Recommendations for Printed Circuit Board Assembly of Infineon WLL Packages

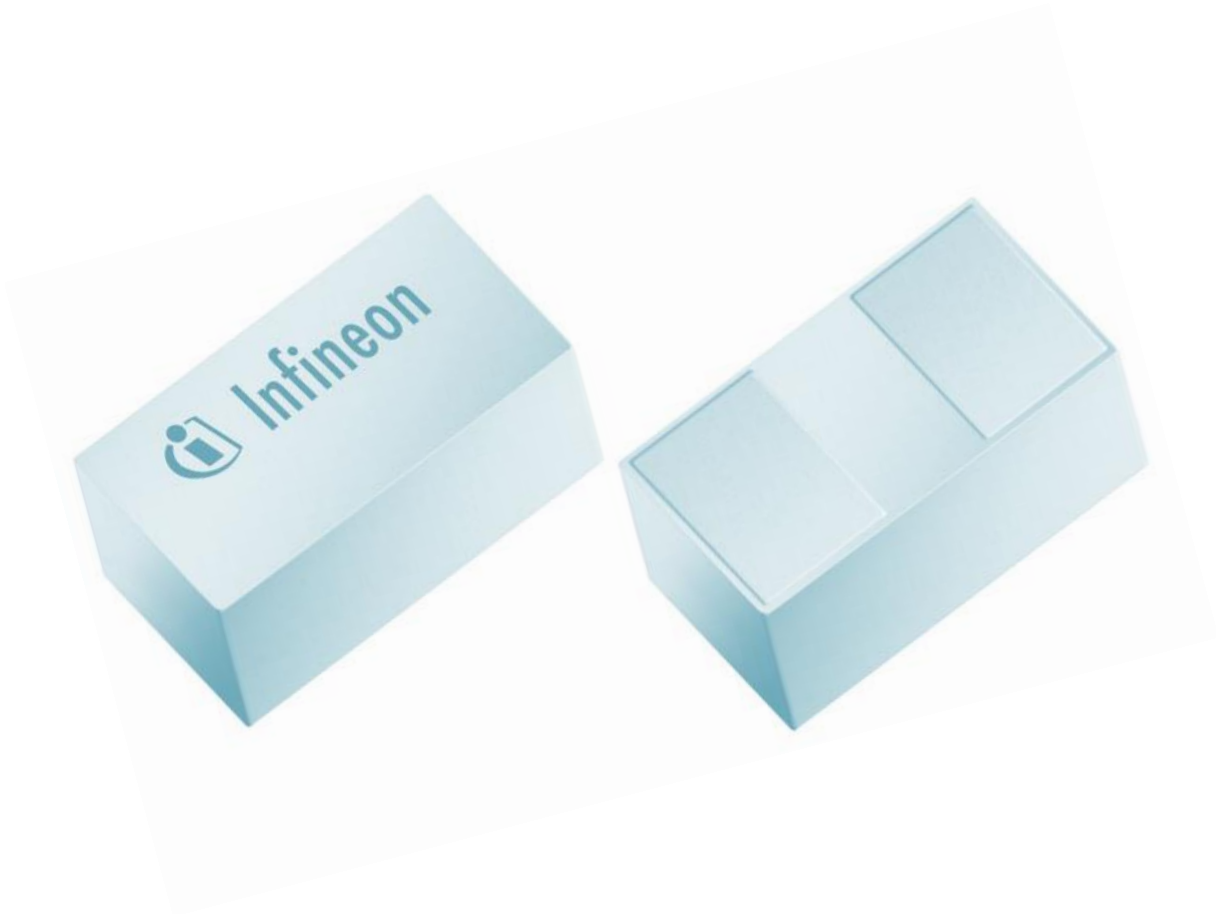

Table of Contents

Table of Contents 2
1 Package Description 3
2 Printed Circuit Board (PCB) 4
2.1 Routing 4
2.2 PCB Pad Design 4
3 Board Assembly 5
3.1 General Remarks 5
3.2 Solder Paste 5
3.3 Solder Stencil 5
3.4 Component Placement 5
3.5 Reflow Soldering 6
4 Cleaning 8
5 Inspection 9
6 Rework 11
Revision History 12

1 Package Description

Infineon's Silicon Green - Wafer Level Leadless (SG-WLL) packages are bare silicon packages for discrete components (. They are especially appropriate for miniaturization in products with limited space on the PCB (Printed Circuit Board). For package board interconnection, the pads have an NiP – Pd - Au (Nickel-Phosphorus Palladium Gold) surface. The remaining surface on the active side of the product is covered by silicon nitride (see Figure 1).

Features:

- Smallest x-y-z-package dimensions
  (imperial code 0201 = metric size 0603, imperial code 01005 = metric size 0402)
- Chip-sized (silicon) package without redistribution layer
- MSL1 due to bare silicon product
- Pb-free package
- No package internal interconnect (e.g. wire bond or flip chip connection)

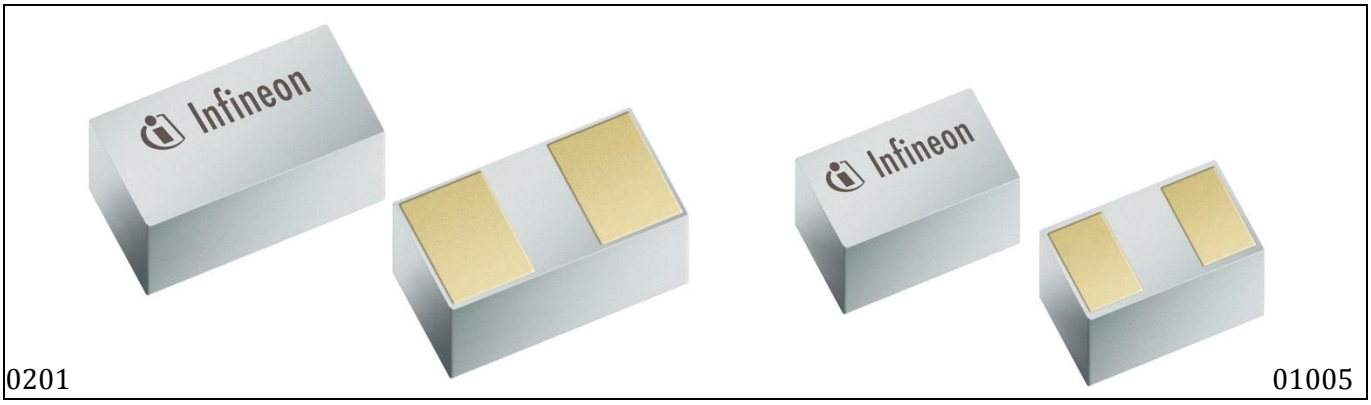

Figure 1 SG-WLL packages

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.

2 Printed Circuit Board (PCB)

2.1 Routing

PCB design and stack-up are key factors for achieving highly reliable solder joints. For example, board stiffness has a significant influence on the reliability (temperature cycling) of the solder-joint interconnect if the application is used in critical temperature-cycling conditions. Board stiffness can also be influenced by placement of large components on a double-sided assembled PCB.

2.2 PCB Pad Design

The interconnect solder joint-to-board is influenced by:

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

Further information can be found in the General Recommendations for Assembly of Infineon Packages by following the link to the section “Printed Circuit Board” at www.infineon.com/packages.

Generally it is recommended to use NSMD pad design for SG-WLL packages. This offers the advantage of eliminating solder-resist tolerances on the respective footprint. Since the SG-WLL packages and their footprints are very small, typical tolerances of the solder resist are quite high in comparison with the pad dimensions. NSMD pads expose parts of the conductor lines connecting the pads to the remaining circuit because the solder-mask opening is wider than the pad itself. To minimize the influence of the solder wetting these open parts during reflow soldering, the connecting lines on the PCB should be as narrow as possible (100 µm or less). Additionally there should be only one connection per solder pad, preferably symmetrical.

Depending on the capabilities of the PCB manufacturer, it might not be possible to separate two NSMD PCB pads of one SG-WLL package by a solder mask dam. A single solder mask opening for both PCB pads is also suitable for generating the two separate solder joints of the component. Therefore, the solder paste printing process should be well-controlled, especially when it comes to the alignment between stencil design and PCB design.

In general, the size of the PCB pads is a key factor for a stable and reproducible soldering process. It is helpful to increase the PCB pad size slightly compared to the package pad size. However it is strongly recommended to distinguish between PCB footprint designs of passive components (typically resistors or capacitors) and SG-WLL packages. Although they are classified by their outline dimensions in the same way (e.g. 0201, 01005), they have different kinds of component terminations. SG-WLL packages are bottom-terminated components, which means the solderable areas of the component are situated only at the bottom of the component but not on any other side. The PCB pad sizes of passive components and SG-WLL packages require different footprint dimensions to provide the optimum solderable area. The recommended PCB pad designs will help to prevent excessive tilting or tombstoning of the SG-WLL packages. For detailed drawings of each recommended PCB pad design, please look up the respective package at www.infineon.com/packages or contact your Infineon sales representative.

In general, assembling SG-WLL packages is similar to the assembly of Infineon TS(S)LP packages. Both provide bottom-located contact pads, and some of them have very similar or identical footprints (e.g. PG-TSSLP-2-1 vs. SG-WLL-2-1).

3 Board Assembly

3.1 General Remarks

Many factors within the board-assembly process influence assembly yield and board-level reliability. Examples include design and material of the stencil, the solder paste material, solder-paste printing process, component placement, and reflow process. We want to emphasize that this document is just a guideline to support our customers in selection of the appropriate processes and materials. Additionally, optimization studies are necessary at the customer's own facilities in order to take into account the actual PCB, the customer's SMT (Surface Mount Technology)equipment, and product-specific requirements.

3.2 Solder Paste

Solder paste consists of solder alloy and a flux system. Normally the volume is about 50% alloy and 50% flux and solvents. In term of mass, this means approximately 90 wt% alloy and 10 wt% flux 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 relative activation level.

Pb-free solder pastes typically contain SAC305 (3.0 % Ag and 0.5 % Cu) or other so-called SnAgCu (SAC) alloys (typically 1-4% Ag and <1% Cu). The solderable surface of SG-WLL packages is suitable for those alloys.

A "no-clean" solder paste is preferred for SG-WLL packages since the solder joints will be formed below the components, where cleaning will be difficult or impossible. The technology used to manufacture the SG-WLL packages does not require a removal of flux residues from a "no-clean" solder paste.

The selected solder paste needs to be suitable for printing the solder stencil aperture dimensions with respect to the standard rules for printing processes in board assembly technology (aperture ratio, least number of solder spheres per opening). Using paste type 4 or higher (with lower grain size of the solder alloy powder) is recommended.

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

3.3 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 size and the stencil thickness. Too much solder paste will cause solder bridging, whereas too little solder paste can lead to insufficient solder wetting between the contact surfaces. In most cases, the thickness of a stencil has to be matched to the needs of all components on the PCB. For SG-WLL packages, the recommended stencil thickness is ≤ 90 µm for package size 0201 and ≤ 70µm for package size 01005. These values are based on the design recommendations and the recommended area ratio value for stencil printing of 0.66. In general, the area ratio describes feasible stencil thicknesses in relation to the aperture sizes.

For small stencil apertures such as the ones that are needed for SG-WLL packages, the quality of the sidewalls inside the stencil apertures is especially important for a stable and repeatable solder-paste printing process. This should be taken into account when choosing the stencil technology.

Please note that these recommendations are guidelines. The most suitable layout for a specific application depends on all the factors mentioned in this document.

3.4 Component Placement

SG-WLL packages are very thin and therefore provided in very thin embossed plastic carrier tape. For smooth processing of the tape in the pick-and-place machine, the mechanical setup of the feeder should avoid any vibration during the movement of the tape into the machine. The tape in the area where the

components will be picked from the tape should be properly clamped. Additionally, any mechanical supporting parts that touch the tape from the bottom that are mounted in the feeder below the tape should be removed. Such mechanical parts are only useful for paper tape.

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 depending on their geometry.

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 at additional 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. We recommend “teaching” your component vision system to recognize the package pads in addition to the package outline.

The maximum tolerable displacement of the components is 25% of the metal pad width on the PCB. For SG-WLL packages, this means the displacement should be below 50µm.

Additionally, it is necessary to select a suitable nozzle for the pick-and-place process. The nozzle size (outer dimensions) should not exceed the package dimensions. Larger nozzles can lead to pick-up errors while removing the components from the tape.

In general, manual placement or handling of SG-WLL packages should be avoided since bare silicon devices need to be handled with special tools and much care. Scrubbing any side of the components should be avoided as well.

For details about factors influencing the component placement, please refer to the General Recommendations for Assembly of Infineon Packages in “Mounting of SMDs,” available at www.infineon.com/packages.

3.5 Reflow Soldering

Soldering determines the yield and quality of assembly fabrication to a very large extent. Generally all typical temperature profiles and standard reflow soldering processes are suitable for board assembly of SG-WLL packages, including:

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

Wave soldering, however, cannot be used.

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, and overheating the PCB with its components has to be avoided. Because of their size and thickness, SG-WLL packages are qualified to withstand a maximum package body temperature of 260°C (this refers to IPC/ JEDEC J-STD-020).

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. Using a nitrogen atmosphere can generally improve solder-joint quality, especially for Pb-free alloys, where nitrogen may contribute to shiny and oxide-free solder joints.

Board Assembly

The temperature profile of a reflow process is divided into several phases, each with a special function (Figure 2). 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 surroundings of the package on the PCB, as well as the PCB thickness, can influence the solder-joint temperature significantly.

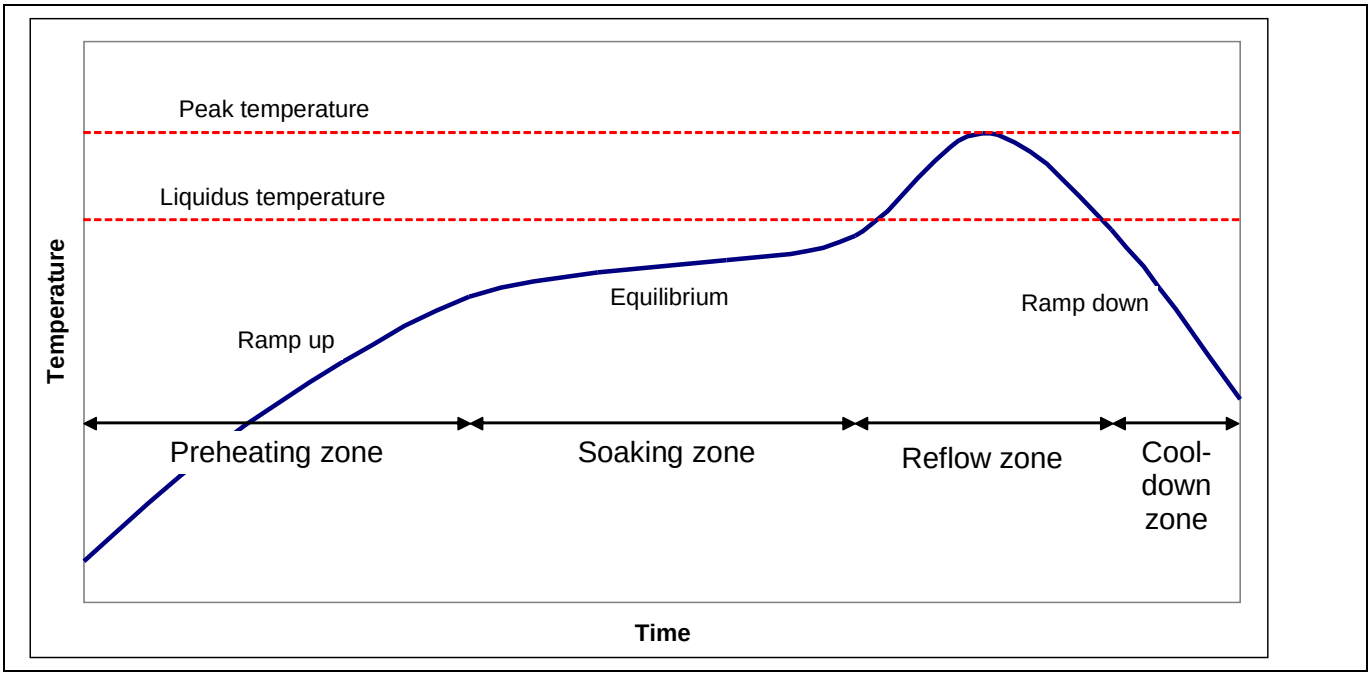

Figure 2 Various phases of a reflow solder profile

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."

SG-WLL packages are generally suitable for double-sided mounting as well.

For further details about the reflow profile please refer to the General Recommendations for Assembly of Infineon Packages in "Mounting of SMDs," available at www.infineon.com\packages.

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 printing, the flux residues usually do not have to be removed after the soldering process. Cleaning beneath a SG-WLL package is not possible because the solder joints are below the component and the gap between package and PCB is small. Cleaning 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 while taking into account the kinds of 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 and/or dried very thoroughly. Contact the solder paste or flux manufacturer for recommended cleaning solutions.

5 Inspection

Because the solder-paste printing process is extremely important to the quality of the solder joint, it is recommended to focus on the quality of the printing result. This can be done by using an automated Solder Paste Inspection (SPI) machine, for example. The solderable surface of the components (NiP – Pd - Au stack) provides very good and stable results in solderability. Therefore the focus should be set to the amount of printed solder paste on the respective PCB pads.

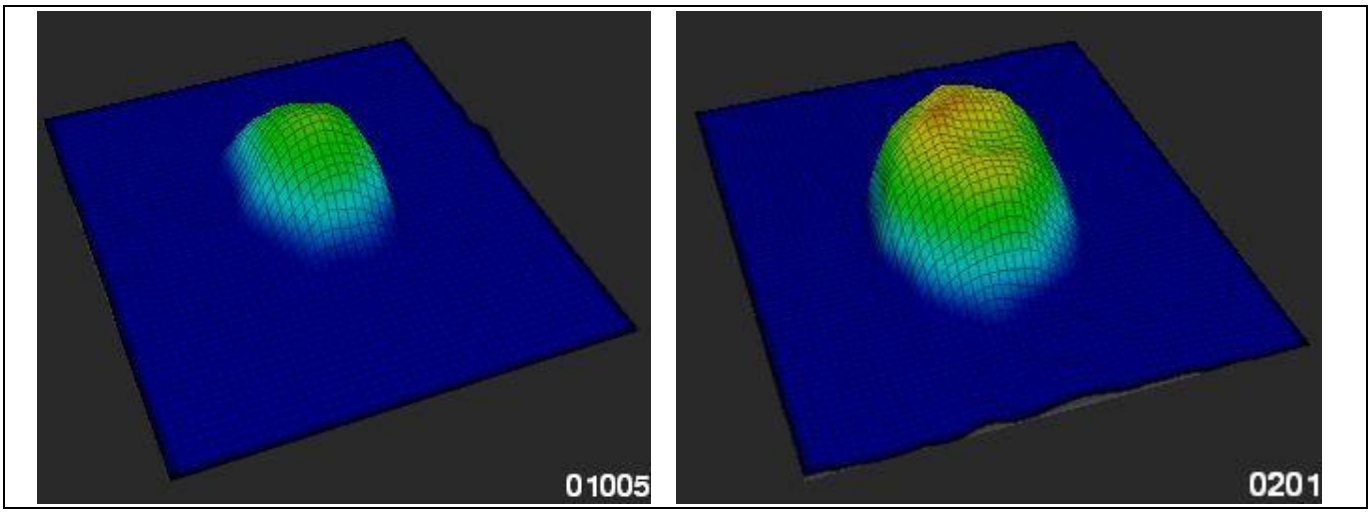

Figure 3 Typical SPI pictures of printed solder paste

A visual inspection of the solder joints with conventional Automatic Optical Inspection (AOI) systems or a manual visual inspection is limited to the outer surface of the solder joints. In most cases, these are visible and can be judged by looking at them from the side, rather than from the top as most AOI systems do. When it comes to SG-WLL packages, optical inspections are usually not very successful. Only big misplacement errors can be detected.

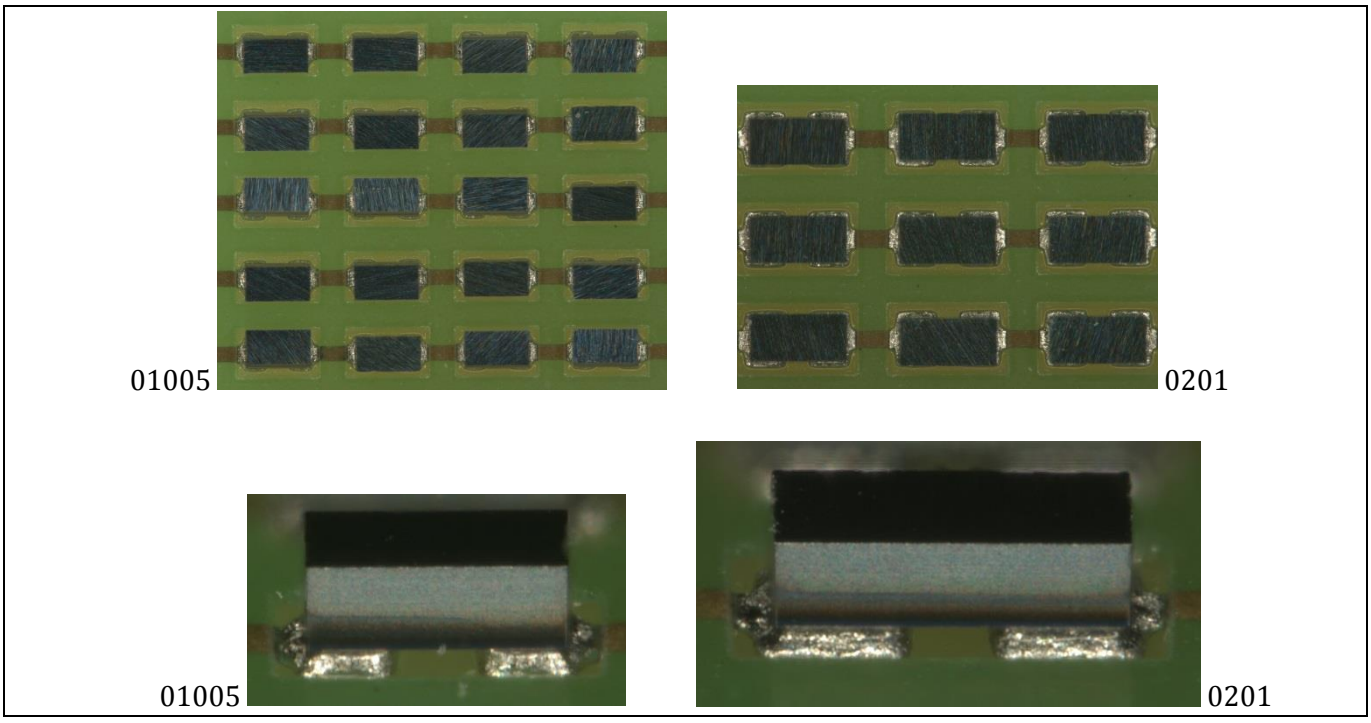

Figure 4 SG-WLL packages soldered to PCB

Inspection

The only reasonable method for efficient inline control is the implementation of 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 inspecting, controlling, analyzing, and data transfer routines. These systems quite reliably 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. For the acceptability of electronic assemblies, please refer also to the IPC-A-610 standard.

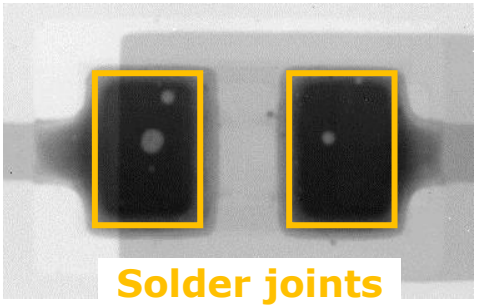

Figure 5 X-Ray image of a SG-WLL-2-1 package soldered to PCB

Cross-sectioned soldered packages can serve as useful tools for sample monitoring or initial process control. They help technicians to get an idea about the quality of the solder joints, intermetallic compounds, and voids.

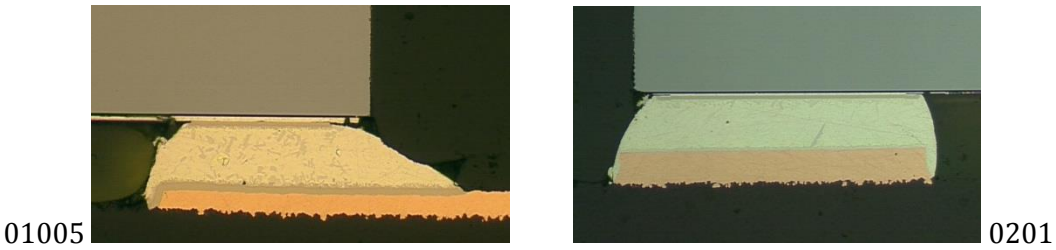

Figure 6 Cross sections through SG-WLL solder joints

Because the packages are bare silicon products, anyone performing an inspection after reflow soldering needs to be aware that a very small tilt of a package will make the package positioning look worse than it really is. The shiny backside surface appears different even if the tilt is not sufficient to cause a functional defect or affect the reliability of the solder joints. In particular, SG-WLL-2 packages show the described phenomenon due to the fact that 2-pin packages always show a tilt. Investigations on the 2nd level reliability have proven that the tilt of 2-pin packages does not affect the solder joint lifetime.

Figure 7 SG-WLL-2 packages soldered to PCB, showing different light reflection due to small tilt

6 Rework

If a defective component is detected after board assembly, the device can usually be removed and replaced by a new one. Due to possible damage while removing the component, a desoldered component should not be reused. Nevertheless, desoldering the old component (if analysis afterwards is planned) and resoldering of the new component have to be done very thoroughly.

Repair of single solder joints is not recommended.

For SG-WLL packages in particular, you need to take into account that the devices are very small and the device material is bare silicon. If a rework (component replacement) is planned, the equipment used should be suitable for the smallest package sizes and for handling bare silicon devices (e.g. no metal tweezers used).

Revision History

Major changes since the last revision

Document version	Description of change
DS1	Initial version
DS2	Including package size 01005

Edition <2016-02-25>
Published by
Infineon Technologies AG
81726 Munich, Germany

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