Recommendations for Printed Circuit Board Assembly of Infineon xWLy Packages

Additional Information

DS2, August 2009

Edition 2009-08

Published by
Infineon Technologies AG
81726 München, Germany

© 2009 Infineon Technologies AG
All Rights Reserved.

Legal Disclaimer

The information given in this document shall in no event be regarded as a guarantee of conditions or characteristics. With respect to any examples or hints given herein, any typical values stated herein and/or any information regarding the application of the device, 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.

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

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)
- No backside protection
- For products with up to 25 I/Os
- Max. size of 2.6 mm x 2.6 mm x 0.65 mm

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)
- Backside protection (optional)
- For products with up to 200 I/Os
- Max. size of 7.0 mm x 7.0 mm

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)
- For products with up to 400 I/Os
- Max. size of 10.0 mm x 10.0 mm

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

- Thinnest packages possible
- Applicable for a wide range of chip sizes
- Flexible rerouting (WLB)
- Applicable for packages with low to medium number of I/Os
- Various ball pitches 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.

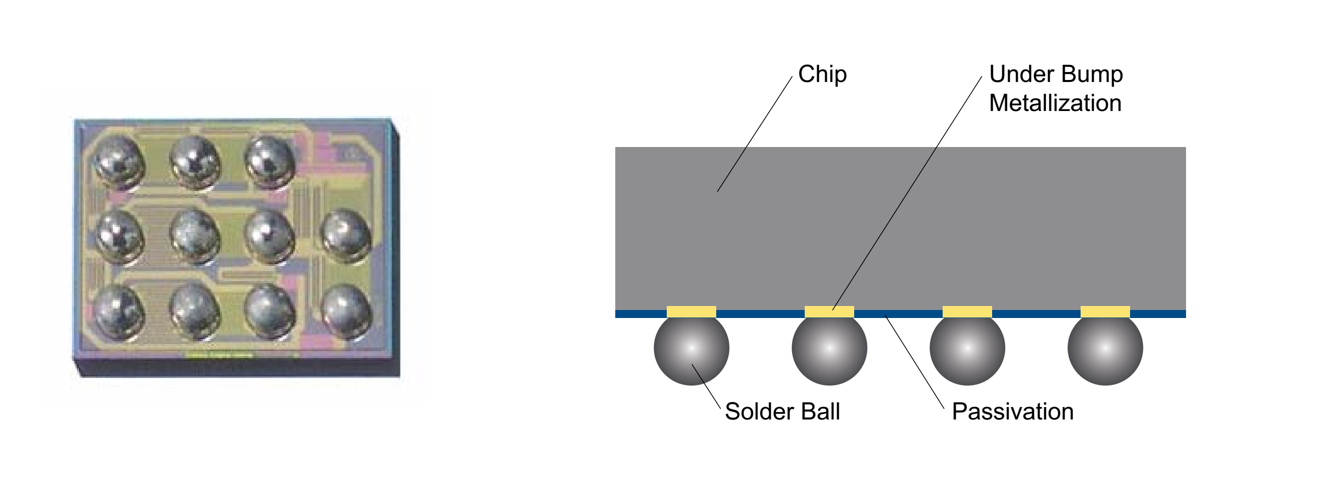

Figure 1 Photo and Schematic Cross Section of Typical SG-WLP

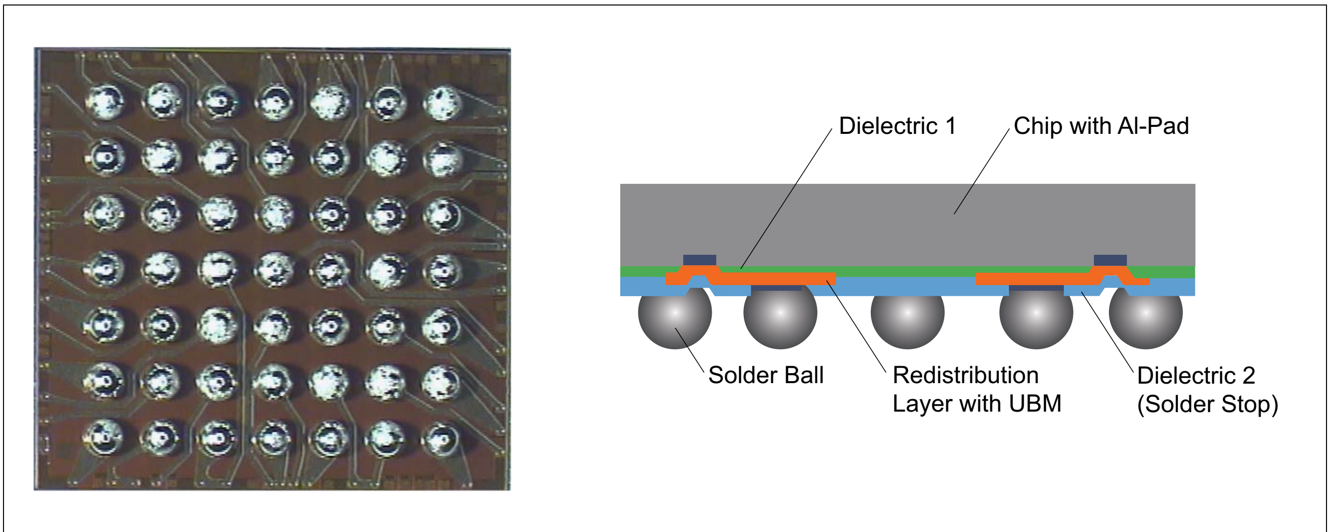

Figure 2 Photo and Schematic Cross Section of Typical SG-VFWLB, SG-WFWLB or SG-UFWLB

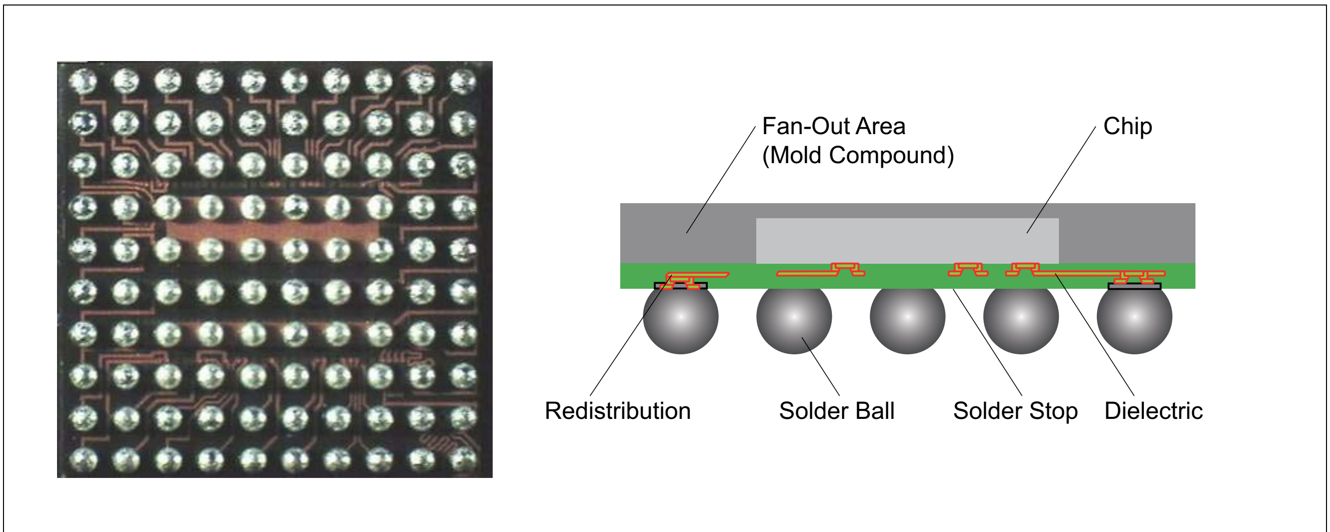

Figure 3 Photo and Schematic Cross Section of Typical PG-WFWLB

2 Package Handling

2.1 ESD Protective Measurement

Semiconductor devices are normally Electrostatic Discharge Sensitive Devices (ESDSs) requiring specific precautionary measures regarding handling and processing. Discharging of electrostatically charged objects over an Integrated Circuit (IC) can be caused by human touch or by processing tools, resulting in high current and/or high voltage pulses that can damage or even destroy sensitive semiconductor structures. On the other hand, ICs may also be charged during processing. If discharging takes place too quickly ("hard" discharge), it may cause load pulses and damage, too. ESD protective measures must therefore prevent contact with charged parts as well as electrostatic charging of the ICs. Protective measures against ESD must be taken during handling, processing, and the packing of ESDS. A few hints are provided below on handling and processing.

2.1.1 ESD Protective Measures in the Workplace

- Standard marking of ESD protected areas
- Access controls, with wrist strap and footwear testers
- Air conditioning
- Dissipative and grounded floor
- Dissipative and grounded working and storage areas
- Dissipative chairs
- Earth ("ground") bonding points for wrist straps
- Trolleys or carts with dissipative surfaces and wheels
- Suitable shipping and storage containers
- No sources of electrostatic fields

2.1.2 Equipment for Personnel

- Dissipative/conductive footwear or heel straps
- Suitable smocks
- Wrist straps with safety resistors
- Gloves or finger cots that are ESD-tested (with specified volume resistivity)

Regular training of staff to avoid ESD failures using this equipment is recommended.

2.1.3 Production Installations and Processing Tools

- Machine and tool parts made of dissipative or metallic materials
- No materials having thin insulating layers for sliding tracks
- All parts reliably connected to ground potential
- No potential difference between individual machine and tool parts
- No sources of electrostatic fields

Detailed information on ESD protective measures may be obtained from the ESD Specialist through Area Sales Offices. Our recommendations are based on the internationally applicable standards IEC 61340-5-1 and ANSI/ESD S2020.

2.2 Packing of Components

Different packings for feeding components in an automatic pick-and-place machine (tape-and-reel, trays,…) and surrounding bags and boxes to prevent damage during transportation or storage are available, depending on component and customer needs. Please refer to product and package specifications (on the IFX homepage) and our sales department to get information about what packing is available for a given product.

The following standards dealing with packing may be applicable for a given package and packing; IFX packs according to the IEC 60286-* series:

IEC 60286-3 Packaging of components for automatic handling - Part 3:
Packaging of Surface-Mount Devices (SMDs) or components on continuous tapes

IEC 60286-4 Packaging of components for automatic handling - Part 4:
Stick magazines for dual-in-line packages

IEC 60286-5 Packaging of components for automatic handling - Part 5:
Matrix trays

Moisture Sensitive SMDs are packed according to IPC/JEDEC J-STD-033*:
Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices

Detailed packing drawings: Packing Information (Internet)

Other References:

ANSI/EIA-481-* Standards Proposal No. 5048, Proposed Revision of ANSI/EIA-481-B 8 mm through 200 mm Embossed Carrier Taping and 8 mm & 12 mm Punched Carrier Taping of Surface Mount Components for Automatic Handling (if approved, to be published as ANSI/EIA-481-C)

EIA-783 Guideline Orientation Standard for Multi-Connection Package (Design Rules for Tape and Reel Orientation)

2.3 Moisture Sensitive Components (MSL Classification)

For moisture-sensitive packages, it is necessary to control the moisture content of the components. Penetration of moisture into the package molding compound is generally caused by exposure to ambient air. In many cases, moisture absorption leads to moisture concentrations in the component that are high enough to damage the package during the reflow process. Thus it is necessary to dry moisture-sensitive components, seal them in a Moisture-Barrier Bag (MBB), and only remove them immediately prior to board assembly to the PCB. The permissible time (from opening the MBB until the final soldering process) during which a component can remain outside the MBB is a measure of the sensitivity of the component to ambient humidity (Moisture Sensitivity Level, MSL). The most commonly applied standard IPC/JEDEC J-STD-033* defines eight different MSLs (see **Table 1**). Please refer to the “Moisture Sensitivity Caution Label” on the packing material, which contains information about the MSL of our products. IPC/JEDEC-J-STD-20 specifies the maximum reflow temperature that shall not be exceeded during board assembly at the customer’s facility.

If moisture-sensitive components have been exposed to ambient air for longer than the specified time according to their MSL, or the humidity indicator card indicates too much moisture after opening an MBB, the components have to be baked prior to the assembly process. Please refer to IPC/JEDEC J-STD-033* for details. Baking a package too often can cause solderability problems due to oxidation and/or intermetallic growth. In addition, packing material (e.g. trays, tubes, reels, and tapes) may not withstand higher baking temperatures. Please refer to imprints/labels on the product packing to determine allowable maximum temperature.

Table 1 Moisture Sensitivity Levels (according to IPC/JEDEC J-STD-033*)

Level	Floor Life (out of MBB)	
	Time	Conditions
1	Unlimited	≤ 30°C / 85% RH
2	1 year	≤ 30°C / 60% RH
2a	4 weeks	≤ 30°C / 60% RH
3	168 hours	≤ 30°C / 60% RH
4	72 hours	≤ 30°C / 60% RH
5	48 hours	≤ 30°C / 60% RH
5a	24 hours	≤ 30°C / 60% RH
6	Mandatory bake before use. After bake, must be reflowed within the time limit specified on the label.	≤ 30°C / 60% RH

2.4 Storage and Transportation Conditions

Improper transportation and unsuitable storage of components can lead to a number of problems during subsequent processing, such as poor solderability, delamination, and package-cracking effects.

These standards should be taken into account:

IEC 60721-3-0	Classification of environmental conditions: Part 3: Classification of groups of environmental parameters and their severities; introduction
IEC 60721-3-1	Classification of environmental conditions: Part 3: Classification of groups of environmental parameters and their severities; Section 1: Storage
IEC 60721-3-2	Classification of environmental conditions: Part 3: Classification of groups of environmental parameters and their severities; Section 2: Transportation
IEC 61760-2	Surface mounting technology - Part 2: Transportation and storage conditions of surface mounting devices (SMD) - Application guide
IEC 62258-3	Semiconductor Die Products - Part 3: Recommendations for good practise in handling, packing and storage
ISO 14644-1	Clean rooms and associated controlled environments Part 1: Classification of airborne particulates

Table 2 Overview over General Storage Conditions

Product	Condition for Storing
Wafer/Die	N2 or MBB (IEC 62258-3)
Component - moisture sensitive	MBB[1)] (JEDEC J-STD-033*)
Component - not moisture sensitive	1K2 (IEC 60721-3-1)

1) MBB = Moisture Barrier Bag

Maximum storage time:

The conditions to be complied with in order to ensure problem-free processing of active and passive components are described in standard IEC 61760-2.

Internet links to standards institutes:

American National Standards Institute (ANSI)

Electronics Industries Alliance (EIA)

Association Connecting Electronics Industries (IPC)

2.5 Handling Damage and Contaminations

Automatic or manual handling of components in or out of the component packing may cause mechanical damage to package balls and/or body.

Any contamination applied to component 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 conductive particles)

2.6 Component Solderability

The solder balls of Infineon's xWLy package and their packings assure good solderability, even after a long storage time. Suitable methods for the assessment of solderability are described in JESD22B 102 or IEC6068-2-58.

Even if wetting of the balls with the given solder paste occurs at temperatures below the melting point of the solder ball alloy, it is necessary to raise the temperature of the solder paste and solder ball at least 15 Kelvin above the melting point to form a reliable solder joint.

3 Printed Circuit Board

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

Typically, fineline PCBs with conductor width/spacing of 75 to 100 µm are necessary for routing for devices with solder ball pitches of 0.4 mm or 0.5 mm. The details of the PCB design strongly depend on the board technology (conventional technology with drilled vias vs. build-up technology with microvias), the conductor width/spacing, number of metal layers, and electrical restrictions.

Figure 6 shows examples of routing for xWLy packages with pitches of 400 µm and 500 µm using a 100-µm conductor path width and micro vias (100-µm diameter) vs. “big vias” (200-µm diameter).

3.2 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 (**Figure 4**): 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.

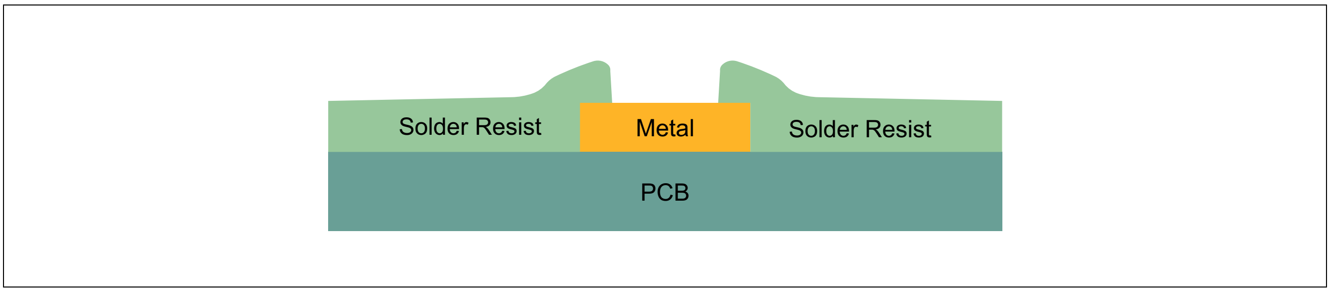

Figure 4 SMD Pad

- Non-Solder-Mask Defined (NSMD) pad (**Figure 5**): 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 no overlapping of the solder pad by solder mask occurs (depending on PCB manufacturers’ tolerances, 75 µm is a widely used value).

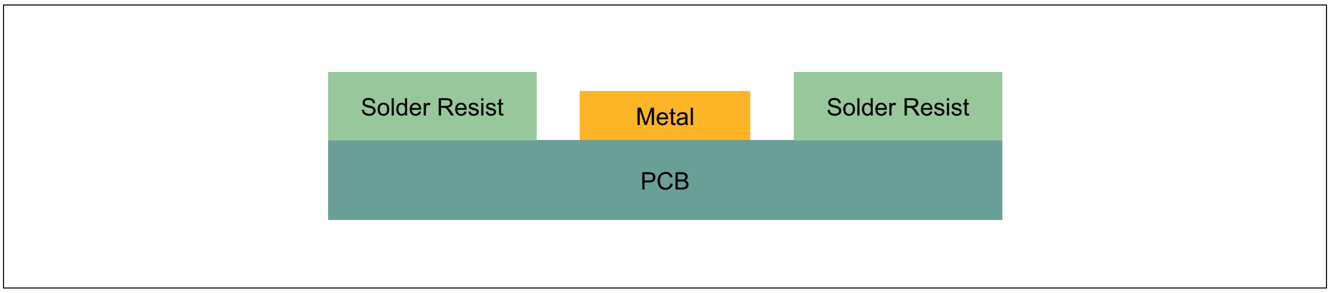

Figure 5 NSMD Pad

NSMD pads provide more space for routing and result in a higher solder joint reliability, and the side walls of the lands are wetted by the solder, which results in less stress concentration. Therefore, NSMD pads are recommended for the solder pads on the PCB.

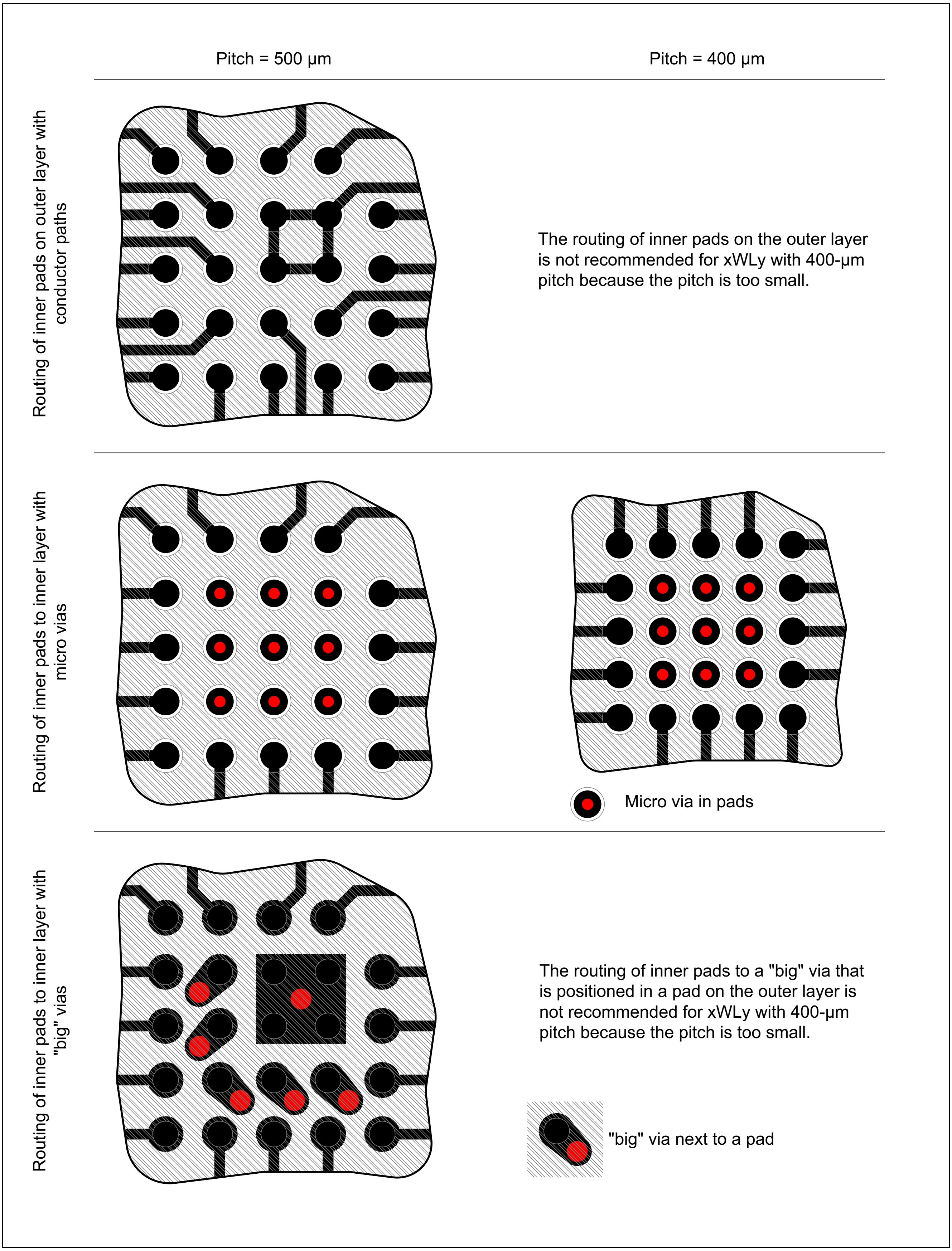

Figure 6 Different Types of Board Pad Layouts and Routing for 500- and 400-µm Ball Pitch

Figure 7 shows the recommended PCB pad design. Refer to Table 3 for the appropriate dimensions, which are ball-pitch specific. The values in the table are typical dimensions. The details depend on the PCB technology used, capability of the suppliers, and the planned 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.

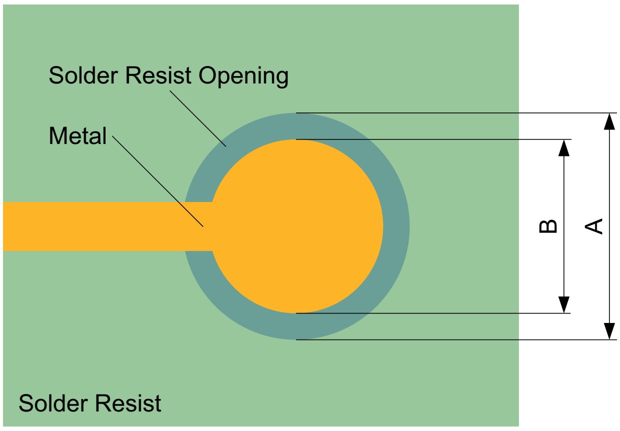

Figure 7 PCB Pad Design Recommendation (NSMD Pad)

Table 3 PCB Pad Dimensions

Ball Pitch [mm]	**0.40**	**0.50**
Solder resist opening diameter A [µm]	typ. 350	350 - 430
Metal pad diameter B [µm]	250	250 - 280

3.3 PCB Pad Finishes

The solder pads have to be easy for the solder paste to wet. In general, all finishes are well-proven for SMT assembly, but the quality of the plating/finish is more important for fine-pitch applications in particular. Because of the uneven surface of Hot Air Solder Leveling (HASL) finish, Pb-free or Pb-containing HASL is less preferred for assembly (especially for pitches < 0.65 mm) compared to completely “flat” platings such as Cu-OSP (OSP: Organic Solderability Preservative) or electroless Sn or NiAu.

From a package point of view, it is difficult to recommend a certain PCB pad finish that will always meet all requirements. The choice of finish also depends strongly on board design, pad geometry, all components on the board, and process conditions, and must be chosen accordingly to the specific needs of the customer.

Infineon’s internal tests have shown that Cu-OSP is a suitable and reliable plating.

Table 4 Typical PCB Pad Finishes

Finish	Typ. Layer Thickness [µm]	Properties	Concerns
HASL (SnAg) *(Hot Air Leveling)*	> 5	Low cost, widely usage, know how in fabrication	Uneven surface, formation of humps, flatness of single pads has to be good for fine-pitch applications
Electroless Tin	0.3 - 1.2	Solder joint consists only of copper and solder, no further metal is added to the solder joint	Long-term stability of protection may be a concern, baking of PCB may be critical
Electroless Silver	0.2 - 0.5	Solder joint consists only of copper and solder, no further metal is added to the solder joint	Long-term stability of protection may be a concern, baking of PCB may be critical
Electroless Ni / Immersion Au *(ENIG)*	3 - 7 / 0.05 - 0.15	Good solderability protection, high shear force values	Expensive, concerns about brittle solder joints
Galvanic Ni / Au	> 3 / 0.1 - 2	Only for thicker layers, typically used for connectors	Expensive, not recommended for solder pads
OSP *(Organic Solderability Preservatives)*	1	Low cost, simple, fast and automated fabrication	Must be handled carefully to avoid damaging the OSP; not as good long-term stability as other coatings; in case of double-sided assembly only suitable with inert gas during reflow

4 PCB Assembly

4.1 Solder Stencil

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. For xWLy packages, 100- to 120-µm thick stencils should be used; please be aware of the area-ratio that results in a maximum stencil thickness depending on a given aperture shape. Stencil apertures should be circular. 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 increased 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. An optical inspection or a cleaning cycle (wet/dry) of the stencil after each printed board is advisable.

Table 5 Recommendations for Stencil Dimensions

Package	Stencil Aperture Diameter [µm]	Stencil Thickness [µm]
xWLy with ball pitch 0.5 mm	250 - 300	100 - 120
xWLy with ball pitch 0.4 mm	250 - 260	100

4.2 Solder Paste

Solder paste consists of solder alloy and a flux system. Normally the volume is split into about 50 Vol% alloy and 50 Vol% flux. In term of mass, this means approx. 90 wt% alloy and 10 wt% flux system. The flux system is intended to remove contamination from the solder joints during the soldering process. The capability of removing contamination is given by the respective activation level. The solder paste metal alloy has to be of Pb-containing eutectic or near-eutectic composition (SnPb or SnPbAg) or Pb-free composition (SnAgCu whereas Ag 3 - 4%, Cu 0.5 - 1%). A “no-clean” solder paste is preferred because cleaning under the soldered xWLy packages may be difficult. The paste must be suitable for printing the solder stencil aperture dimensions. Generally, Type 3 is used for pitches greater than 0.5 mm, but in Infineon’s internal testing has proven to be suitable for the recommended stencil apertures of xWLy packages. Nevertheless Type 4 paste may help to get better solder paste release at printing process during mass production.

Solder paste is sensitive to age, temperature and humidity. Please comply with the handling recommendations of the paste manufacturer.

4.3 Component Placement

xWLy packages have to be placed accurately depending on their geometry. Positioning the packages manually is not recommended.

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). They 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.

If they are slightly displaced, packages with solder balls as in the xWLy packages have the favorable property of self-alignment during the reflow process due to the solder's high surface tension. As a rule-of-thumb, a maximum tolerable displacement of the components is 30% of the metal pad diameter on the PCB (for NSMD pads). Consequently, for xWLy packages the solder ball to PCB pad misalignment has to be less than 75 µm to assure a robust mounting process. Generally this is achievable with a wide range of placement systems.

The following factors are important:

- Especially on large boards, local fiducials close to the device can compensate for large PCB tolerances
- The ball-recognition capabilities of the placement system should be used, not the outline centering.
- To ensure the identification of the packages by the vision system, adequate lighting as well as the correct choice of the measuring modes are necessary. The right settings are described in the equipment manuals.
- Too much placement force can squeeze out solder paste and cause solder-joint shorts. On the other hand, not enough placement force can lead to insufficient contact between package and solder paste, which can lead to open solder joints or badly centered packages.

Tools for Pick & Place

Pick & place of packages is typically done with vacuum nozzles. For picking up large (> 3 mm x 3 mm) xWLy packages, the following recommendation have to be taken into account:

- Rubber vacuum nozzle tips shall be used for the pick & place process
- Metal & ceramic tools can create scratches on the die surface and have to be avoided
- Tool diameter has to be smaller as the die dimension (prevent edge damage of the die)
- Scrubbing on the surface of the die with any tools shall be avoided
- Manual handling by operator shall be done only by specially instructed personnel

Bending Stress

To avoid the risk of damaging the WLB device, some rules shall be followed to minimize bending stress. In particular, local stress tension forces on the bare die device shall be avoided during all handling steps at the customer:

- Stress conditions as shown in **Figure 8** Case 1, which cause a convex bending of the device with tension stress on the WLB device shall be avoided. (Such stress conditions could occur e.g. during electrical test, when the forces on top and bottom of the package are applied by probe pins (pogo pins)).
- To bypass stress conditions as shown in **Figure 8** Case 1, measures shall be taken as shown in **Figure 8** Case 2. If a force is applied to the center of the device bottom side, the device shall be supported at the opposite side to provide a central counterforce and to prevent package bending.

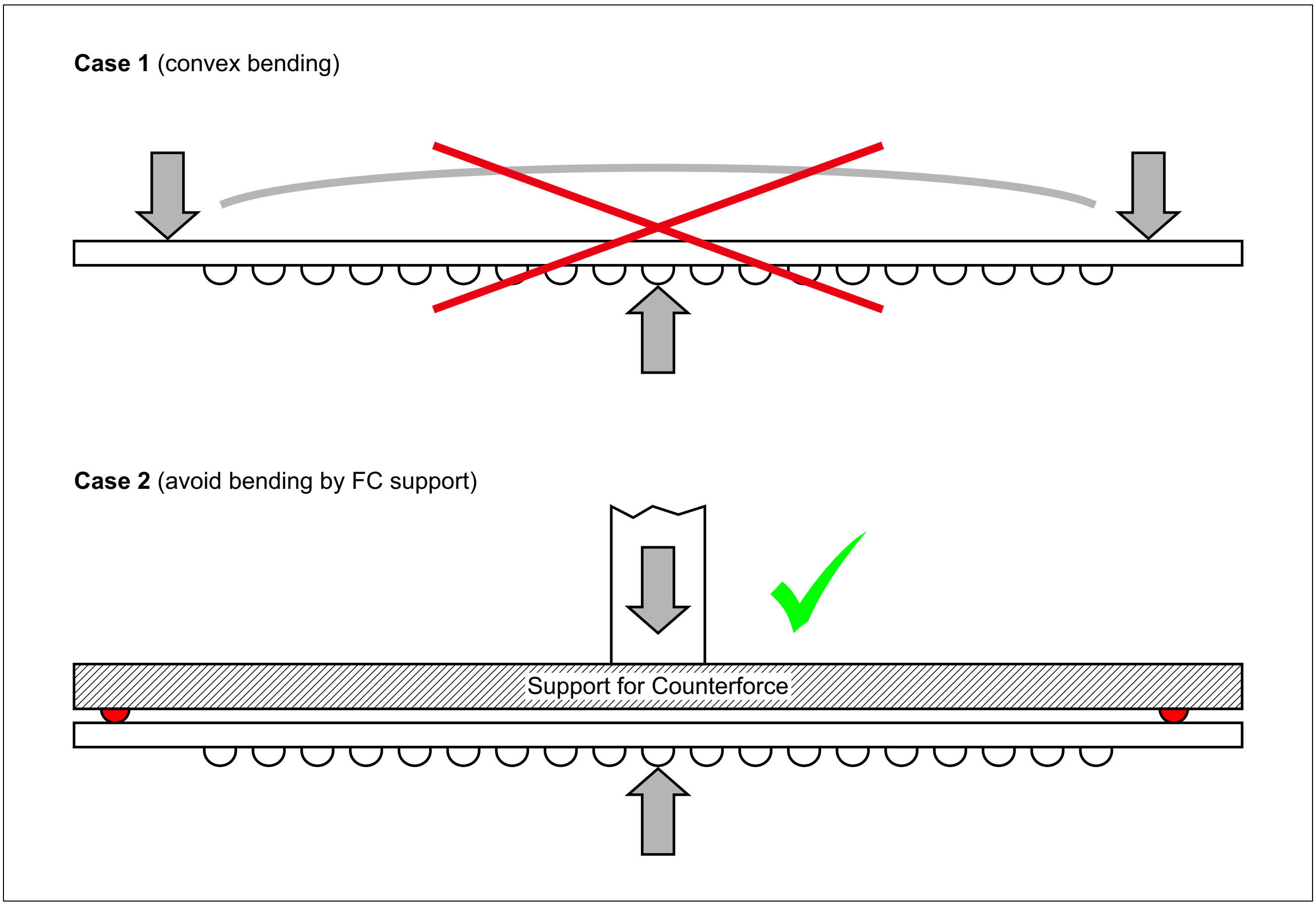

Figure 8 Case 1 shows bending stress applied on WLB by insuccient support. Case 2 shows a nozzle supporting the package and applying a counterforce.

4.4 Soldering

Soldering determines the yield and quality of assembly fabrication to a very large extent. Generally all standard reflow soldering processes such as:

- Forced convection
- Vapour phase
- Infrared (with restrictions)

and typical temperature profiles are suitable for board assembly of the xWLy packages. Wave soldering is not possible. During the reflow process, each solder joint has to be exposed to temperatures above solder melting point (liquidus) for a sufficient time to get the optimum solder-joint quality, whereas overheating the PCB with its components has to be avoided. Please refer to the bar code label on the packing for the 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, including the whole ball matrix underneath the xWLy packages. The most highly recommended type of soldering is forced-convection reflow. Using a nitrogen atmosphere can generally improve solder-joint quality, but is normally not necessary for soldering SnPb metal alloys. For Pb-free alloys nitrogen may contribute to shiny and oxid-free solder-joints.

The temperature profile of a reflow process is one of the most important factors of the soldering process. The temporal progression of the temperature profile is divided into several phases, each with a special function. **Figure 9** shows a general forced convection reflow profile for soldering xWLy packages. **Table 6** shows an example of the key data for such a solder profile for SnPb and for Pb-free alloys. 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, too. Additionally, most PCBs have more than one package type mounted on them, and therefore the reflow profile has to be matched to all components' and materials' demands. We recommend measuring the solder joints' temperatures by using thermocouples beneath the respective packages. Note that components with large thermal masses don't heat up at the same speed as lightweight components, and the position and the surrounding of the package on the PCB, as well as the PCB thickness can influence the solder-joint temperature significantly. Therefore no concrete temperature profile can be given.

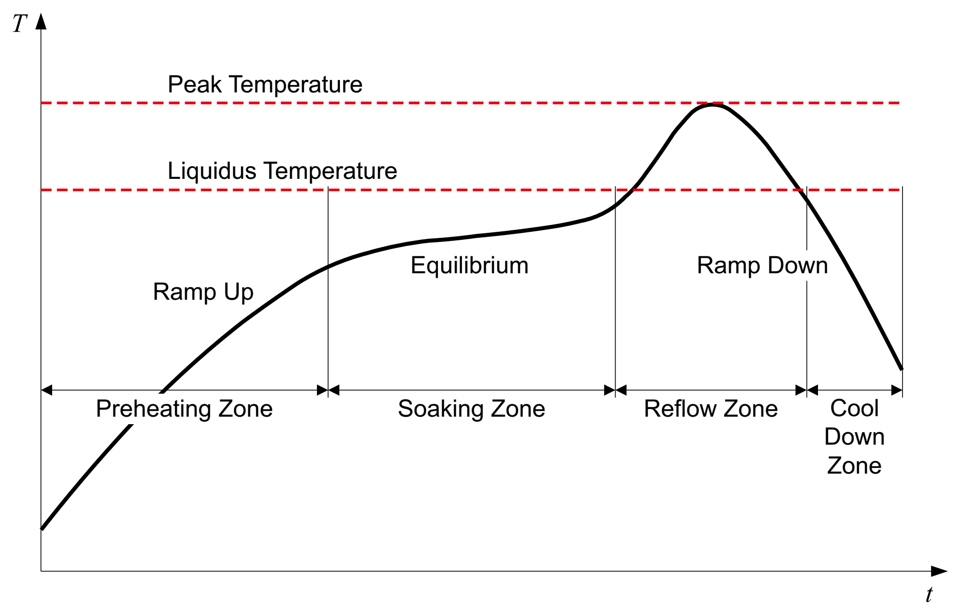

Figure 9 General Forced Convection Reflow Solder Profile

The following table is an example, not a recommendation (for reference only).

Table 6 Example of the Key Data for a Forced Convection Reflow Solder Profile

Parameter	Pb-containing Alloy (SnPb or SnPbAg)	Pb-free Alloy (SnAgCu)	Main Requirements come from
Preheating rate	2.5 K/s	2.5 K/s	Flux system (Solder paste)
Soaking temperature	140 - 170°C	140 - 170°C	Flux system (Solder paste)
Soaking time	80 s	80 s	Flux system (Solder paste)
Peak temperature	225°C	245°C	Alloy (Solder paste)
Reflow time over Liquidus	60 s	60 s	Alloy (Solder paste)
Cool down rate	2.5 K/s	2.5 K/s	

4.4.1 Double-Sided Assembly

xWLy packages are generally suitable for mounting on double-sided PCBs. First, one side of the PCB is fitted with components and soldered. Afterwards, the second side of the PCB is fitted 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 air draft 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.

4.4.2 Compatibility of Solder Alloys

Due to the possible usage of different solder alloys for package balls and solder paste printed on the PCB, the compatibility of these alloys has to be taken into account.

Generally, soldering of xWLy packages with a typical reflow profile normally used for Pb-containing solder paste is not recommended. The solder sphere does not completely melt, which may result in reduced reliability. Therefore, the recommended peak temperature is at least 235°C.

4.5 Cleaning and Subsequent Processing

4.5.1 Cleaning

After the reflow soldering process, some flux residues can be found around the solder joints. If a “no-clean” solder paste has been used for solder-paste printing, the flux residues usually don’t have to be removed after the soldering process. Be aware that cleaning beneath a xWLy package is difficult because of the small gap between package substrate and PCB and is therefore not recommended. Whether or not the solder joints have to be cleaned, however, the cleaning method chosen (e.g. ultrasonic, spray, or vapor cleaning) and cleaning solution have to be selected with consideration of the packages to be cleaned, the flux used in the solder paste (rosin-based, water-soluble, etc.), and environmental and safety aspects. Removing and drying even small residues of the cleaning solution should also be done very thoroughly. Contact the solder-paste manufacturer for recommended cleaning solutions.

4.5.2 Underfilling

Infineon Technologies performed reliability tests on testboard level (e.g. drop test, temperature cycling on board, bend test) without underfilling showing positive results passing typical market requirements.

Based on these results additional underfilling is not necessary for xWLy packages, and we strongly recommend not to use underfill as this may deteriorate the reliability performance of the package construction. Infineon can not assure the reliability assessment of various underfill materials available on the market. Thus it is under customer's responsibility to investigate the reliability performance of the resulting package configuration even though underfill will be used.

The following aspects have to be taken into account if underfill is intended to be used for Wafer Level Packages:

- **Reliability:** Usually the application of underfill material under the package could be a method to improve the board-level reliability with respect to mechanical and/or thermo-mechanical stress, especially for very big packages with small ball pitches. But IFX-internal reliability tests on board using different underfills on xWLY packages showed, that specific underfill materials can be a trade-off of Droptest performance, TCoB performance and reworkability.
- **Rework/Repair:** A distinguishing factor for underfill material selection is the material's capability to be removed from the PCB during rework.
  A typical class of underfill materials is duroplastic. Such an underfill therefore can not be removed without applying chemicals or very high temperatures - both exceeding the stability and reliability of PCB and component. Another class are so-called reworkable underfills. They typically get soft at high temperatures and the adhesive forces at or near to the reflow temperature are weak enough to clean the material away from the PCB. Nevertheless such materials stick very hard to a component and the applied temperatures and mechanical forces are very high.

Note: It is not possible to rework xWLy packages after desoldering from the PCB - neither underfilled with duroplastic or reworkable underfill, nor not underfilled at all!

Table 7 Material properties and the impact on TCoB performance, droptest reliability and reworkability of three different types of underfill materials tested by Infineon Technologies

	Underfill Type 1 "Duroplastic"	**Underfill Type 2 "Reworkable"**	**Underfill Type 3 "Reworkable"**
Glass transistion (Tg) [°C]	150	60	35
CTE below Tg [ppm]	30	55	69
CTE above Tg [ppm]	100	192	215
Impact on TCoB performance[1)]	increased	decreased	decreased
Impact on Droptest performance[1)]	increased	increased	increased
Reuse of desoldered xWLy components	no	no	no
Reuse of PCB	no	yes	yes

1) Compared to reliability without underfill application

- **Underfilling Process:** A liquid epoxy resin is dispensed on the PCB next to the package outline at a distance that allows the material to wet the gap between package and board. Through capillary action, the underfiller is then driven into the gap and fills it completely.
  Please take care that during application of the underfill material the xWLy package is not touched by the underfilling tool, which may cause mechanical damage.
  The process parameters depend on the underfill material, the package size, ball matrix, and design of the PCB. To achieve shorter flow times, the substrate is usually heated up to 50°C to 90°C during dispensing to reduce the fluid viscosity. For small package sizes (approx. < 40 mm^2), it is sufficient to deposit the underfiller only on one side in a line-shaped dispense move (length approx. 75% of package side). For larger packages, it is recommended to dispense an L-shaped pattern on two adjacent sides. For very large packages, it may be necessary to dispense an additional second L-shape after the earlier dispensed material has spread out completely to achieve a complete filling. U-shaped patterns are an alternative for large packages but increase the risk for entrapping air by colliding flow fronts.

Typically filled or unfilled epoxy resins are used in a liquid state for package underfilling, and are hardened afterwards. Ultra-fine fillers are not needed, and it is possible to use low-cost materials with larger filler particles (max. filler size should be 1/3 of the gap).

4.6 Inspection

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 10 shows a typical X-ray image of a soldered SG-WLP-25-1 package to a 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 Ball Grid Arrays (BGAs) of 25% maximum voiding rate (for X-ray inspection top-down 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.

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. During introduction of new products into SMT production, we recommend these methods to assess the soldering quality in detail.

Please be aware that 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 also comparable to the process used to inspect and analyze a BGA.

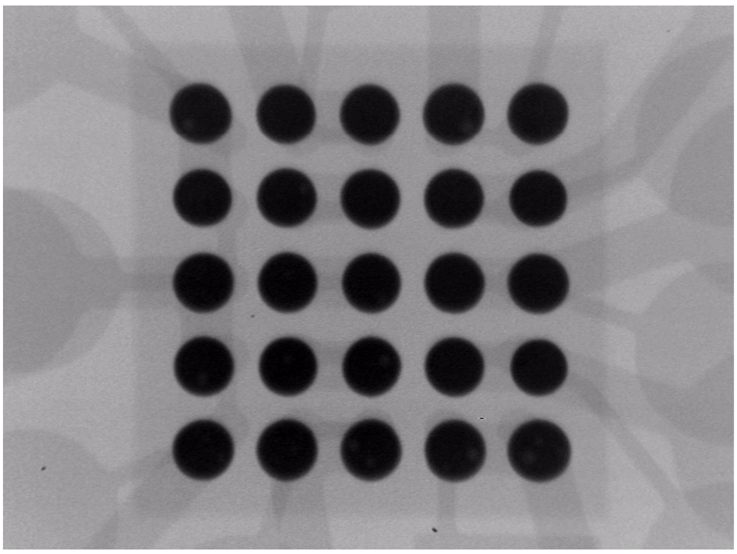

Figure 10 Typical X-ray Image of Soldered SG-WLP-25-1

5 Rework

Talking about rework in case of Wafer Level Packages three different failure modes and derived process flows have to be distinguished.

1. The PCB is expected to be defective:
   In this case it is inevitable to discard the whole assembly.
2. Solder joint(s) of a xWLy component is expected to be defective:
   A desoldering and discarding of the component is necessary.
   Repair of components' single solder joints is not possible. Reusing of a desoldered component is not possible.
   The desoldered component must be replaced by a new one.
3. The Wafer Level Package is expected to be defective:
   Reusing of a desoldered component is not possible. A desoldering and discarding of the component is necessary.
   The desoldered component must be replaced by a new one.

If a component is suspected to be defective and the component supplier shall perform a failure analysis with this component, please do not remove this component from the PCB, but send the entire PCB with components to Infineon Technologies. This guarantees that no further defects are introduced to the component by removal from PCB, because this may hinder the failure analysis at the supplier's facility or make the failure analysis even impossible. This is valid for underfilled components as well as for not underfilled components!

If an underfill is used which is not labelled as "reworkable", repair of PCB and/or components is not possible. A failure analysis at IFX is possible when the component together with the PCB board is sent to IFX.

In case of using underfill labelled as "reworkable" it is possible to reuse a non-defective PCB (case 2 and 3). For details please refer to the information provided by the underfill or rework station supplier. A failure analysis at IFX is possible when the component together with the PCB board is sent to IFX.

Please note that damage caused by mechanically removing underfill from PCB may result in reduced reliability.

5.1 Tooling

The rework process is commonly done on special rework equipment. There are a lot of systems available on the market, and the equipment should fulfill the following requirements for processing these 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 freely-programmable temperature profiles (e.g. by PC controller) it is possible to adapt the profiles to different package sizes and masses. PCB preheating from the underside is recommended. Infrared heating can be applied, especially for preheating the PCB from the underside, but it should be only augmenting the hot air flow from the upper 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. 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).

5.2 Device Removal

For removing a component and replace it by a new one (case 2 and 3), please follow these precautions. Do not repair the removed component and do not place it back to the PCB!

- *Moisture*: Depending on the moisture sensitivity of PCB and neighbored components, the assembly may have to be dried before desoldering components. Otherwise, too much moisture may have been accumulated and damage may occur (popcorn effect).
- *Temperature profile*: During the soldering process, it should be assured that peak temperature is not higher and temperature ramps are not steeper than for the standard assembly reflow process to avoid thermal or thermo-mechanical damage of the PCB and surrounding components (see **Chapter 4.4**).
- *Mechanics*: Be careful not to apply high mechanical forces for removal, which may result in PCB damage.

Note:If the xWLy component has been removed from the PCB board by the PCB assembly house the component supplier (IFX) can not guarantee that a failure analysis of the removed component can be performed by Infineon Technologies. If failure analysis of defective components by Infineon Technologies is required please send the complete PCB including the defective component to Infineon Technologies.

5.3 Site Redressing

After removing the defective component, the pads on the PCB have to be cleaned of solder residues.

This may be done by vacuum desoldering or wick.

If the component was underfilled, the remaining underfill on the PCB also has to be removed. Some solvents may be necessary to clean the PCB of flux residues (and underfill residues if applicable).

Don't use steel brushes because steel residues can lead to bad solder joints. In all cases, harsh mechanical treatment may damage PCB pads and conductors.

Before placing a new component on the PCB, solder paste should be applied to each PCB pad by printing (special micro stencil) or dispensing. Another method that may lead to a decreased solder stand-off compared to non-repaired components is to apply flux only by dispensing or with a brush (often so-falled sticky flux is used for this purpose). Only no-clean flux and solder paste should be used.

5.4 Reassembly and Reflow

After preparing the site, the new package can be placed onto the PCB. Placement accuracy and placement force should be comparable to the automatic pick-and-place process. During soldering, it should be assured that the package peak temperature is not higher and temperature ramps are not steeper than for the standard assembly reflow process.

www.infineon.com

Published by Infineon Technologies AG