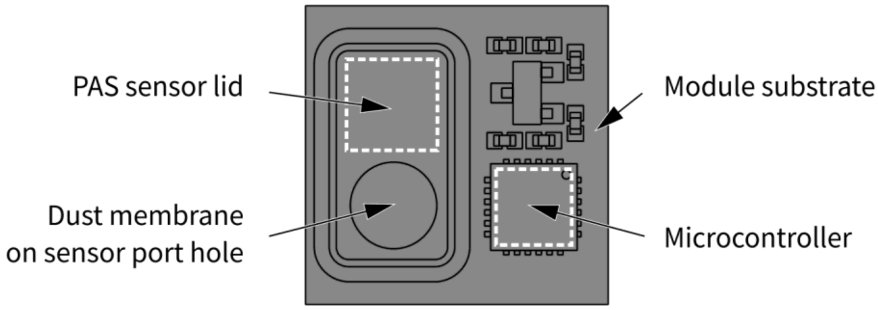

Figure 5 The preferred areas for nozzle picking are the top of the microcontroller or the top of the sensor lid (marked by white outline). The nozzle shall not seal with the dust membrane of the sensor port hole.

The ratio between the small area for nozzle placement of the pick-up tool and the module outline and mass may requires a reduction of the pick-up head movement speed.

The use of mechanical gripper tools instead of vacuum nozzles and pipettes may be a practical option for component placement. These tools feature precision machined, vacuum-actuated jaws for securely gripping a variety of parts and shapes. The grippers easily pick components from several input types, such as tape, trays and other feeding solutions. Please contact your placement machine vendor for further information.

For further information about factors influencing 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

The widely used method of reflow soldering in a forced convection oven is recommended for the base PCB assembly of LGA type modules. Soldering in a nitrogen atmosphere can generally improve the solder joint quality but is not necessary to create a reliable joint. Vapor phase and vacuum soldering can damage the MEMS devices and shall therefore not be used for solder reflow.

Note: *Do not use vapour phase or vacuum reflow processes for LGA type module soldering, as they can damage the MEMS devices.*

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. It is recommended to optimize the reflow profile in such a way that excessive flux or solder spattering is avoided.

Maximum reflow conditions and cycles

Modules that are moisture-sensitivity level (MSL) classified by Infineon have been tested in accordance with the J-STD-020 standard. The maximum allowed reflow profile must not be exceeded during board assembly. The number of reflow cycles is restricted to one. 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 LG-MLGA-14-1 module product is limited to a maximum reflow temperature of 245°C with a maximum of one reflow cycle. Please also refer to the relevant product datasheet.*

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 MEMS devices on the PAS MLGA module are sensitive to mechanical impact, and to particle or fluid contamination. The sensor port hole is protected by a dust membrane that is capable of preventing particle ingress. The contamination during base PCB assembly should, however, be kept as low as possible in order to prevent clogging of the dust membrane. The introduction of wet cleaning steps into the mounting process shall be prevented; e.g. by using a “no-clean” solder paste flux during board assembly. If cleaning with pressured air or nitrogen is used, blowing the gasses directly into the sensor port hole must be avoided.

Note: It is not allowed to apply wet cleaning such as board washing. This may clog the gas inlet of the sensor port.

For further information about the special cleaning precautions for MLGA modules, please contact your local sales, application, or quality engineer.

5 Inspection

5.1 Optical solder joint inspection

The LGA type modules feature bottom-only terminations. The solder joints are formed underneath the module. A visual inspection of the solder joints with conventional automatic optical inspection (AOI) systems is not possbile. **Figure 6** shows a photograph of a MLGA module after properly mounted to a base PCB. Even a side-view can only reveal a certain number of the solder joints.

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.

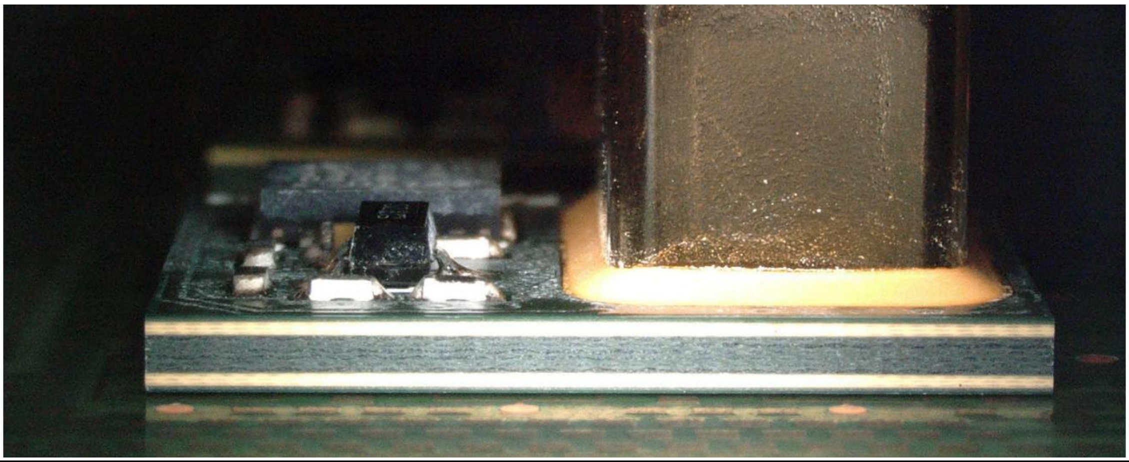

Figure 6 Side view of a mounted MLGA module. The solder joints below the module substrate cannot be seen.

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

5.2 X-ray solder joint inspection

Automated x-ray inspection (AXI) systems are appropriate for efficient inline control of components such as LGA type modules whose terminations cannot be inspected properly by optical systems. AXI 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.

Figure 7 shows an x-ray image of a properly soldered MLGA module. The base PCB solder joints as well as the module components are visible. When choosing a specific x-ray set up the contrast of the target solder joints might be enhanced against interfering structures. It can also be beneficial to use an angular view to reveal the base PCB solder joints that are covered by the solder joints on the substrate top side.

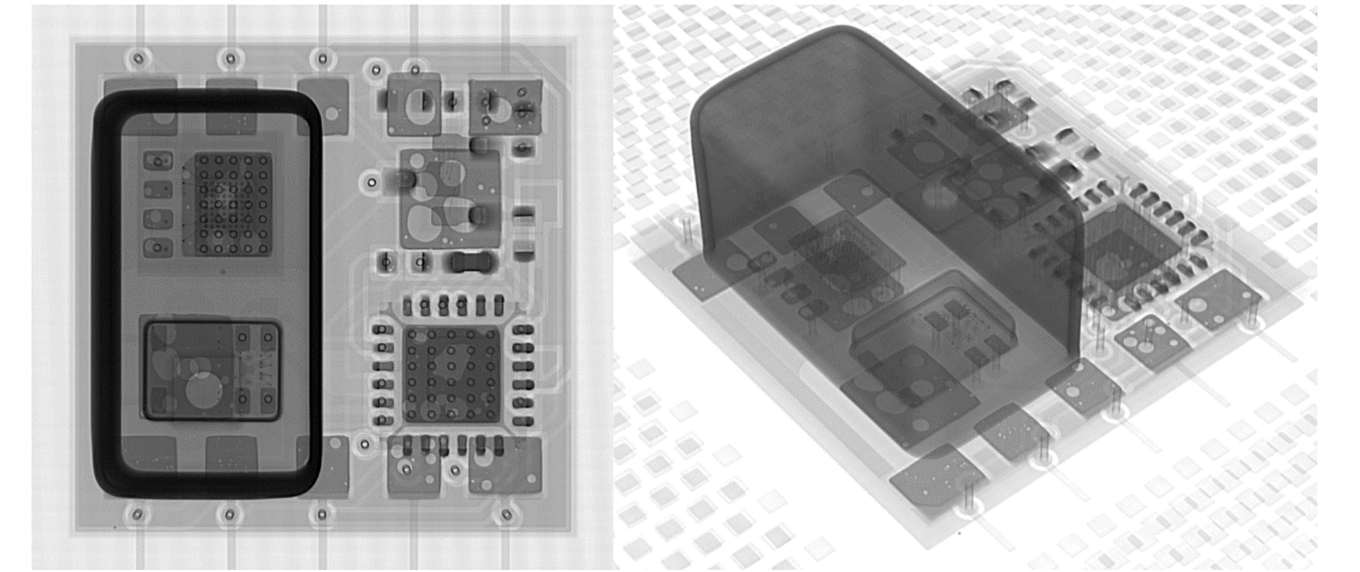

Figure 7 Top-view (left) and angular (right) x-ray image of a soldered MLGA module. The module-to-base board solder joints as well as the internal setup of the module assembly are visible.

6 Rework

Single solder joint repair of modules with land grid array configuration is not impossible, and is therefore generally not recommended. Furthermore, the reuse of de-soldered components is not possible. The de-soldered components must be replaced by new ones.

If a component is suspected to be defective and a failure analysis is planned, Infineon usually expects customers to de-solder the component prior to return to Infineon. The entire module shall be returned in a proper condition according to the original component outlines.

In some special cases such as solder joint inspection Infineon may request that the base PCB or part of the base PCB with the module 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 base 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. https://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
Section 3.4 “Reflow soldering”	Reduction of maximum allowed reflow cycles for board assembly.

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

Edition 2022-01-19

Published by

Infineon Technologies AG

81726 Munich, Germany

Do you have a question about this document?

Email: erratum@infineon.com

Document reference