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

About this document

Scope and purpose

This document provides additional information on the board mounting of Infineon devices with the package type LG-MLGA which are module devices for DC-DC-power applications in consumer applications (e.g. LG-MLGA-45-1/-3 and LG-MLGA-72-5/-6). Further packages may also be covered depending on their link to this product category. Please consider that other LG-MLGA may be covered by board assembly recommendations for PAS modules (e.g. LG-MLGA-14-1) or land grid array packages. Other module type packages and their board assembly aspects are described in further documents. Please be sure that the recommendation given fits to your given product/package.

The recommendations cover the board assembly and adjacent processes. Reliability and qualification topics are handled in further documents.

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.

For Q&A you can use also Infineon's community pages: https://community.infineon.com



Table of contents

About this document **1**
Table of contents **2**
Acronyms and abbreviations **3**
1 Component description **4**
1.1 MLGA component type 4
1.2 Component features and general handling guidelines 5
2 Printed circuit board **8**
2.1 Routing 8
2.2 Pad design 8
3 PCB assembly **10**
3.1 Solder paste stencil 10
3.2 Solder paste 10
3.3 Component placement 11
3.4 Reflow soldering 12
4 Cleaning **15**
5 Inspection **16**
5.1 Optical solder joint inspection 16
5.2 X-ray solder joint inspection 17
6 Subsequent processes **19**
6.1 Processes after reflow containing additional thermal treatment like burn-in, conformal coating, underfilling 19
6.2 Heat sink mounting 19
7 Rework **21**
References **22**

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
PAS	photo-acoustic spectroscopy
PCB	printed circuit board
SAC	tin silver copper (SnAgCu)
SMD	solder mask defined
SMD	surface-mount device
SMT	surface-mount technology
SOM	system on module

1 Component description

This document provides information about the surface-mount technology (SMT) board assembly of Infineon module land grid array (MLGA) components for DC-DC-power modules. A system on module (SOM) provides a specific function by system integration on board-level. The concerted devices allow for an efficient circuitry and power distribution, a compact module design and integrated inductor coils. The module-internal interconnects consist of Pb-free solder. No further materials such as glue or other adhesives are used to hold the components in place on the substrate.

This document does not discuss land grid array (LGA) packages in general or integrated packages such as integrated quad-flat no-lead (IQFN), nor PAS modules. 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. Different types of integration are used to serve the application needs. Infineon power modules can consist of multiple chip-embedded (CE) power stages and passive components mounted on a substrate (Figure 1). Other types use chip-embedded power stages as substrate and therefore don’t need an extra substrate as carrier (Figure 2). The terminations are of bottom-only type with no solderable area reaching the substrate outline. This LGA configuration allows for a small footprint.

- LG-MLGA

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

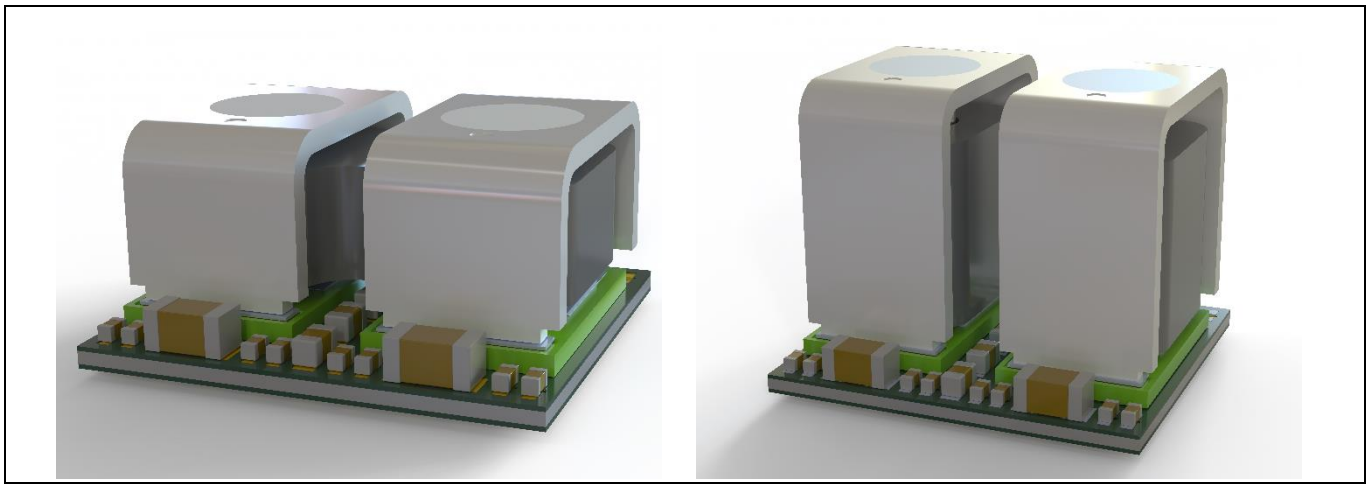


Figure 1 **MLGA power module using a substrate as carrier, with chip-embedded power stages, passive components, and inductors mounted on the carrier.**

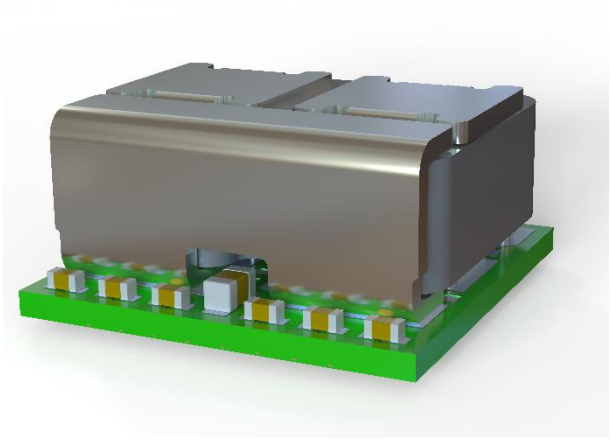


Figure 2 **MLGA power module using no extra substrate as carrier, but the combined chip-embedded power stages serve as substrate to mount passive components and inductors.**

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 damaging the device by following the guidelines outlined below.

ESD precautions

- Despite the potential presence of protection circuitry, devices exposed to high-energy electrostatic discharge (ESD) may be damaged. Since charged devices and circuit boards can discharge without detection, proper 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

- Moisture sensitivity level (MSL) ratings provide an indication of the floor life of the module after it is removed from its packaging. Please refer to section 3.4.
- Do not handle modules with bare hands, as the plating will tarnish within hours after touch. Once oxidized solder will not properly flow on the pads. For details about the plating please refer to the next page.
- 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 (e.g. with force limitation feature and made of materials that do not damage the module's surface) is recommended to limit the mechanical force exerted on the module components.

- The topology of MLGA power modules allows pick-up by a vacuum nozzle that acts on the top area or part of it. A special nozzle design may be of advantage (please refer to section 3.3). A pick-up by using a gripper is also possible.
- Due to the topology of MLGA power modules, cleaning, if required, must be done very thoroughly (please refer to section 4).

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, which are coordinated with each other. The core power semiconductor devices are of chip-embedded type, and the compact inductor design and surrounding circuitry result in the lowest losses. Figure 3 shows schematic drawings of the inner setup of the Infineon MLGA DC-DC-power modules.

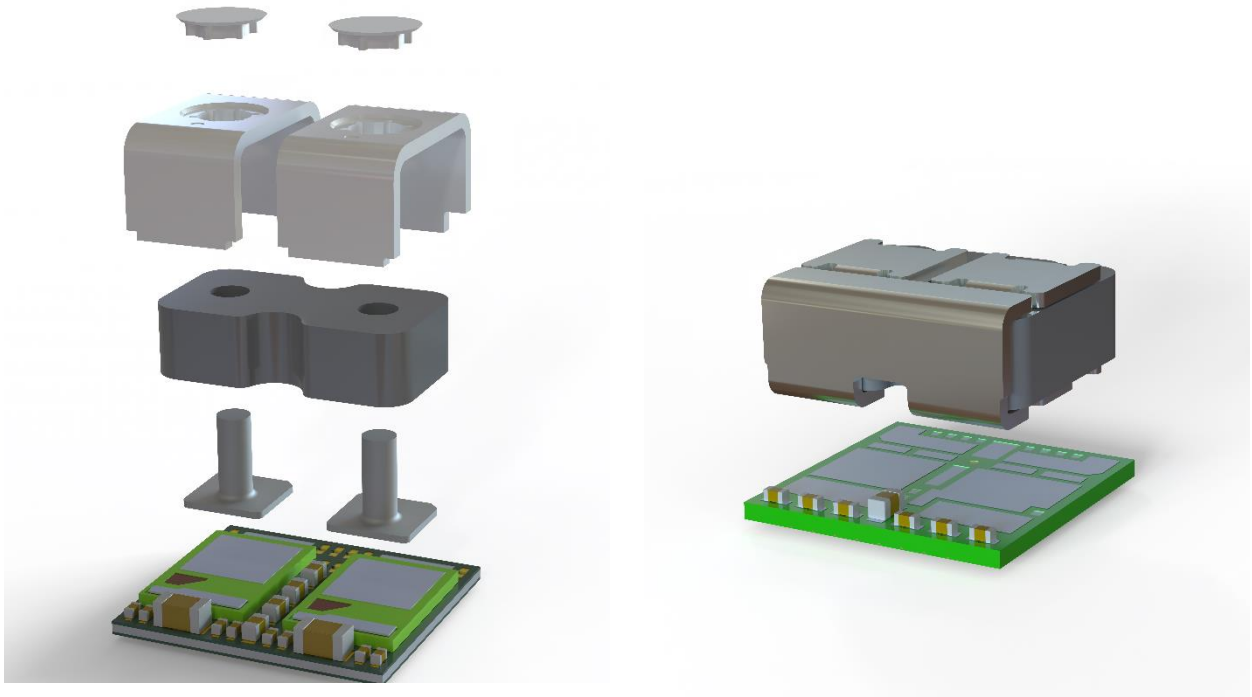


Figure 3 **Schematic showing the inner setup of a LG-MLGA-72 (inductor assembled out of single parts) and LG-MLGA-45 (inductor assembled as block).**

Termination design

The MLGA modules have terminations with a land grid array configuration. The solder joint is consequently formed at the bottom only. Figure 4 shows a schematic of the potential solder joint shape. Generally, non-solder mask defined (NSMD) termination pads are used for MLGA modules wherever possible and allow the solder to grip the module pad outline when forming the joint. Electrical and thermal requirements may make it necessary to have solder mask defined edges, which is not a disadvantage in terms of wettable surfaces deviation, since substrate tolerances (position of solder mask relative to copper layer) are very small and can be controlled in a much better way than for circuit board assemblies.

Termination plating

The final finish of MLGA module terminations consists of nickel/gold (Ni/Au) also ENIG (electroless nickel immersion gold) or so-called ENIPIG (electroless nickel immersion palladium immersion gold). The package's solderability is tested according to IEC 60068-2-58 [2].

Please ensure to keep the modules in a clean environment (e.g. in a vacuum-sealed bag or a nitrogen cabinet) prior to manufacturing assembly to minimize the potential for contamination especially of the termination plating.

Cleaning of contaminated modules is not recommended since contaminations may be transferred to areas that are not easily accessible, or terminations may be contaminated in a way that compromises the solderability but is not visible after cleaning.

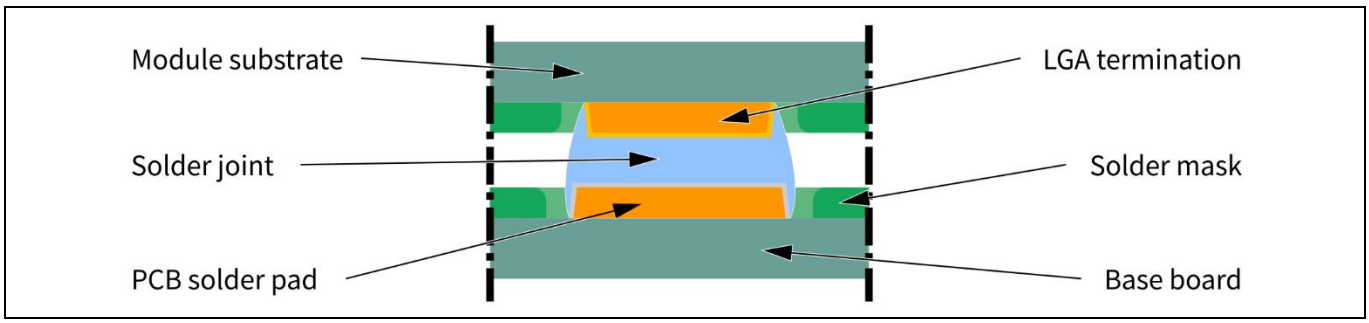


Figure 4 **Schematic showing the termination design of an LGA type module and the potential solder joint shape.**

2 Printed circuit board

2.1 Routing

The printed circuit board layout and construction are key factors for achieving solder joints with high reliability.

MLGA power modules are clustered according to their general functionality. The footprint for each cluster or product family is kept the same despite the differences in electrical functions and technology on the topside of the substrate. For instance, the simple design of 72 square-shaped package pads allows an easy and reliable board assembly process. Having the same clustering of several of these squares for the same I/O for every product of the LG-MLGA-72 family (TDM225x) makes PCB layout easy. Other product families of MLGA style (for instance LG-MLGA-45) are also following these general principles.

When using double-side mounting, packages that are relatively large and have many or large pads should generally not be placed opposite each other on the top and bottom side of the PCB. It stiffens the assembly and causes solder joints to fatigue earlier than in a design where the components are offset. This must be considered when using an opposing placement of large components to optimize the electrical features of the whole assembly. 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, which is typically not the case e.g. for server applications that demand power optimization features prior to system-level reliability.

2.2 Pad design

The quality and reliability of the 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 which allows simple circuitry for power applications and is shown in the specific product datasheets. The base PCB pads are typically designed by enlarging the component pad outline circumferentially (i.e. by adding 50 µm). Since the given packages for Infineon's DC-DC-power modules have relatively large pads (meaning they allow to use stencil apertures which are minimum 1.2 times bigger than the smallest aperture defined by the area ratio rule for stencils used in the specific application) there is no need in most applications to enlarge the small pads further. With such a design a reliable shaped solder joint can form while also considering certain print and alignment tolerances.

The NSMD pad registration is recommended to aim at highest solder joint reliability. The SMD pad registration can be beneficial for having the best thermal performance. It allows for large conductor areas below the solder mask. However, the offset and usually higher tolerances of the solder mask compared to the copper must be considered. Whatever the trade-off of the given application Infineon is strictly recommending to not use a mixing of both solder mask registration types in one footprint pad design which would lead to a high deviation of wettable surfaces on the board and therefore non-uniformly shaped solder joints.

Printed circuit board

Please keep in mind that an optimal base PCB design generally depends on the specific application as well as on the specific design guidelines of the chosen board manufacturer.

Generic footprints are available on the Infineon website: www.infineon.com/packages under the Module section and the specific entry for LG-MLGA (e.g. LG-MLGA-72-5).

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

Typically 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 given in Infineon's generic footprint, a print reduction difference by area between larger center and smaller peripheral I/O package pads (if applicable) of approx. 10 to 15 percentage points is recommended, depending on the size of the wettable surfaces and the stencil thickness. The latter usually must 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. The need for a certain solder joint thickness and stand-off to avoid direct contact between the bottom solder mask layer of the module and the top solder mask layer of the PCB, which can result in insufficient solder joints and reliability issues, must be considered.

Individual design adaptations to achieve the optimum amount of solder should consider, stencil thickness, base PCB pad finish, quality and solder masking, via layout, and the solder paste type. In all cases, application-specific experiments are recommended.

Further details and specific stencil aperture recommendations can be found in the package database available on the Infineon web page [1]. Please choose a specific component when searching the database, this will provide an example of the stencil aperture layout.

For further information about solder stencil design, 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

Generally, the use of standard industry pick-and-place equipment is recommended to limit the mechanical force applied to the package and 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).

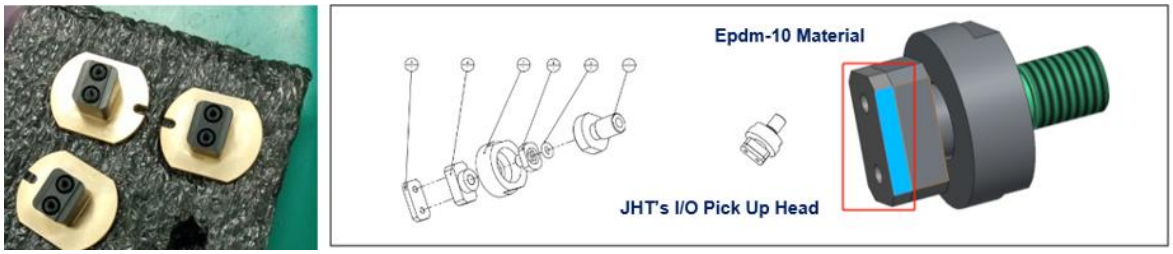
MLGA type modules have a weight which may cause a shift of components during movement on the nozzle. Therefore, it is recommended to test the maximum suitable acceleration.

MLGA type modules have a high topography since no mold compound used typically for semiconductor packages is shaping and levelling the topside. Therefore, in this case, the nozzles or tools used for the mounting equipment must be chosen with care, special tools may have to be designed and adapted by the equipment manufacturer or a tooling provider. Generally, three different types of tools and process can be used:

- A standard vacuum nozzle (with an off-centered nozzle position) may be the most straightforward approach, however, this method carries the highest risk of component loss.
- A special vacuum nozzle acting on the top of the highest portion of the power module – typically these are the flat top areas of the inductor.
- A gripper that should grab approximately half the height of the inductor to prevent mechanical damage to other components soldered on the substrate.

Figure 5 shows typical pick-up nozzles designed and fabricated by two representative equipment manufacturers.

Pick-up nozzle Type A:



Pick-up nozzle Type B:

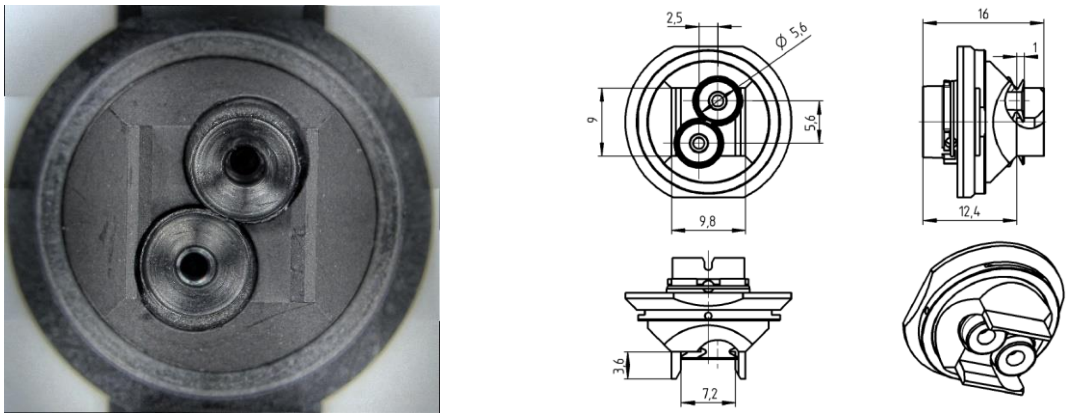


Figure 5 Two different types of pick-up nozzles.

Please contact your placement machine vendor for further information.

Be aware that the use of special nozzles or grippers implies that machine parameters (e.g. pick-up speed) must be adapted to these tools. Also, the specific component packing (tray or tape), feeder, specific mounter head, etc., must be considered when using special nozzles or grippers.

Also note that the ratio between the small nozzle placement area of the pick-up tool and the module outline and mass may require a reduction of motion speeds. Typically, the mass of Infineon's modules is below 3 grams, e.g. LG-MLGA-72 modules range between 1.5 and 2.5 grams, LG-MLGA-45 between 1.0 and 1.5 grams depending on the specific inductor type.

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 circuit board 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.

When using vapor phase or vacuum soldering, be aware that high temperature gradients or sudden pressure changes can affect the integrity of the module assembly. Temperature induced failures may be cracks and

delamination. Pressure induced failures may be mechanical displacement of devices or the inductor and, in the worst case, open solder joints or malfunction.

Due to the relatively heavy weight of the modules described, it may not be beneficial to introduce mechanical shock or vibration. This can result in a shifted module or shifted components on the module.

When a double-sided assembly is intended and such a module is placed upside down in the second reflow, the weights of the components on the module and the weight of the module compared to the individual pad areas must be considered, resulting in certain surface tensions that add up to an adhesive force holding the component in place. As a rule of thumb surface tension of the solder can be compared to the force of gravity to determine if the given pad areas are large enough to hold the component in place. Note that the specific reflow oven used for soldering may introduce vibration or shock to the board, which can cause the module to fall off the PCB even if the surface tension is greater than gravity.

Infineon performed tests and simulations have proven that Infineon's DC-DC-power modules and the components on it are light enough to be held by surface tension of molten solder when using the recommended PCB footprint and introducing less than 0.5 g as maximum vibration in the reflow process.

The use of special holders or boats during reflow which hold the component in place (e.g. by using springs pressing the module and parts on it against the PCB) during 2nd reflow is not recommended. Applying any force beyond the module/component weight may lead to a severe change of the solder joint shape. In worst case resulting in solder bridges between pads of the module (soldered to the customer's PCB) or at solder joints of components mounted on the module (which are also pressed down by the fixture).

It must be considered that holders may warp in a different way than the PCB during reflow soldering which may cause additional forces on the module.

The soldering profile should follow the solder paste manufacturer's recommendations to achieve optimal solder joint quality. The position of the component and the surrounding area 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.

Consider that components as part of the module are soldered using solder paste containing SAC305 alloy and no-clean flux. Therefore, their solder joints will re-melt when the module is assembled on the PCB and flux residues may liquefy. If the temperature gradients are too steep, these flux residues can heat up so quickly that they can explosively spread to areas of the board adjacent to the module.

Maxima 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.

Note: The reflow profile must be measured on top of the component – in the end the hottest position.

The number of reflow cycles is typically restricted to three according to the JEDEC standard. 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].

For details about the different influencing factors to achieve the best reflow profile please refer to technical report given by IEC TR 60068-3-12 [6].

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

Because MLGA modules are not protected by a mold compound, for example, they are sensitive to mechanical impact, and to particle or fluid contamination. Contamination should be minimized during base PCB assembly prevent electrical malfunction or corrosion.

The use of wet cleaning as part of the assembly process should be avoided, for example, by using a “no-clean” flux during board mounting.

If wet cleaning, such as board washing, is inevitable for other reasons, the customer or electronic manufacturer’s service must ensure that the cleaning is performed in such a way that the washing process does not introduce contaminations to the module. This can happen by displacing (corrosive) particles/substances or by changing the chemical conditions of e.g. flux residues by the cleaning medium.

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

LGA-type modules feature bottom-only terminations. The solder joints are formed underneath the module. Therefore, a visual inspection of the solder joints with conventional manual or automatic optical inspection (AOI) systems is not possible. Figure 6 shows a photograph of a MLGA module properly mounted to a base PCB. Even a side-view can reveal only a certain number of solder joints. Therefore, the criteria of IPC-A-610 [7] can be hard to check.

For engineering tasks, cross-sectioning can offer detailed information about the solder joint quality. Analyses like x-ray or laminography can be used to determine the cross-sectioning plane. Due to its destructive character, cross-sectioning during monitoring is naturally not practical.

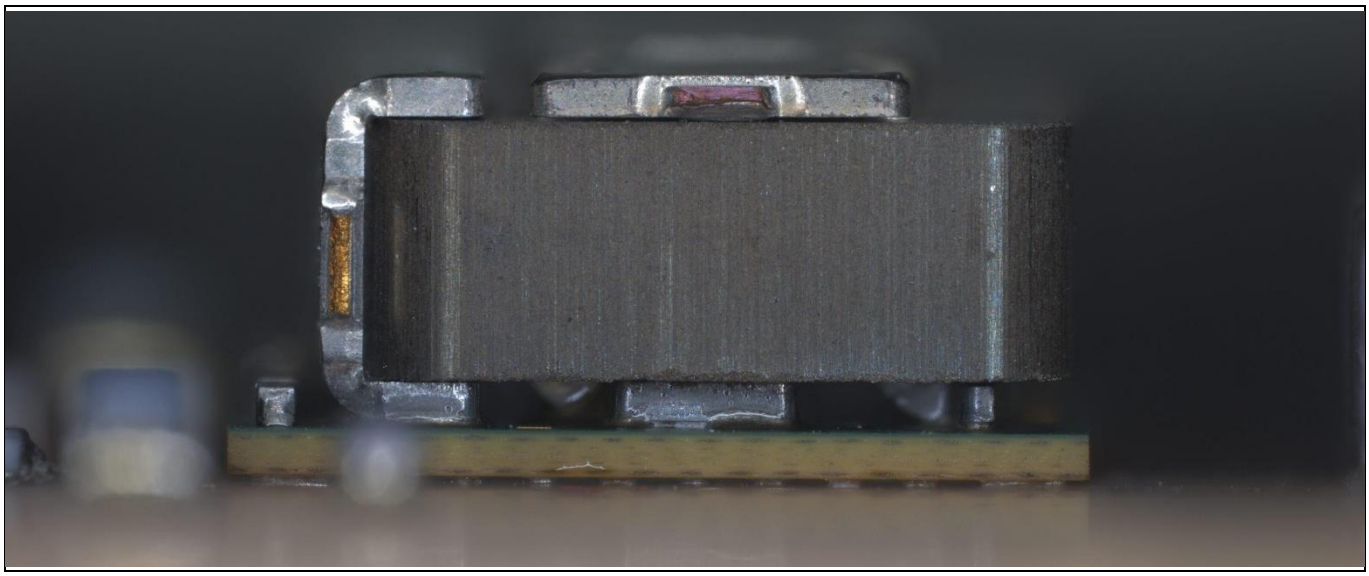


Figure 6 **Side view of a mounted MLGA module. The solder joints below the module substrate cannot be inspected.**

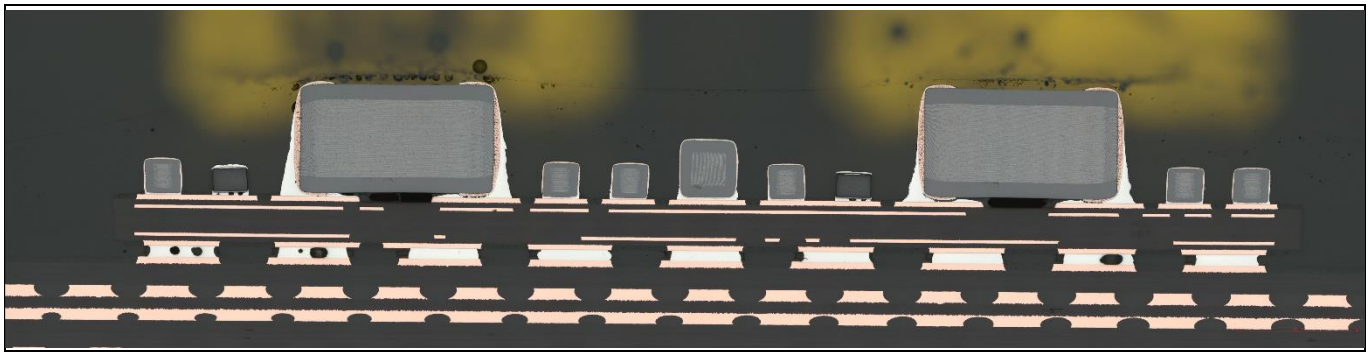


Figure 7 **Cross section through a mounted MLGA module.**

For the acceptability of electronic assemblies, please also refer to the IPC-A-610 standard [5].

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. Since the DC-DC-power modules consist of several layers, that make the inspection in 2D difficult, we recommend 3D (so-called laminography or Planar CT). These reliable systems enable the user to detect soldering defects such as poor soldering, bridging, or voiding. However, other defects, such as broken solder joints, are not as easily detected by x-ray.

Figure 8 shows 2D x-ray images 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.

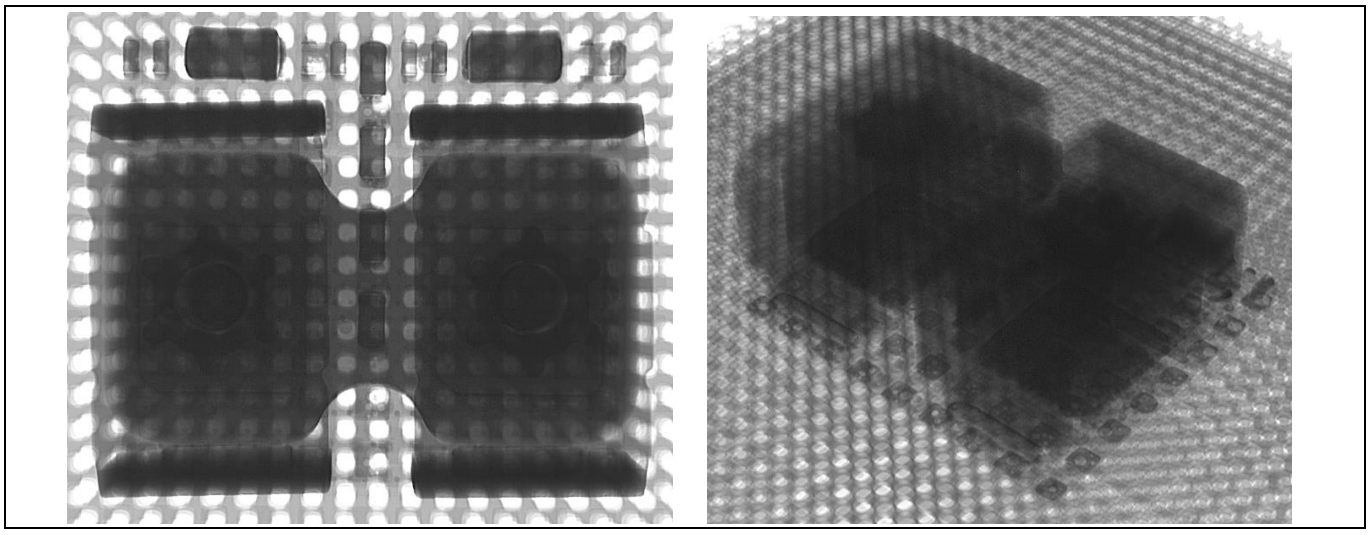


Figure 8 **Top-view (left) and angular (right) x-ray image of a soldered MLGA module. The module-to-base board solder joints as well as the internal setup of the module assembly are visible, but the overlapping structures make a reliable inspection of all solder joints difficult.**

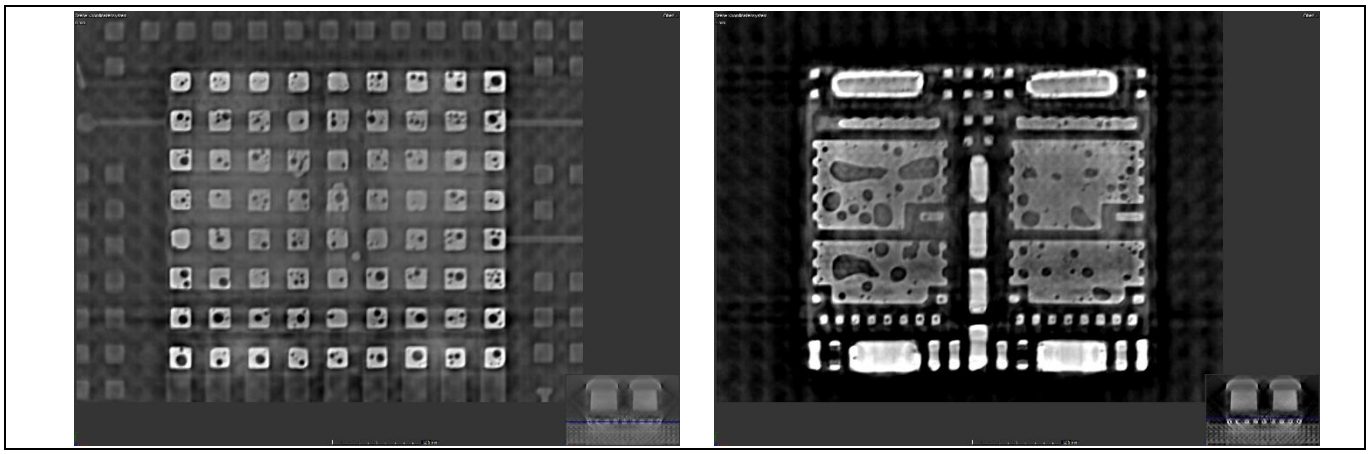


Figure 9 Laminography (also Planar CT) images of a soldered MLGA module. The solder joints in the left image (module-to-base PCB) show voids typical for LGA terminals, as well as solder beads caused by solder paste splatter during reflow. The right image shows solder joints on the topside of the module's substrate (module-internal solder joints).

6 Subsequent processes

6.1 Processes after reflow containing additional thermal treatment like burn-in, conformal coating, underfilling

Additional thermal treatments which do not exceed a maximum temperature of 200°C are not considered as a typical reflow cycling but contribute to the overall thermal budget in the lifetime of the component.

Short exposure times as necessary for curing processes (for instance for conformal coating or underfill) are in that way very uncritical. For such processes the applied material and its interaction with component and PCB is an additional factor which must be considered in terms of reliability.

Long exposure times as used in certain burn-in procedures are on the other hand not neglectable and have beside the thermal budget also influence on reliability.

6.2 Heat sink mounting

The module is designed to efficiently conduct the heat away from the silicon dies inside the power stages, both through the top of the module via the copper clips and through the bottom side of the module via the PCB. Both paths have their advantages. If a heat sink needs to be applied, the following aspects must be considered to avoid mechanical damage and reliability issues.
As shown in the figure below, the thermal energy can be dissipated from the module by attaching a heat sink or a cold plate to the top of the module. A thermal interface material (TIM, thermal gap pad or thermal putty, applied as a paste or as a sheet) between the module and the heat sink/cold plate helps to improve the thermal path by reducing the interfacial contact resistances. The TIM datasheets and application notes recommend a certain mounting pressure for lower thermal resistance. Infineon recommends a mounting pressure less than 100 psi. For the modules described in this document, the pressure is applied to the tops of the two conductors, an area of 28 mm^2, and results in a force of 19 N. Exceeding this mounting pressure may damage the module and cause parametric shifts in performance. Also note that a higher force does not necessarily reduce the thermal resistance by a significant amount. The potential for mechanical and electrical damage and long-term effects on the solder joint (reliability) must be considered. If mechanical force is further applied after heat sink mounting, any temperature increases near (e.g. burn-in above 200°C; assuming SAC305 as solder alloy) or above the solder alloy liquidus temperature (e.g. additional reflow soldering) must be avoided. Otherwise, the solder joints of the affected components will be damaged, solder may be squeezed out and decrease creepage distances or, in the worst case, lead to bridging and electrical shorts.

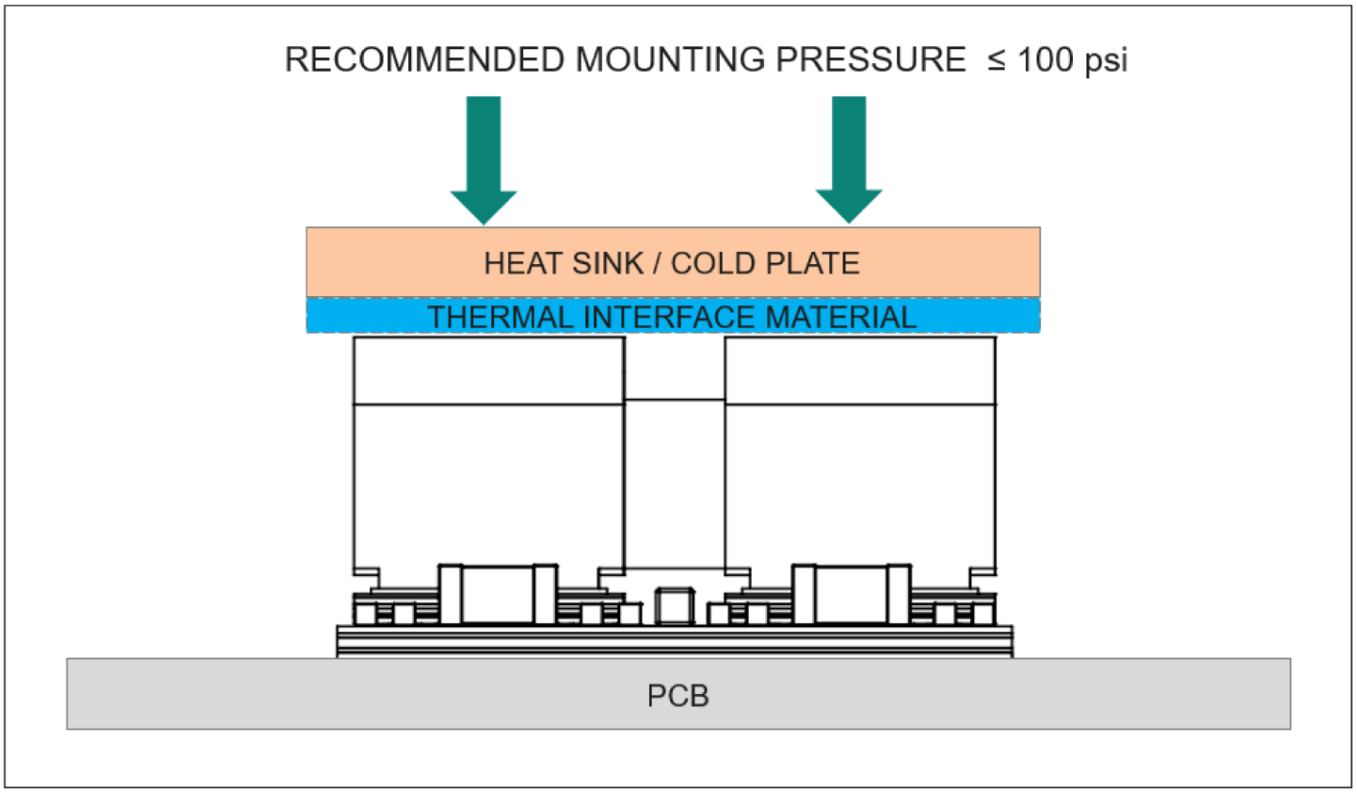


Figure 10 Schematic of a common heat sink for in this case two DC-DC-power modules.

7 Rework

Please avoid damaging the module and its solder joints mechanically or thermally by disassembling, for instance a heatsink.

Single solder joint repair of modules with land grid array configuration is rather impossible and is therefore generally not recommended.

Furthermore, the reuse of desoldered components is not possible. The de-soldered components must be replaced by new ones. Please consider that the desoldering process liquifies not only the solder joints between the module and the base board, but also the module-internal solder joints.

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. Since this de-soldering process may harm the module very easily, an analysis of potential failures may be jeopardized. If a return of the (entire) module in a proper condition can't be guaranteed according to the original component outlines, Infineon requires that no desoldering is done. Best is then to send the whole assembly, or if not possible a thoroughly cut out portion of it to our attention. Especially if a solder joint inspection must be done by Infineon a non-desoldered module is essential.

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.

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] International Electrotechnical Commission: IEC TR 60068-3-12. Environmental testing - Part 3-12: Supporting documentation and guidance - Method to evaluate a possible lead-free solder reflow temperature profile.

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

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

Edition 2026-02-05

Published by

Infineon Technologies AG

81726 Munich, Germany

© 2026 Infineon Technologies AG. All Rights Reserved.

Do you have a question about this document?

Email: erratum@infineon.com

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.

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.