

Recommendations for Printed Circuit Board Assembly of Infineon Laminate Packages

Additional Information

DS1 2012-03

Edition 2012-03

**Published by
Infineon Technologies AG
81726 Munich, Germany**

**© 2013 Infineon Technologies AG
All Rights Reserved.**

LEGAL DISCLAIMER

THE INFORMATION GIVEN IN THIS APPLICATION NOTE IS GIVEN AS A HINT FOR THE IMPLEMENTATION OF THE INFINEON TECHNOLOGIES COMPONENT ONLY AND SHALL NOT BE REGARDED AS ANY DESCRIPTION OR WARRANTY OF A CERTAIN FUNCTIONALITY, CONDITION OR QUALITY OF THE INFINEON TECHNOLOGIES COMPONENT. THE RECIPIENT OF THIS APPLICATION NOTE MUST VERIFY ANY FUNCTION DESCRIBED 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.

Information

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

Warnings

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

Infineon Technologies components may be used in life-support devices or systems only with the express written approval of Infineon Technologies, if a failure of such components can reasonably be expected to cause the failure of that life-support device or system or to affect the safety or effectiveness of that device or system. Life support devices or systems are intended to be implanted in the human body or to support and/or maintain and sustain and/or protect human life. If they fail, it is reasonable to assume that the health of the user or other persons may be endangered.

Table of Contents

1	Package Description	4
2	Printed Circuit Board.....	6
2.1	Routing.....	6
2.2	PCB Pad Design.....	6
2.3	Over All PCB Design	6
3	Board Assembly	8
3.1	General Remarks	8
3.2	Solder Stencil	8
3.3	Solder Paste.....	9
3.4	Component Placement	9
3.5	Soldering	9
4	Cleaning	12
5	Inspection	13
6	Underfill Application.....	14
7	Rework	15

1 Package Description

This application note deals with assembly instruction for the Infineon Blade package family, including the Laminate Green Ultra thin Small Non Outline Non-Leaded Packages (LG-USON) as well as the Laminate Green, Ultra Integrated Quad Flat Nonleaded Packages (LG-UIQFN).

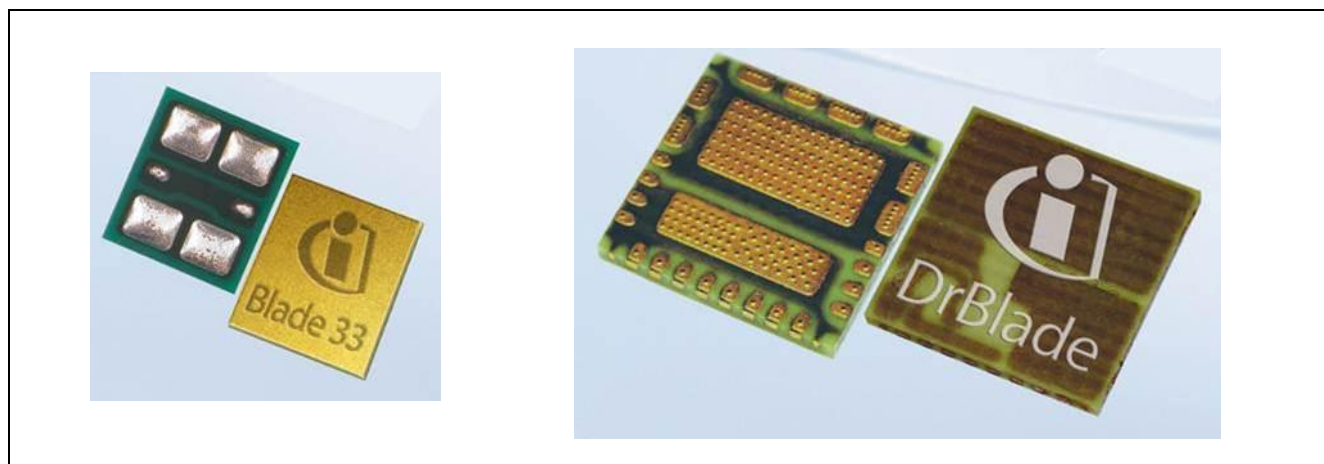


Figure 1 Examples of Infineon Laminate Packages: left: LG-USON-6-1; right: LG-UIQFN-32-2

Table 1 Package name Definitions

LG = Laminate Green PG = Plastic Green	T = thin V = very thin W = very very thin U = ultra thin X = extremely thin (S)D = (shrink) dual I = integrated	SON = small outline non leaded QFN = quad flat non leaded
---	--	--

The Infineon Blade package family consists of near chip-scale laminate encapsulated packages with an exposed top-side copper leadframe using vias to the bottom of the package to provide electrical and thermal contact to the printed circuit board. The exposed topside offers an additional thermal and electrical path for high-performance applications.

Features

- Thin package with an optimal package-to-chip ratio.
- Low package resistance by laminate/via interconnect technology
- Contacts are Ni/Au plated or presoldered with SnAgCu deposits compatible with Pb-free soldering.
- Optional topside cooling

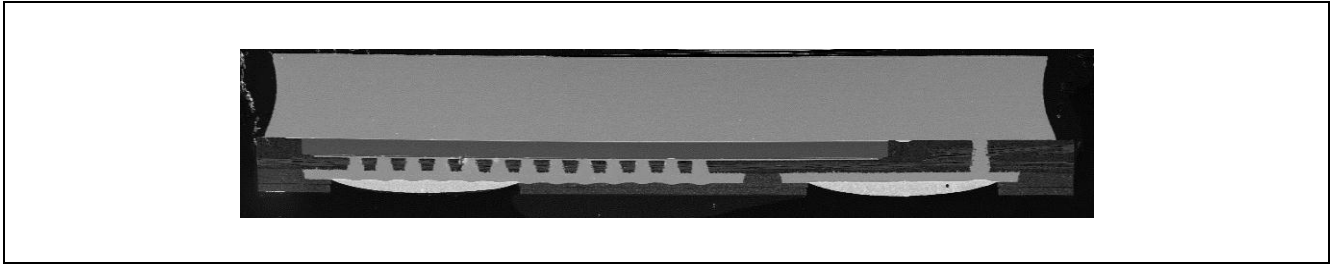


Figure 2 Example of a cross section of an Infineon Laminate Package (LG-USON)

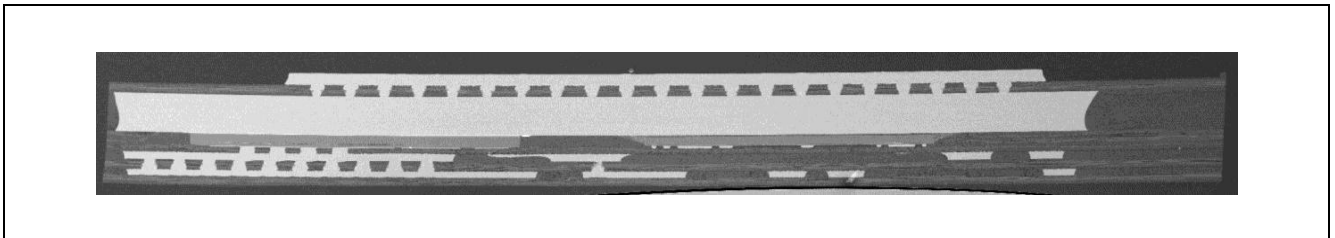


Figure 3 Example of a cross section of an Infineon Laminate Package (LG-UIQFN)

Semiconductor devices are sensitive regarding excessive Electrostatic Discharge, Moisture, mechanical handling and contamination. Therefore they require specific precautionary measures regarding transport, storage, handling and processing. For details please refer to the General Recommendations for Assembly of Infineon Packages in section "Package Handling" at www.infineon.com/packages.

2 Printed Circuit Board

2.1 Routing

The PCB design and construction are key factors for achieving solder joints with high reliability. Packages with exposed pads should not be placed at the same opposite locations on either side of the PCB (if double-sided mounting is used), because this results in a stiffening of the assembly with earlier solder joint fatigue compared to a design in which the component locations are offset. Furthermore, it is known that the board stiffness itself has a significant influence on the reliability (temperature cycling) of the solder joint interconnect, if the system is used in critical temperature cycling conditions.

2.2 PCB Pad Design

The interconnect solder joint to board is influenced by:

- General pad technology (Solder Mask Defined; short: SMD and Non Solder Mask Defined; short NSMD)
- Specific pad dimensions
- Pad finish (also called metallization or final plating)
- Via layout and technology

Further information and PCB layout recommendations can be found in the General Recommendations for Assembly of Infineon Packages in section "Printed Circuit Board" (available online under <http://www.infineon.com/packages> - Download-Service). Please choose a specific package. Beside package and packing drawings and further package specific information the details about the PCB layout for this package can be found there. Please note that the recommendations can only give dimensions for the solder mask openings (if SMD is recommended) or the copper (if NSMD is recommended). Generally the smallest dimensions for corresponding copper pads respectively solder mask openings depend on the capability of the board manufacturer. For high current applications SMD pads are recommended and the copper dimensions for drain and source pads should be chosen as big as possible to enlarge the conductor cross-sections. Please note that there is no exact congruency of PCB pads and package pads.

2.3 Over All PCB Design

Depending on the type of package there will be copper lines visible at the package side walls. This is caused by the technology used for producing Infineon laminate packages. In some cases those copper lines are not internally connected to the solder pad located on the bottom of the package next to it. But at least there will be a package internal connection to any electrical potential of the device. When it comes to the board design of the application, using an Infineon laminate package this must be taken into account when looking at the distances insulating different electrical potentials against each other.

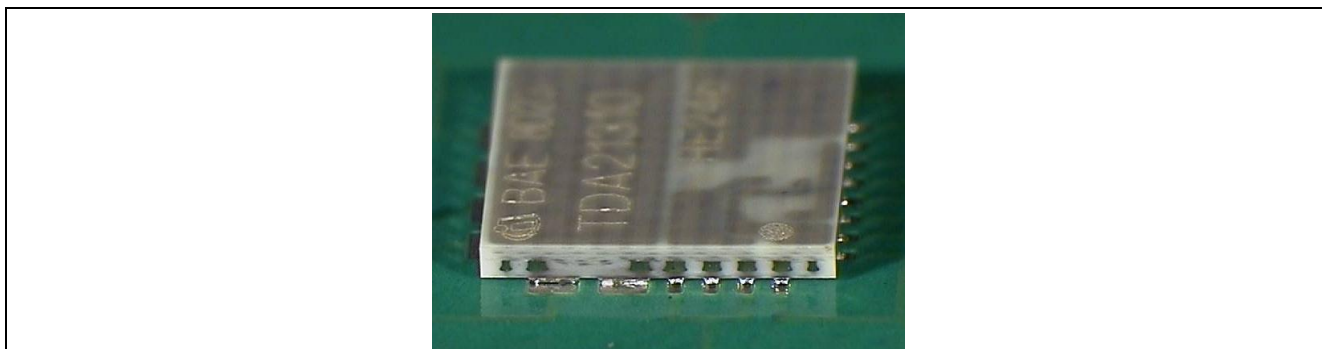


Figure 4 LG-UIQFN package soldered on board, showing copper lines at the package side wall

3 Board Assembly

3.1 General Remarks

Infineon Blade components are plated with Ni/Au finish; the bottom contacts (drain, source, gate) may have a solder deposit of Pb free composition (SnAgCu for LG-USON) or have exposed and plated leads (Ni/Au for LG-UIQFN). Infineon Blade packages are compatible with Pb-free reflow soldering.

Wave soldering Infineon Blade packages is not recommended. For some package types the package pads are hidden below the package. In other cases a wave-soldering-specific board layout would lead to a violation of isolation distances. Furthermore some Infineon Blade packages have metallic topsides which would wet during wave soldering.

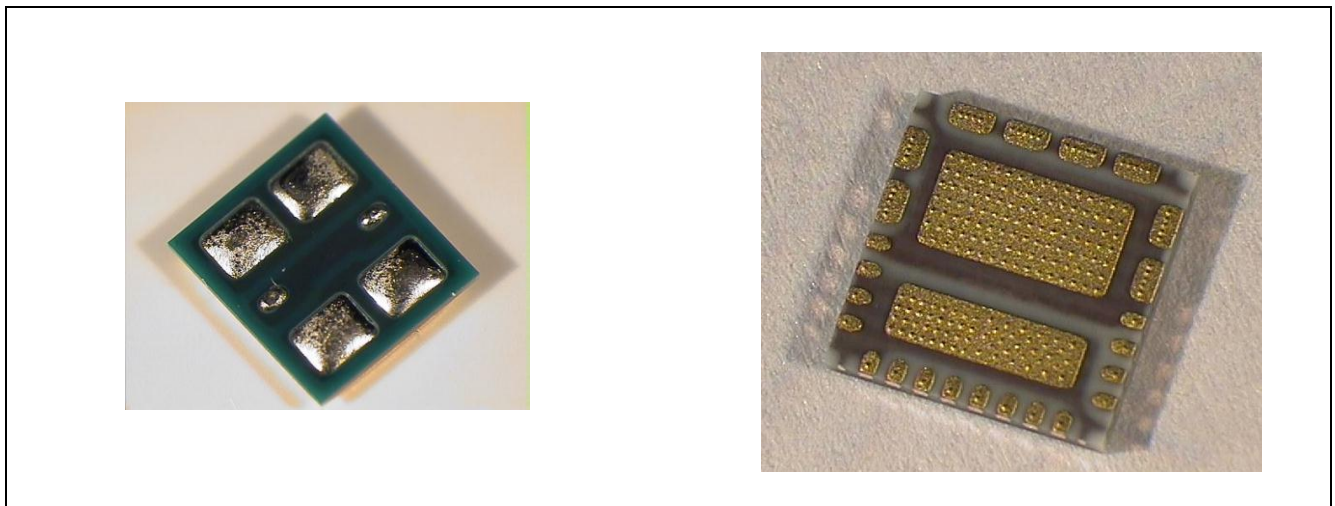


Figure 5 Bottom view of Infineon Laminate Packages; left: pre-soldered pads of a LG-USON; right: Ni/Au Pads of a LG-UIQFN

3.2 Solder Stencil

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. Too much solder paste will cause solder bridging, whereas too little solder paste can lead to insufficient solder wetting between all contact surfaces. In most cases the thickness of a stencil has to be matched to the needs of all components on the PCB. For typical Infineon Blade packages, 100- to 150- μm thick stencils are recommended. To ensure a uniform and high solder paste transfer to the PCB, laser-cut stencils (mostly made from stainless steel) should be preferred.

The apertures in general should be of the same size and shape as the metal pads on the PCB. However, during the solder print operation, the squeegee bends down into larger openings so that less solder volume is deposited. To reduce this effect, the stencil opening for the exposed drain pad can be segmented into four smaller areas ("pockets"). This reduction of solder also prevents device tilting.

Further details about specific stencil aperture recommendations can be found in Infineon's package data base in the internet. Please choose a specific package, where you can find an example for the stencil aperture layout of this package. Please note that the recommendations can only give a rough guideline. The ideal layout for a specific application depends on the mentioned factors. For instance

typical boundary conditions in IFX-internal investigations are no vias in pad and 120 µm stencil thickness. Vias may influence solder results; for Infineon Blade type packages vias in the PCB should be used only for thermal vias under an exposed pad (e.g. for LG-UIQFN) and should be covered or filled.

3.3 Solder Paste

Solder paste consists of solder alloy and a flux system. Normally the volume is split into about 50% alloy and 50% flux and solvents. In term of mass, this means approximately 90 wt% alloy and 10 wt% flux system and solvents. The flux system has to remove oxides and contamination from the solder joints during the soldering process. The capacity for removing oxides and contamination is given by the respective activation level. The contained solvent adjusts the viscosity needed for the solder paste application process. The solvent has to evaporate during reflow soldering. Pb-free solder pastes typically contain SAC305 (3.0 % Ag and 0.5 % Cu) or other so-called SAC-alloys (typically 1-4% Ag and <1% Cu).

A “no-clean” solder paste is preferred for Infineon Blade packages where cleaning below the component is difficult. The paste must be suitable for printing the solder stencil aperture dimensions; type 3 paste is recommended. Solder paste is sensitive to age, temperature, and humidity. Please follow the handling recommendations of the paste manufacturer.

3.4 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 according to their geometry. Positioning the packages manually is not recommended but is possible, especially for packages with big terminals and pitch. An automatic pick-and-place machine is recommended to get reliable solder joints.

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

The maximum tolerable displacement of the components is 20% of the metal pad width on the PCB. For example, Infineon Blade packages with 0.3 mm pad width, the device-pad-to-PCB-pad misalignment has to be less than 60 µm to assure a robust mounting process (even if the self-centering effect during reflow soldering may allow much more misplacement). Generally this is achievable with a wide range of placement systems. For details about factors influencing the component placement please refer to the General Recommendations for Assembly of Infineon Packages in section “Mounting of SMDs”.

3.5 Soldering

During the reflow process, each solder joint has to be exposed to temperatures above the solder melting point or “liquidus” for a sufficient time to get the optimum solder joint quality, whereas overheating the PCB with its components has to be avoided.

LG-USON packages are qualified acc. IPC/JEDEC J-STD-020. Please refer to the bar code label on the packing for the maximum peak package body temperature. When using infrared ovens without convection, special care may be necessary to assure a sufficiently homogeneous temperature profile

for all solder joints on the PCB, especially on large, complex boards with different thermal masses of the components.

The recommended type of process is forced convection reflow. Nitrogen atmosphere is recommended. Figure 6 shows a general forced convection reflow profile suitable for soldering Power-SON packages. Table 2 shows an example of the key data of such a reflow profile that can be used for the Pb-free alloys listed above. For further details about the reflow profile, please refer to the General Recommendations for Assembly of Infineon Packages in section “Mounting of SMDs” at www.infineon.com/packages.

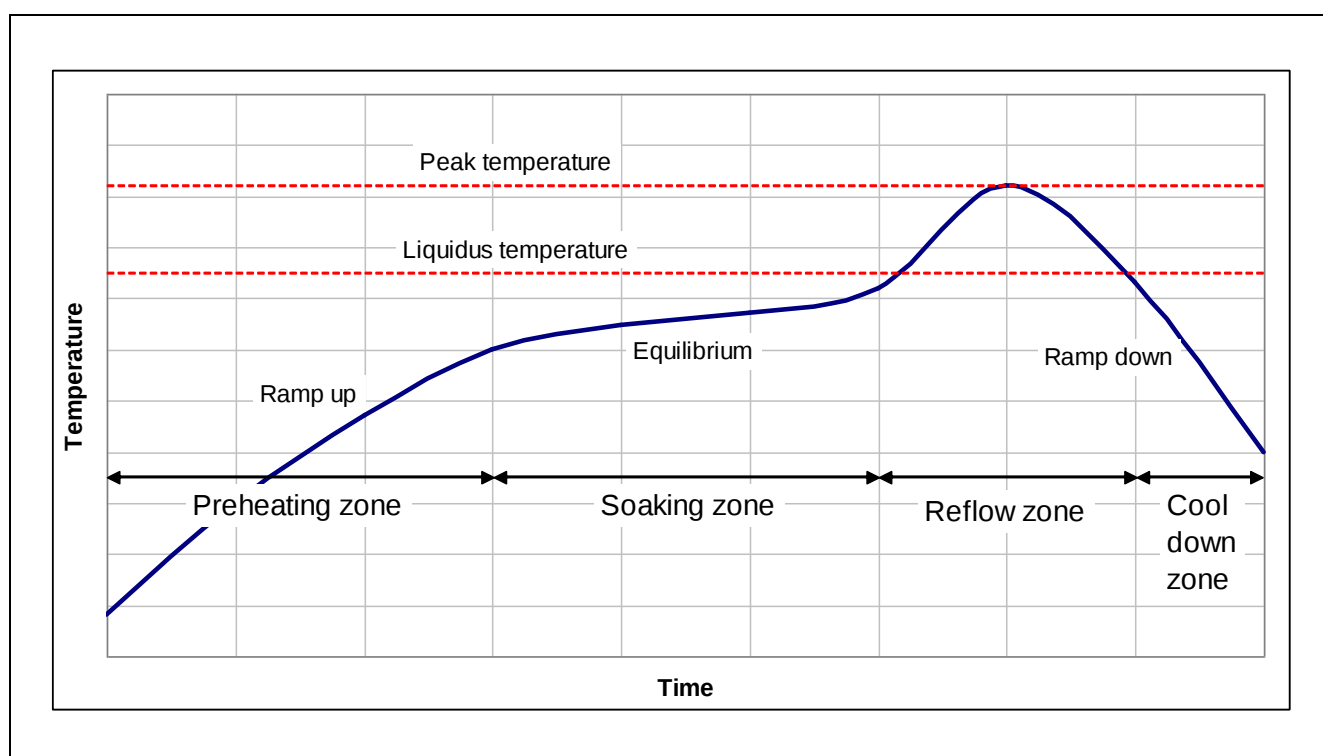


Figure 6 General forced convection reflow solder profile

Table 2 **EXAMPLE** for the key data of a forced convection reflow solder profile

parameter	minimum value	typical value	max. value (acc. IPC/ JEDEC J-STD-020)	main influence
preheating rate	1.0 K/s	2.5 K/s	3.0 K/s	flux system (solder paste)
soaking temperature	140 – 170°C	140 – 170°C	150 – 200°C	flux system (solder paste)
soaking time	50 s	80 s	120 s	flux system (solder paste)
peak temperature	230°C	245°C	260°C	alloy (solder paste)
reflow time above melting point (liquidus)	40 s	60 s	150 s	alloy (solder paste)
cool-down rate	1.0 K/s	2.5 K/s	8.0 K/s	

4 Cleaning

After the reflow soldering process, some flux residues can be found around the solder joints or spreading over the whole PCB. If a “no-clean” solder paste has been used for solder paste, the flux residues usually do not have to be removed after the soldering process. Be aware that cleaning beneath an Infineon Blade package is difficult because of the small gap between package substrate and PCB and is therefore not recommended. If the solder joints have to be cleaned, the cleaning method (e.g. ultrasonic, spray or vapor cleaning) and solution have to be selected in consideration of the packages to be cleaned, the flux used in the solder paste (rosin-based, water-soluble, etc.), and environmental and safety aspects. Even small residues of the cleaning solution should be removed/dried very thoroughly. Contact the solder paste or flux manufacturer for recommended cleaning solutions.

For Infineon laminate packages “no-clean” solder paste is recommended.

5 Inspection

Compared to typical SMD components that have gullwing leads, the drain, source and gate solder joints of LG-USON packages are formed underneath the package. A visual inspection of the solder joints with conventional Automatic Optical Inspection (AOI) systems is limited to the position of the device. X-ray Inspection is recommended instead. Automatic X-ray Inspection (AXI) systems are appropriate for efficient inline control. 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, analysing and data transfer 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.

As a rule-of-thumb, a 25% maximum voiding rate (X-ray inspection top-down view) for the perimeter pads is a starting point. The bigger exposed pad may tend to more or less voiding, depending on board pad size, via and stencil layout, solder paste, and reflow profile. Generally such big solder pads do not provide enough surface for gas, which generates during reflow, to escape. Therefore solder joints of big pads generally tend to show more voiding.

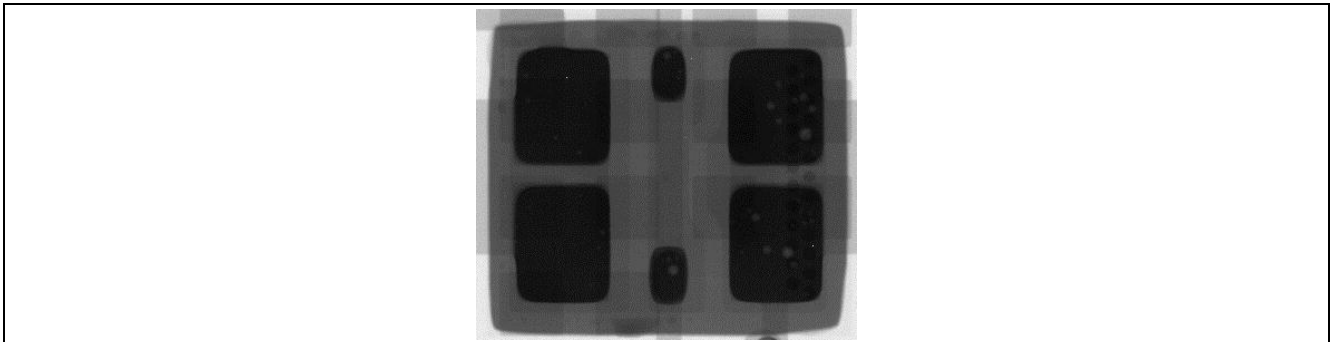


Figure 7 X-Ray image of a reflow soldered Infineon laminate package (LG-USON)

Cross-sectioning of a soldered package as well as dye penetrant analysis can serve as tools for sample monitoring only, because of their destructive character. Nonetheless, these analysis methods must be used during engineering of new products at customers' production sites to get detailed information about the solder-joint quality. Figure 8 shows typical cross-sections through a solder joint.

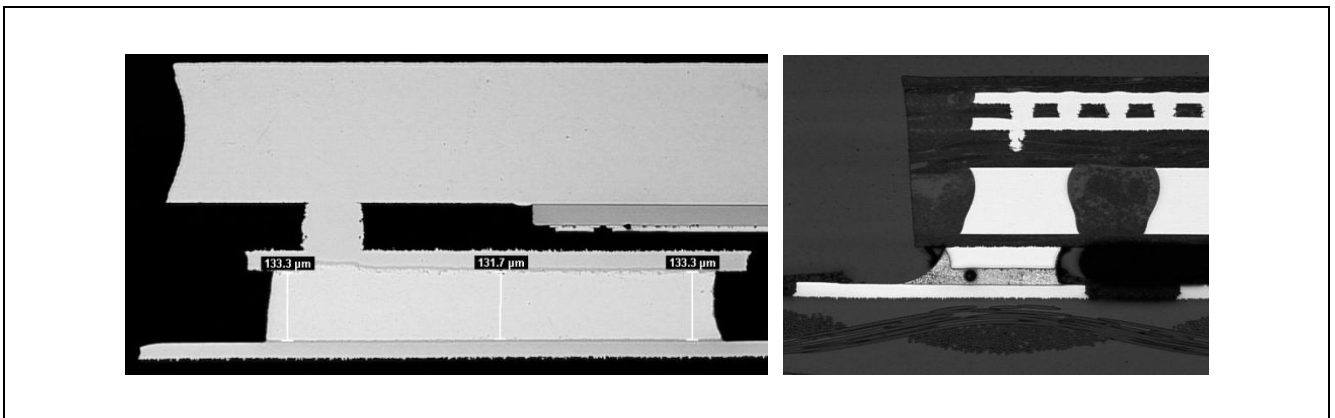


Figure 8 Cross sections of soldered Infineon laminate packages; left: LG-USON; right: LG-UIQFN

6 Underfill Application

The board-level reliability of Infineon Blade packages is robust enough to do without underfill. Nonetheless if underfill should be applied, we recommend assessing the electrical and (thermo-) mechanical behavior of assembled and underfilled components over the lifetime of the electronic device. Some underfill materials may negatively influence the lifetime of the assembled devices.

7 Rework

If a defective component is detected after board assembly, the device can be removed and replaced by a new one. Due to possible damage while removing the component, a desoldered component should not be reused. Desoldering the old component (if analysis afterwards is planned) and resoldering of the new component has to be done very thoroughly. Single solder joint repair of Power-SON packages is very difficult, if not impossible, and therefore not recommended at all.

The following restrictions have to be taken into account prior to and during rework process:

- Dry the PCB and components prior to rework. A proper drying procedure for SMD packages is described in the international standard IPC/JEDEC J-STD-033. Please also refer to the recommendations of your PCB manufacturer and take all specific needs of components, PCB and other materials into account.
- Use tools which do not damage the component mechanically. Please note that mechanical forces need not lead necessarily to external damage, which could be detected more easily, but may cause internal damage and reduce the component reliability. A proper handling system with vacuum nozzle may give the most gently process and is therefore recommended. However, the impact of rework tools has to be assessed properly.
- Whatever heating system is used (hot air, infrared, hot plate, ...), it has to be ensure that the applied temperature profile at the component never exceeds the maximum profile specified in the international standard IPC/JEDEC J-STD-020. Internal investigations have shown that in any case a recording of the temperature profile is needed.
- Generally: A higher degree of manual handling increases the effort for documentation, training and monitoring of the rework process(es). Investigations have shown that if distance, time, and airflow are properly controlled, a hot air temperature of 300°C can be used, for example, without violating the maximum allowed reflow profile. On the other hand the same hot air temperature setting was able to heat up the component to above 280°C, which exceeds the component's specification by far, if other process parameter were changed.
- If a desoldered component should be reused please consider that components are qualified to withstand maximum three reflow cycles. In this specific case two are already applied during rework and only one is left for the board assembly.

For further information about tooling and process steps for rework, please refer to the General Recommendations for Assembly of Infineon Packages in section "Rework".

www.infineon.com