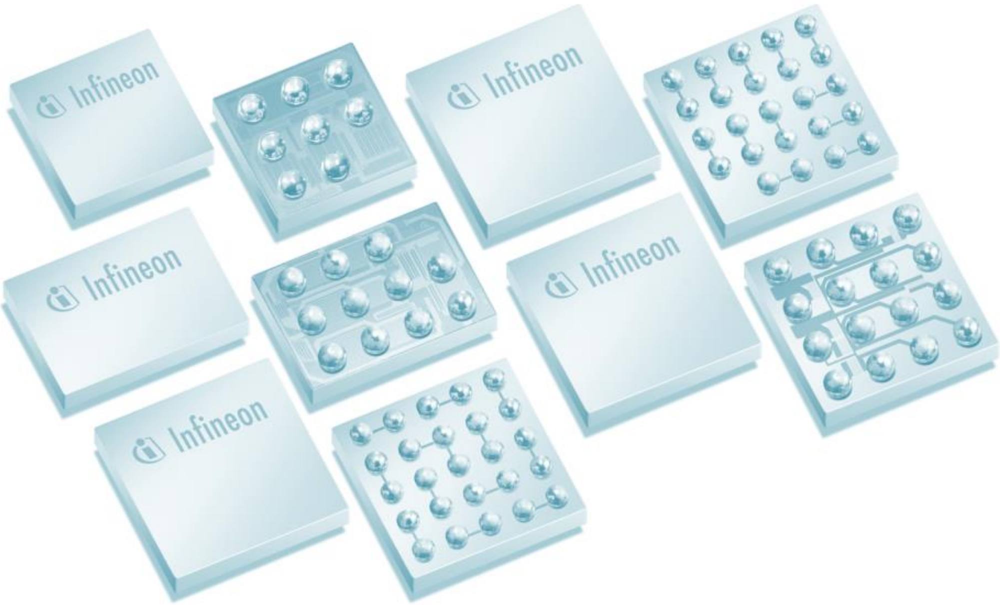
Recommendations for Board Assembly of Infineon xWLy Packages



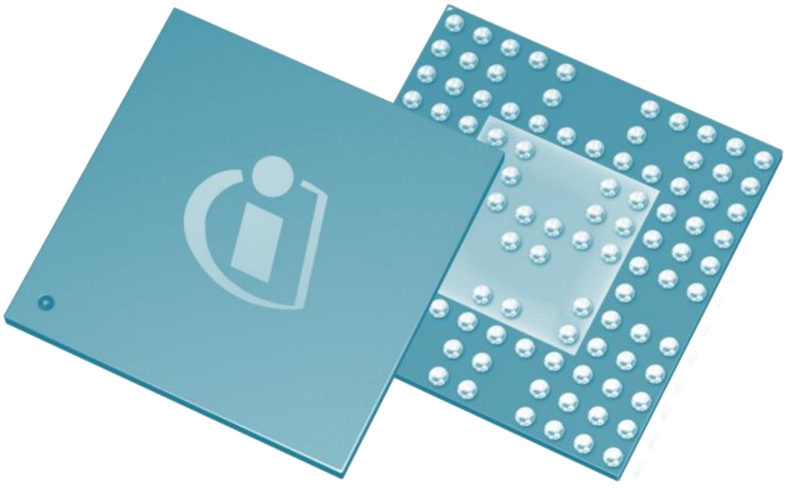


Table of Contents

1 Package description

1.1 Package nomenclature

Infineon's wafer-level packages are available as three different types:

- SG-WLP (please refer to Figure 1)
  - Chip-size package without redistribution layer
  - No additional passivation, dielectric, or solder-stop layer
  - Pb-free solder balls (250 or 300 µm)
  - Typically for products with up to 25 I/Os

- SG-VFWLB/SG-WFWLB/SG-UFWLB (please refer to Figure 2)
  - Chip-size package with fan-in redistribution
  - Dielectric/solder-stop layer
  - Pb-free solder balls (250 or 300 µm)
  - Typically for products with up to 65 I/Os

- PG-WFWLB (please refer to Figure 3)
  - Chip scale package with fan-in/fan-out redistribution
  - Dielectric/solder-stop layer
  - Extension of silicon die by dielectric material
  - Pb-free solder balls (250 or 300 µm)
  - Backside protection (optional)
  - Typically for products with up to 300 I/Os

Package description

Abbreviations and acronyms used in product names

S:	Silicon (only fan-in redistribution)
P:	Plastic (fan-in and fan-out redistribution possible)
G:	Green
V:	Package thickness $<$ 1.0 mm
W:	Package thickness $<$ 0.8 mm
U:	Package thickness $<$ 0.65 mm
F:	Fine pitch
WLP:	Wafer-Level Package
WLB:	Wafer-Level Ball Grid Array
xWLy:	Acronym for Infineon's Wafer-Level Package family including SG-WLPs, SG-VFWLBs, SG-WFWLBs, SG-UFWLBs and PG-WFWLBs

General package features:

- Applicable for a wide range of chip sizes
- Flexible rerouting (WLB)
- Applicable for packages with low to medium number of I/Os
- Ball pitches 0.4 mm and 0.5 mm available
- Excellent electrical and thermal performance

Please note that the word "package" in this document always refers to the component type (excluding the electrical function; in this case SG-xWLy packages). By comparison, the word "packing" always deals with items that cover and protect packages during transportation or storage, and/or hold packages during pick-and-place. For example, packing types include tape-and-reel, trays, and cardboard boxes.

Semiconductor devices are sensitive to excessive electrostatic discharge, moisture, mechanical handling, and contamination. Therefore they require specific precautionary measures to ensure that they are not damaged during transport, storage, handling, and processing. For details, please refer to the General Recommendations for Assembly of Infineon Packages in "Package Handling" (available at www.infineon.com\packages).

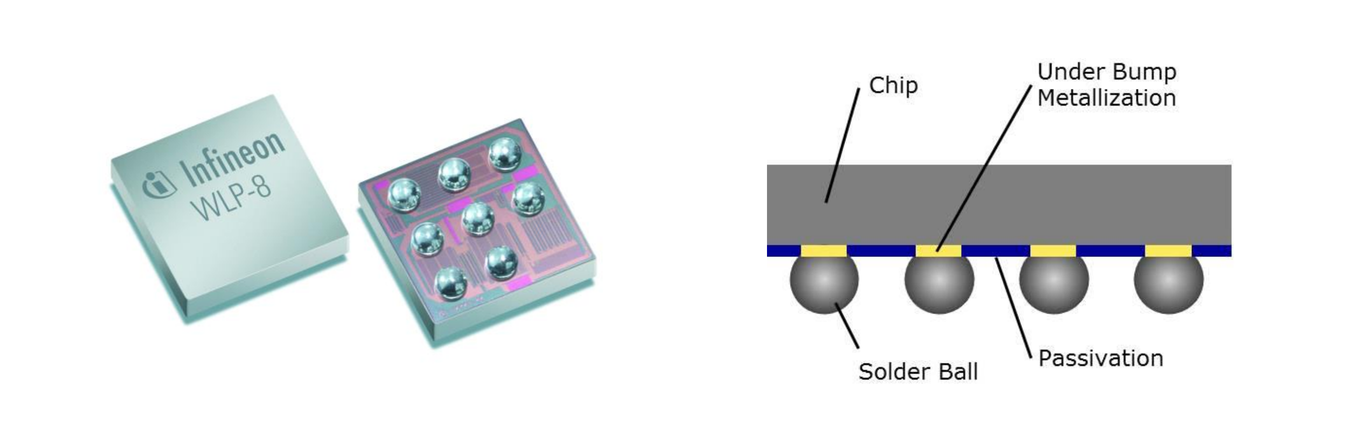


Figure 1 Illustration and schematic cross-section of an SG-WLP

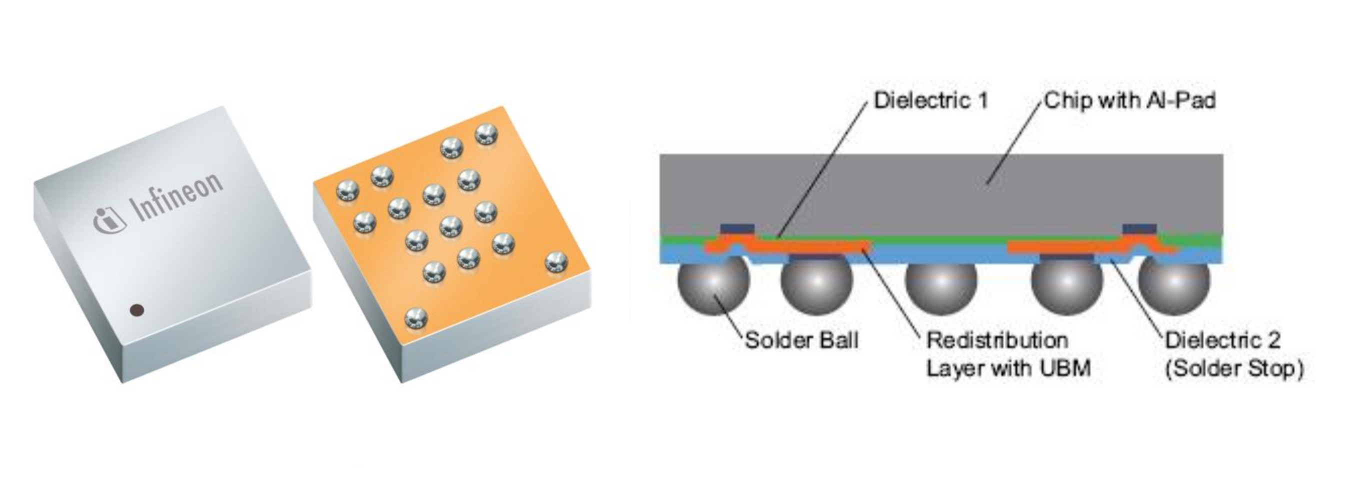


Figure 2 Illustration and schematic cross-section of an SG-VFWLB, SG-WFWLB or SG-UFWLB

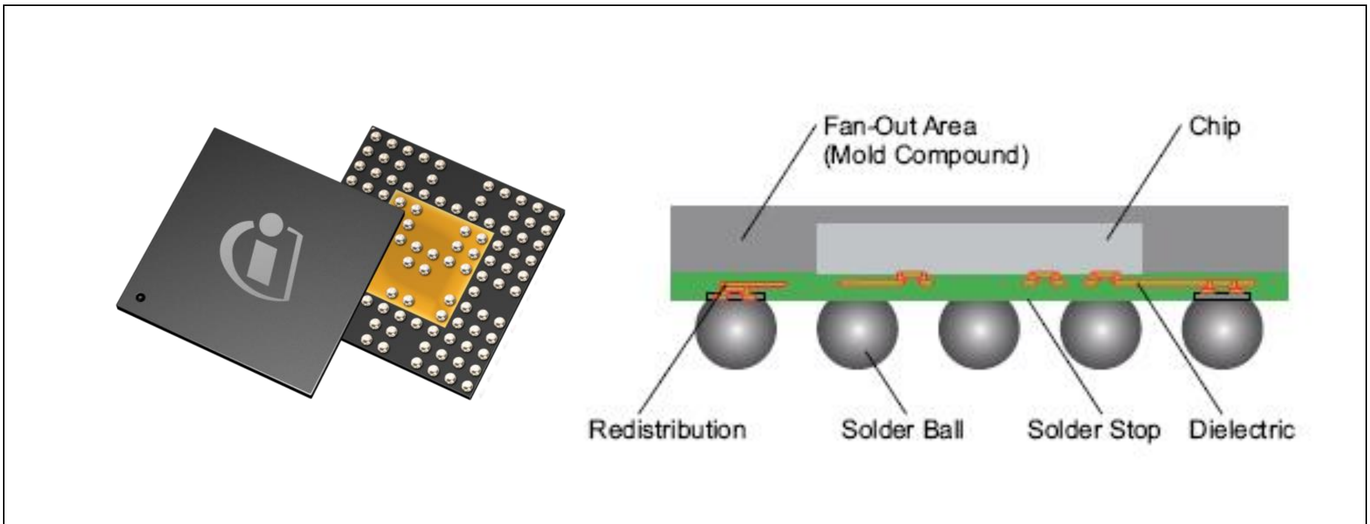


Figure 3 Illustration and schematic cross-section of a PG-WFWLB

1.2 Package handling

Automatic and especially manual handling of components in or out of the component packing may cause mechanical damage to WLB/WLP package balls and/or body. The mechanical properties of the package body are mainly determined by the silicon, which is much more brittle than the mold compound of "classic" package types. Any manual handling of xWLy packages during board assembly should be avoided. Parameters and settings for automated SMT equipment are mentioned in section 3.3. If it is necessary to handle packages for any other purpose such as analysis, plastic tweezers should be used instead of metal ones.

After finishing the board assembly processes, WLB/WLP packages are fixed/soldered to the PCB, but their bodies are not protected from the environment. Therefore, during handling of the assembled Printed Circuit Boards (PCBs) it is necessary to prevent any mechanical impact to packages. For instance, critical items are support pins for solder-paste printing of the second PCB side, insertion of assembled PCBs into magazines, clamping or support in adapters for electrical testing, or insertion of the assembled PCBs into the housing of the application.

Additionally, contamination applied to components or packing may cause or induce processes that (together with other factors) may lead to a damaged device. The most critical issues are:

- Solderability problems
- Corrosion
- Electrical shorts (due to conductive particles)

In many applications, the package must be electrically isolated from its mounting surface. The isolation material has a comparatively high thermal resistance, which raises junction operating temperatures.

2 PCB

2.1 PCB routing

The array configuration of solder spheres on xWLy packages implies different concepts for routing the signal, power, and ground pins on the Printed Circuit Board (PCB).

Generally the PCB design and construction is a key factor for the reliability of the solder joints. For example, xWLy packages should not be placed at the same locations on opposite sides of the PCB if double-sided mounting is used, because this stiffens the assembly and results in much faster solder-joint fatigue compared to a design in which the component locations are displaced with respect to each other. Furthermore it is known that the lower stiffness of thinner boards (e.g. 1.0 mm) improves solder joint reliability (temperature cycling) compared to thicker boards (e.g. 1.6 mm).

To ensure proper routability of Ball Grid Array (BGA) packages with 0.5 or 0.4-mm pitches, PCB technology providing line/space dimensions of 100 µm or less (e.g. 75µm) is necessary. Depending on the ball matrix of the package, microvia technology might be needed to route several signals by the second layer.

SG-xWLy packages are chip-scale bare die packages using no mold compound covering the silicon. Due to the physical properties of silicon, these kinds of packages are sensitive to mechanical load whether or not they already have been soldered to a PCB. Therefore the packages should always be protected against mechanical force. One way to do that is with a special PCB layout. Every application consists of a lot of passive components for various electrical purposes. If some of those components can be placed close to the xWLy, they will create a protective circle around it (see Figure 4 and 5). Depending on the package size, it may make sense to place more than one component per side as a protective measure. Passive components around the xWLy package should be a little bit thicker than the xWLy package (most of these are 0.7 mm thick). X and Y dimensions as well as the number of passive components per side need to be chosen according to the package size of the xWLy. This will ensure that any mechanical impact from the side will not be able to contact the xWLy package. That will improve the robustness of the application design.

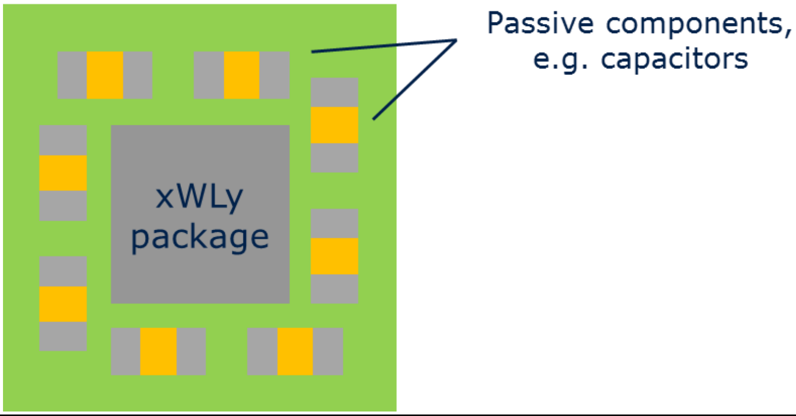


Figure 4 Schematic view of recommended placement of passive components around an xWLy package

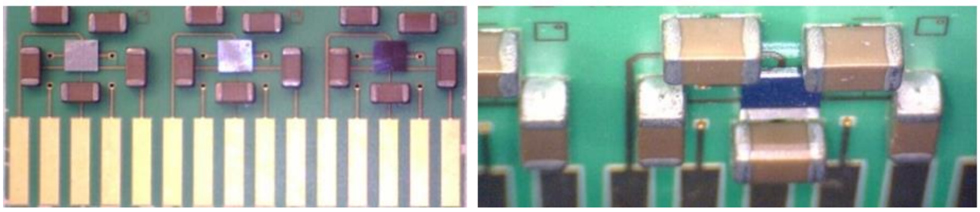


Figure 5 Typical PCB layouts using passive components to protect xWLy packages

2.2 PCB pad design

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

- Solder-Mask Defined (SMD) pad: The copper pad is larger than the solder-mask opening above this pad. Thus the land area is defined by the opening in the solder mask.
- Non-Solder-Mask Defined (NSMD) pad: Around each copper pad there is solder-mask clearance. It is necessary to specify the dimensions and tolerances of the solder-mask clearance so that the solder mask does not overlap the solder pad. Depending on PCB manufacturers' tolerances, 75 µm is a widely used value for clearance.

NSMD pads provide more space for routing and result in more reliable solder joints because the side walls of the lands are wetted with the solder, which leads to less stress concentration in the solder joint. Therefore, NSMD pads are recommended for the solder pads of an xWLy footprint on the PCB.Figure 6 shows the recommended PCB pad designs. Refer to Table 1 for the appropriate dimensions, which are ball-size-specific. The values in the table are typical dimensions. The details depend on the PCB technology used, capability of the suppliers, and the routing. If drilled via holes are placed between the pads, they should be closed (plugged or plated) to prevent solder flow into the vias. If microvias are placed inside the pads, relatively flat vias should be specified. Deep dips inside the pads may cause increased solder-joint voiding.

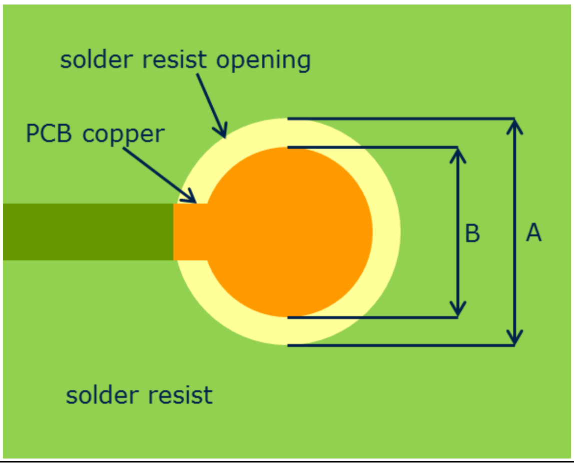


Figure 6 PCB pad design recommendation (NSMD pad)

Table 1 PCB pad size recommendations

Ball diameter	250 µm	300 µm
Solder resist opening "A"	350 µm	375 µm
Copper pad "B"	250 µm	275 µm

3 PCB assembly

3.1 Solder-paste application

The solder paste is applied onto the PCB metal pads by screen printing. The volume of the printed solder paste is determined by the stencil aperture and the stencil thickness. In most cases, the thickness of a stencil has to be matched to the needs of all components on the PCB. To respect the stencil aperture "area ratio" for the typically round-shaped apertures for xWLy packages, 100 or 80-µm-thick stencils should be used. The aperture diameter can be the same size as the metal pad diameter on the PCB. Overprinting may give a better release of solder paste from the stencil aperture, but can increase the risk of solder bridging.

To ensure a uniform and high solder-paste transfer to the PCB, lasercut (mostly made from stainless steel) or electroformed stencils (nickel) are preferred. For low-pitch devices such as Infineon's xWLy packages, a regular cleaning cycle (wet/dry) of the stencil is advisable.

A "no clean" solder paste is preferred for board assembly of xWLy packages because removing flux residues under an xWLy package is very difficult. Regarding the grain size of the paste, type 3 is suitable for pitches greater than 0.5 mm. Test results at Infineon have shown that type 3 works well for the recommended stencil apertures of xWLy packages (Table 2). Nevertheless, paste type 4 may help to improve transfer efficiency for the stencil printing process during mass production.

Table 2 Stencil aperture size recommendations

Ball size	Stencil aperture diameter	Typical stencil thickness
250 µm	250 µm	80 – 100 µm
300 µm	275 µm	100 µm

3.2 Solder-paste inspection

For inspection of solder-paste depots after printing, vision systems can be used which are either integrated into the printer or separate automated Solder-Paste Inspection (SPI) equipment. The solder-paste x-y-coverage and solder-paste volume can be measured. Adequate acceptance criteria have to be defined based on the manufacturing setup.

3.3 Component placement

xWLy packages need to be placed accurately depending on their geometry. Manual placement is not recommended.

Pick-and-place of packages is typically done using vacuum nozzles. To avoid pick-up issues (e.g. picking up the component edgewise), the size of the nozzle should be slightly smaller than the package outline. Choosing the wrong nozzle might lead to damage of the package edge or wear out the nozzle prematurely. The preferred nozzle material is rubber/plastic. Metal and ceramic tools may cause scratches on the component surface and should therefore be avoided.

To assess the robustness of Infineon xWLy packages during pick-and-place, a simulation was initiated to evaluate the impact of the pick up/placement force on the package (see Figure 7). The calculation was based on the following parameters:

- Package: SG-WFWLB-5-1 (~1.4 x 1.8mm)
- Nozzle size: 0.8 x 0.8 mm
- Nozzle position centered on package top
- Rigid surface below component

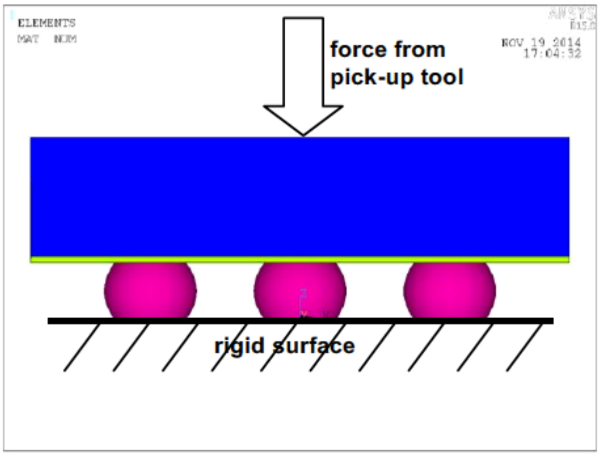


Figure 7 Illustration of the simulation applying force to package top

While typical pick-and-place forces are below 5 N, the simulation showed that there won't be mechanical damage to the device even if the impact is caused by a force of 50 N. Previous experience with many similar Surface mount assemblies also confirms that a properly set up pick-and-place process will not damage the device.

During pick-and-place, a method recognizing the solder balls instead of just the outline is recommended. With the right machine setting for the lighting, this will be the most accurate method.

Packages having solder balls benefit from the self-alignment effect of the surface tension of liquid solder during the reflow process. This makes the packages tolerate a misplacement of 30% of the metal pad diameter on the PCB without negative effects on the resulting solder joint. Since state-of-the-art placement systems are more accurate than that, there is no need for special treatment of the xWLy packages regarding placement accuracy. The placement force should be adjusted to a level that does not lead to excessive solder paste squeezed out next to the solder balls to avoid an increased risk of electrical shorts.

To avoid the risk of damaging the component, bending stress should be minimized. In particular, local stress such as tension forces on the xWLy package should be avoided during all handling steps:

- Convex bending of the device with tension stress on the xWLy package (Figure 8, Case 1) must be avoided. Such stress conditions could occur during electrical testing, for example, when the forces on top and bottom of the package are applied by probe pins (pogo pins).
- If force is applied to the center of the device's bottom side, the device must be supported on the opposite side (Figure 8, Case 2) to provide a central counterforce to prevent package bending.

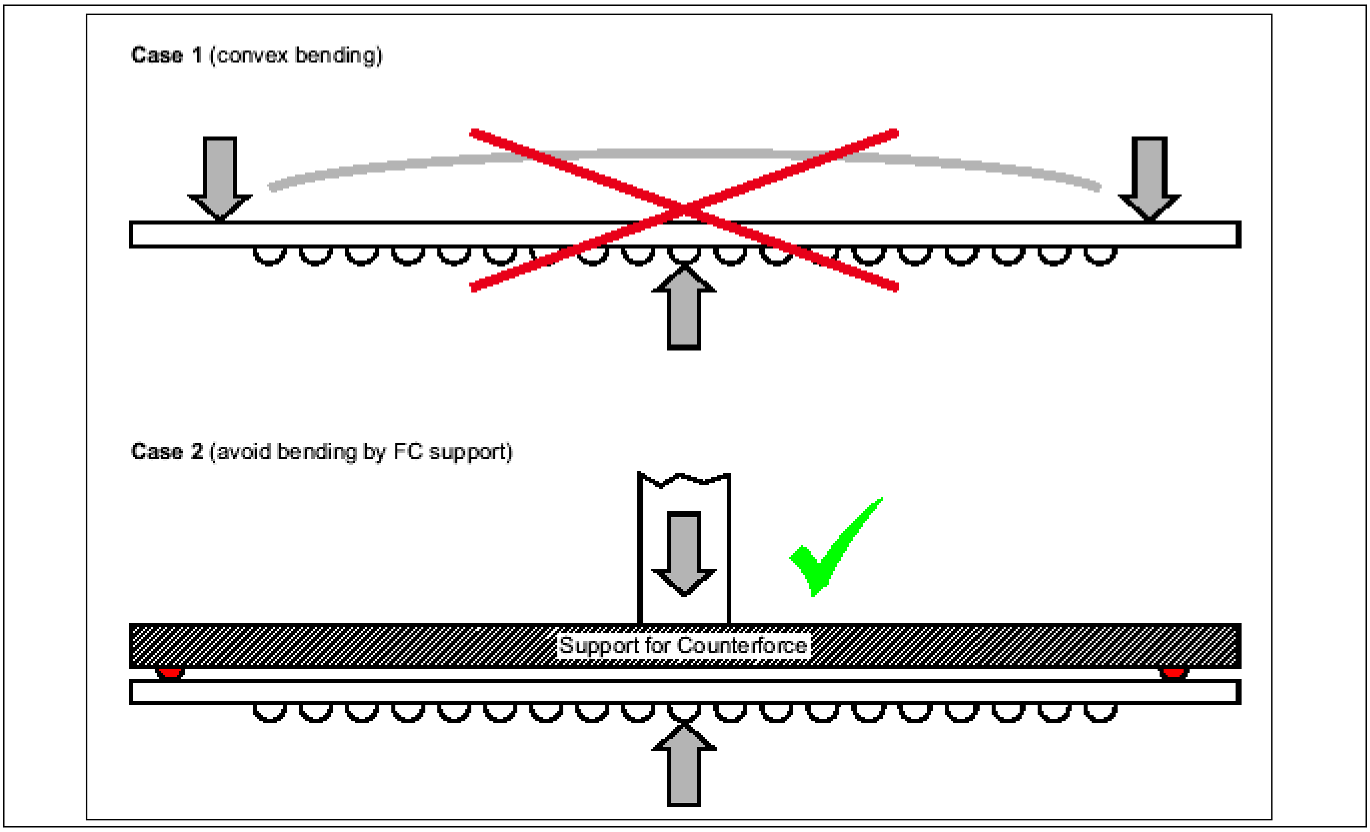


Figure 8 Case 1 shows bending stress applied on Wafer-Level Package by insufficient support. Case 2 shows a nozzle supporting the package and applying a counterforce.

3.4 Reflow soldering

The temperature profile of a reflow process is divided into several phases, each with a special function (Figure 9). The individual parameters are influenced by various factors, not only by the package. It is essential to follow the solder-paste manufacturer's application notes. Usually PCBs contain more than one package type and therefore the reflow profile has to meet the requirements of all components and materials. We recommend measuring the solder-joint temperatures by thermocouples beneath the respective packages. Components with large thermal masses do not heat up at the same speed as lightweight components. In addition, the position and the surrounding of the package on the PCB, as well as the PCB thickness, can influence the solder-joint temperature significantly.

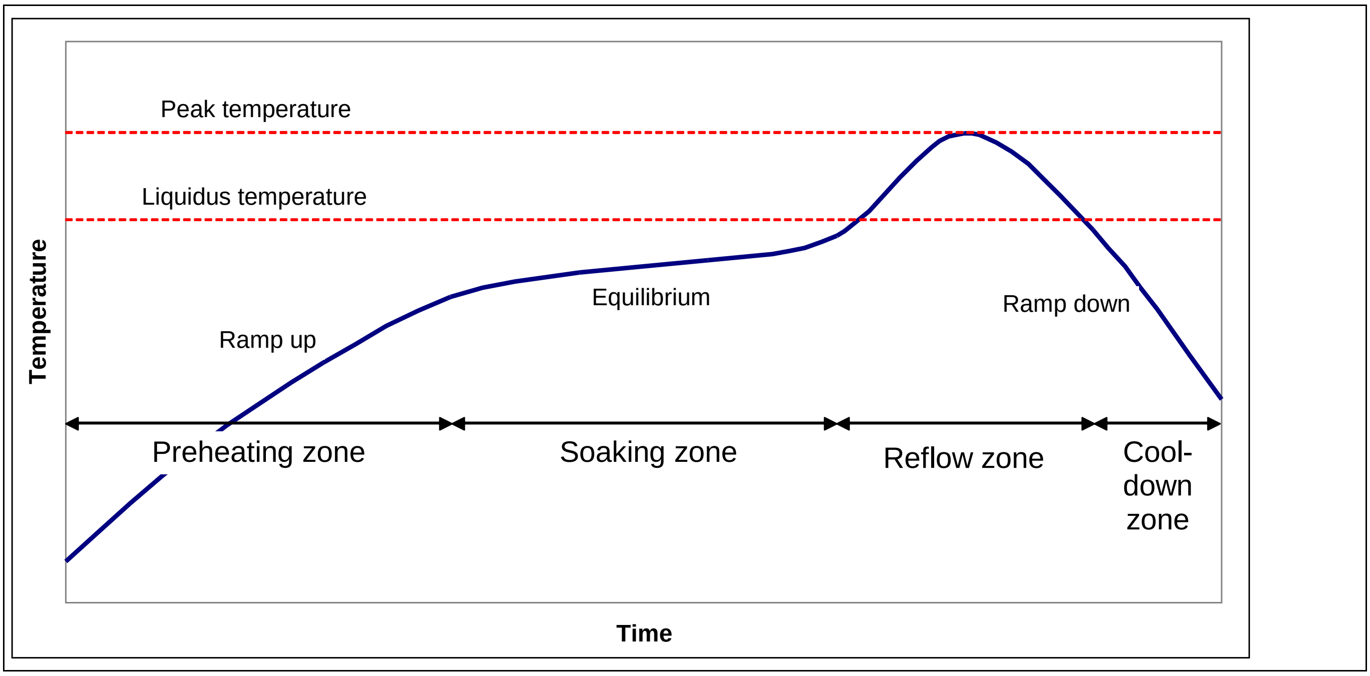


Figure 9 Various phases of a reflow solder profile

Infineon's surface-mount packages are qualified according to the J-STD020 standard. This standard states the following about the maximum peak temperature: "Tolerance for peak profile temperature (Tp) is defined as a supplier minimum and a user maximum".

3.5 Double-sided assembly

xWLy packages are generally suitable for mounting on double-sided PCBs. First, one side of the PCB is assembled with components and soldered. Afterwards, the second side of the PCB is assembled with components and soldered again. Be aware that packages (especially those with a high mass) could fall off the board during the second reflow process, while it is face down. In such cases, these packages have to be assembled during the last (second) reflow process. As a rule-of-thumb, a weight limit of 0.2 g/mm^2 soldered area (NSMD pad) can be assumed, and therefore light packages such as xWLy are in general not affected. Components can also fall off as a result of vibration and the airflow in the reflow oven.

If the solder-joint thickness is a critical dimension, please be aware that solder joints of components on the first side will be reflowed again in the second reflow step. In the reflow zone of the oven (i.e. where the solder is liquid), the components are only held by wetting forces from 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. The shape of the joints will be frozen at temperatures below the melting point of solder and therefore result in a higher stand-off on the first side after the reflow process.

In a double-sided assembled PCB, the locations of components mounted on the opposite side of the PCB need to be taken into account. A component soldered to the PCB will influence the thermomechanical behavior of the PCB itself. If another package is mounted exactly on the opposite side of the PCB, it may significantly influence the lifetime of the solder joints of an xWLy package, for example.

4 Cleaning

After the soldering process, flux residues can be found around the solder joints. If a "no-clean" solder paste or flux has been used, the residues usually do not have to be removed after the soldering process. If the solder joints have to be cleaned, the appropriate cleaning method (e.g. ultrasonic, spray or vapor cleaning) and solution will depend on the packages to be cleaned, the flux used (rosin-based, water-soluble, etc.), and environmental and safety aspects. Even small residues of the cleaning solution should also be removed or dried very thoroughly. Contact the flux or solder-paste manufacturer for recommended cleaning solutions.

Be aware that removing residue under an xWLy package is challenging because of the small gap between package body and PCB surface. In general, using a no-clean solder paste or flux is recommended without a cleaning step afterwards.

5 Underfilling

Infineon Technologies has performed reliability tests (e.g. drop test, temperature cycling on board, and bend test) on test boards without underfilling. The results have been positive, passing typical market requirements. Based on these results, additional underfilling is not necessary for xWLy packages, and we strongly recommend not using underfill. Depending on the properties of the underfill material and properties of other parts of the application (such as the PCB), underfilling can have either a positive or a negative impact on the reliability of solder joints or the package construction in board level condition. Infineon cannot offer reliability assessments of various underfill materials available on the market. Thus it is the customer's responsibility to investigate the reliability and performance of xWLy packages if using underfill is being considered.

6 Inspection after soldering

The only reasonable method to achieve efficient in-line control is to use Automatic X-Ray Inspection (AXI) 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-transfer routines. These systems enable the user to reliably detect soldering defects such as poor soldering, bridging, voiding, and missing parts.

Figure 10shows a typical X-ray image of a SG-WLP-25-1 package soldered to an IFX test board. For the acceptability of electronic assemblies, please refer to the IPC-A-610 standard. IPC-A-610 gives a criterion for the void size in solder joints of BGAs of 25% maximum voiding rate (for X-ray inspection topdown view). As a rule-of-thumb, this criterion can also be used for xWLy packages. Please note that the maximum acceptable size of voids depends on the required reliability. On the other hand, the size and number of voids are affected by different factors such as solder paste (especially flux), reflow profile, microvias and their flatness, board pad layout, board pad finish, etc.

xWLy packages can also be inspected with endoscopes, which are especially helpful for detecting failures of solder joints at the outer balls. The optical head of the system moves around the package near the PCB area. The user can look along the solder rows by adjusting the focus. The pictures from such an endoscopic system are much easier to interpret than X-ray images.

Solder joints of xWLy packages are comparable to those of fine-pitch BGA packages. Therefore, the process of inspecting and analyzing an xWLy package is comparable to the process used to inspect and analyze a BGA.

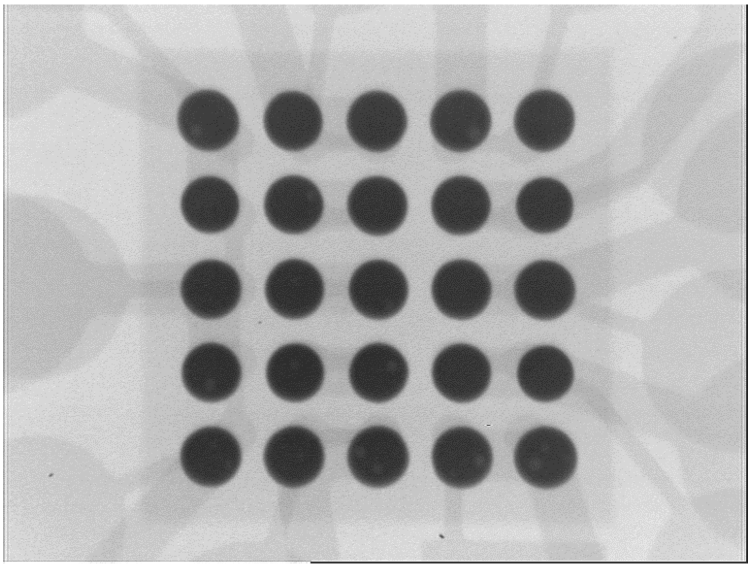


Figure 10 Example of an X-Ray image of an Infineon WLB package

7 Rework

Reworking WLPs may follow any of three different process flows depending on the failures that have to be distinguished:

- The PCB is expected to be defective:
  The whole assembly will be discarded.
- Solder joint(s) of an xWLy component is expected to be defective:
  The component must be desoldered and discarded.
  Repair of a single xWLy solder joint is not possible. Reusing of a desoldered component is not possible.
  The desoldered component must be replaced by a new one.
- The WLP is expected to be defective:
  Reusing a desoldered component is not possible. The component must be desoldered and discarded. **The desoldered component must be replaced by a new one.**

Attention: ***If a component is suspected to be defective and the component supplier wishes to perform a failure analysis with this component, please do not remove this component from the PCB. The entire PCB with components should be sent to Infineon Technologies. This guarantees that no further defects are introduced to the component by removal from the PCB, which may hinder the failure analysis at the supplier's facility or even make the failure analysis impossible. This is valid for underfilled components as well as for components without underfill!***

If an underfill that is not labelled as "reworkable" is used, repair of the PCB and/or the component is not possible. If underfill labelled as "reworkable" is used, it is possible to reuse a non-defective PCB. For details, please refer to the information provided by the underfill or rework station supplier. Failure analysis at Infineon is possible when the component together with the PCB board is sent to Infineon. Please note that damage caused by mechanically removing underfill from PCB may result in reduced reliability.

The rework process is commonly done on special rework equipment. There are a lot of systems available on the market that meet the following requirements for reworking xWLy packages:

- Heating:
  Hot air heat transfer to the package and PCB is strongly recommended. Temperature and air flow for heating the device should be controlled. With programmable temperature profiles (e.g. by PC controller) it is possible to adapt the profiles to different package sizes and thermal masses. PCB preheating from the bottom side is recommended. Infrared heating can be applied, especially for preheating the PCB from the bottom side, but it should be only supplementing the hot air flow from the top side. Nitrogen can be used instead of air.
- Vision system:
  The bottom side of the package as well as the site on the PCB should be observable. A split optic should be used for precise alignment of package to PCB, especially in case of a component replacement. Microscope magnification and resolution should be appropriate for the pitch of the device.
- Moving and additional tools:
  The device should be relocatable on the whole PCB area. Placement accuracy is recommended to be better than ±100 µm. The system should have the capability of removing solder residues from PCB pads (special vacuum tools).
- Tools for package handling:
  It is recommended to use only vacuum-supported tools comparable to what is used in the pick-and-place process for handling xWLy packages. As for pick-and-place, the right nozzle size should be chosen. It is also important not to apply any uncontrolled force to the package during rework processes.

Trademarks of Infineon Technologies AG
µHVIC™, µIPM™, µPFC™, AU-ConvertIR™, AURIX™, C166™, CanPAK™, CIPOS™, CIPURSE™, CoolDP™, CoolGaN™, COOLiR™, CoolMOS™, CoolSET™, CoolSiC™, DAVE™, DI-POL™, DirectFET™, DrBlade™, EasyPIM™, EconoBRIDGE™, EconoDUAL™, EconoPACK™, EconoPIM™, EiceDRIVER™, eupec™, FCOS™, GaNpowIR™, HEXFET™, HITFET™, HybridPACK™, iMOTION™, IRAM™, ISOFACE™, IsoPACK™, LEDrivIR™, LITIX™, MIPAQ™, ModSTACK™, my-d™, NovalithIC™, OPTIGA™, OptiMOS™, ORIGA™, PowIRaudio™, PowIRStage™, PrimePACK™, PrimeSTACK™, PROFET™, PRO-SIL™, RASIC™, REAL3™, SmartLEWIS™, SOLID FLASH™, SPOC™, StrongIRFET™, SupIRBuck™, TEMPFET™, TRENCHSTOP™, TriCore™, UHVIC™, XHP™, XMC™

Trademarks updated November 2015

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

Edition <2016-01-11>

Published by
Infineon Technologies AG
81726 Munich, Germany

© 2016 Infineon Technologies AG.
All Rights Reserved.

Do you have a question about this document?

Email: erratum@infineon.com

Document reference

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 product or any use thereof can reasonably be expected to result in personal injury.