Additional Information, DS1, June 2006

Recommendations for Printed Circuit Board Assembly of Infineon PG-TSSOP-38-4 Packages

Never stop thinking

Edition 2006-06-14

Published by
Infineon Technologies AG
81726 München, Germany

© Infineon Technologies AG 2006.
All Rights Reserved.

Legal Disclaimer

The information given in this document shall in no event be regarded as a guarantee of conditions or characteristics (“Beschaffenheitsgarantie”). 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 your 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 your nearest Infineon Technologies Office.

Infineon Technologies Components may only be used in life-support devices or systems 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.

1 Package Description

Infineon's PG-TSSOP-38-4 (Plastic Green Thin Shrink Small Outline Package, **Figure 1**) is a Pb-free plastic encapsulated SMD device with a copper leadframe using 38 gull-wing leads along two sides of the package body.

Features

- Thin package
- Leads and whole package leadframe plated with a stacked layer system of Ni + Pd + an alloy of Au-Ag on the very top (ultra-thin pre-plated frame)
- Green product (lead-free and halogen-free materials)
- RoHS compliant
- Package outline according to JEDEC MO-153

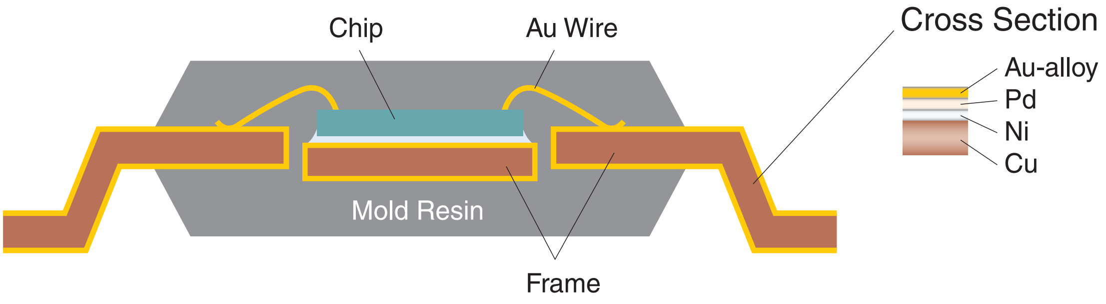


Figure 1 PG-TSSOP-38-1
Sketch of cross sectional view (best visible if printed in color mode)

2 Package Handling

2.1 ESD Protective Measures

Semiconductor devices are normally electrostatic discharge sensitive devices (ESDS) requiring specific precautionary measures in respect of handling and processing. Only in this way is it possible to insure that the components can be inserted into assemblies without becoming damaged. Discharging of electrostatic charged objects over an IC, caused by human touch or by processing tools may cause high current respectively high voltage pulses, which may 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 damages, either. ESD protective measures must therefore prevent a contact with charged parts as well as a charging of the ICs. Protective measures against ESD include both the handling and processing and the packing of ESDS. A few hints are provided below on handling and processing.

2.1.1 Workplace-ESD Protective Measures

- 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 bonding point for wrist strap
- Trolleys with dissipative surfaces and wheels
- Suitable shipping and storage containers
- No sources of electrostatic fields

2.1.2 Equipment for Personal

- Dissipative/conductive footwear or heel straps
- Suitable smocks
- Wrist strap with safety resistor
- Volume conductive gloves or finger cots
- Regular training of staff

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

Relevant standard which should be considered here

IFX packs according to the IEC 60286-* series

IEC 60286-3	Packaging of components for automatic handling - Part 3: Packaging of surface mount components on continuous tapes.

Moisture Sensitive Surface Mount Devices 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 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 popcorn effects.

List of relevant standards which should be considered

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 practice in handling, packing and storage
ISO 14644-1	Clean rooms and associated controlled environments Part 1: Classification of airborne particulates

Table 1 General Storing Conditions - Overview

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)

3 Printed Circuit Board (PCB)

3.1 Introduction

Generally speaking printed circuit board design and construction are key factors for achieving high board assembly yield and also sufficient reliability. Examples are PCB pad designs of the leads or CTE and stiffness of the PCB. Also board finish has to be considered. As TSSOP packages are gull-wing type SMDs, for the majority of applications a standard procedure for routing the signal, power and ground pins on the PCB can be used. Nevertheless we want to emphasize, this document is just a guideline to support our customers in board design. Additionally, studies at the customers may be necessary for optimization, which take into account the actual PCB manufacturer's capabilities, the customer's SMT processes and product specific requirements.

3.2 PCB Pad Design

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

- ”Solder mask defined“ pad (**Figure 2**): 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.

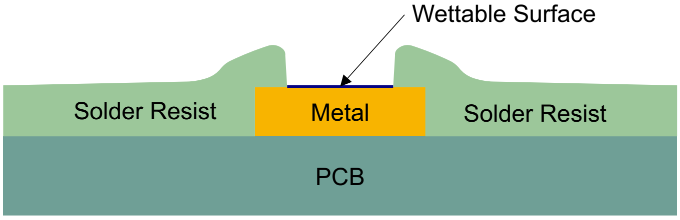


Figure 2 **SMD Pad**

- ”Non solder mask defined“ pad (**Figure 3**): Around each copper pad there is solder mask clearance. It is necessary to specify the dimensions and tolerances of the solder mask clearance in this way, that no overlapping of the solder pad by solder mask occurs (depending on PCB manufacturers tolerances, 75 µm is a widely used value).

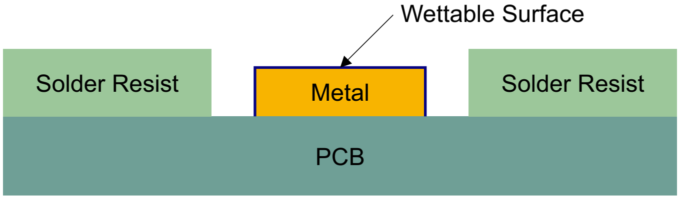


Figure 3 **NSMD Pad**

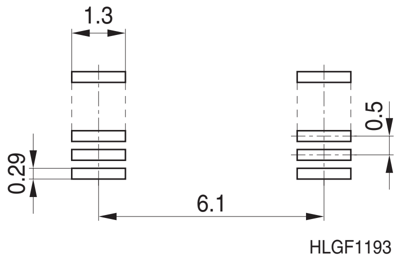
Because non solder mask defined pads provide more space for routing and result in a higher solder joint reliability (also the side walls of the lands are wetted by the solder, which results in less stress concentration), non solder mask defined pad type is recommended for the solder pads on the PCB.

A mixture between solder mask defined and non solder mask defined pads for one component is not recommended. Heavy misalignment between solder mask and board pads can lead to a unbalanced wettable surfaces and solder joints.

Figure 4 shows the recommended PCB pad design. Please note, that there is no exact congruency of PCB pads and package pads.

If microvias are placed inside the solder pads, it is recommended to specify a good flatness of the vias. Deep dips inside the pads may cause solder joint voiding.

As a general guideline please refer to IPC 7351/7355
(Generic Requirements for Surface Mount Design and Land Pattern Standard).



All dimensions in mm

Figure 4 PCB Pad Design Recommendation

3.3 PCB Pad Finishes

The solder pads must have good wettability to the soldering material (solder paste). In general all finishes are well proven for SMT assembly, but especially for fine pitch applications the quality of the plating/ finish gets more important. Because of the uneven surface of Hot Air Solder Leveling (HASL) finish, Pb-free or Pb containing HASL is less preferred for PG-TSSOP-38-4 assembly compared to completely ”flat“ platings like Cu-OSP (OSP: Organic Solderability Preservative) or electroless Sn or NiAu.

From package point of view it is not possible to give a definite recommendation for PCB pad finish. It also depends strongly on board design, pad geometry, component types on the board and process conditions.

For Pb-free soldering the PCB must be able to resist the higher temperatures which are occurring at the lead-free process. This question should be discussed with the PCB-supplier. Generally spoken, the wettability of tin-lead solder paste on most surface finishes is better compared to lead-free solder paste.

4 Board Assembly

4.1 General Remarks

Many factors within the board assembly processes have influence on assembly yield and board level reliability. Examples are design and material of the stencil, 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, studies at the customers may be necessary for optimization, which take into account the actual printed circuit board, the customer's SMT equipment and product specific requirements. The lead-free PG-TSSOP-38-4 can generally be assembled with either SnPb based or Pb-free SnAgCu based solder pastes and reflow processes.

4.2 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 a PCB. For PG-TSSOP-38-4 packages it is recommended to use 100-150 µm thick stencils. The apertures should be of the same size and shape as the metal pads on the PCB (for recommendations please refer to the PCB section of this document).

To ensure a uniform and high solder paste transfer to the board, lasercut (mostly made of stainless steel) or electroformed stencils (Nickel) should be preferred. Rounding the corners of the apertures (radius ~50 µm) can be supportive for the paste release.

4.3 Solder Paste

Solder paste consists of solder alloy and a flux system. Normally the volume is split into about 50% alloy and 50% flux. In terms of mass this means approx. 90 wt% alloy and 10 wt% flux system. The flux system has the function to remove contaminations from the solder joints during the soldering process. The capability of removing contaminations is given by the respective activation level. The solder paste metal alloy has to be of leaded eutectic or near-eutectic composition (SnPb or SnPbAg) or lead-free composition (SnAgCu whereas Ag: 3-4 wt%, Cu 0.5-1 wt%). The paste must be suitable for printing the solder stencil aperture dimensions, Type 3 paste should be sufficient. Solder paste is sensitive to age, temperature and humidity. Please notice the handling recommendations of the paste manufacturer.

4.4 Component Placement

PG-TSSOP-38-4 packages have to be placed accurately according to their geometry. The positioning of the packages by hand 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 a correct centering of the complete package.

The maximum tolerable displacement of the components should not exceed 20% of the metal pad width on the PCB (for non solder mask defined pads). In consequence, for PG-TSSOP-38-4 packages the device pad to PCB pad misalignment has to be better than 70 µm to assure a robust mounting process. This is generally achievable with a wide range of placement systems.

However also higher misplacement can result in well soldered devices. In this case the self-centering effect during reflow due to the solder's high surface tension can align the package to the board pads. Only the customer can decide under the consideration of his special processes and materials if the self-alignment effect can be used to tolerate a higher misplacement rate.

The following remarks are important:

- Especially on large boards local fiducials close to the device can compensate a large amount of PCB tolerances.
- It is recommended to use the lead recognition capabilities of the placement system, not the outline centering.
- To ensure the identification of the packages by the vision system, an adequate lighting as well as the correct choice of the measuring modes are necessary. Please refer to the equipment manuals for the best settings.
- Too much placement force can lead to squeezed out solder paste and cause solder joint shorts. On the other hand too low placement force can lead to insufficient contact between package and solder paste and this can lead to open solder joints or badly centered packages. The customer's equipment manual as well as the application notes of the solder paste manufacturer may help to find the appropriate settings.

4.5 Soldering

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

- forced convection
- vapor phase
- infrared (with restrictions)
- wave soldering (with restrictions)

and typical temperature profiles are suitable for board assembly of the PG-TSSOP-38-4. At the reflow process each solder joint has to be exposed to temperatures above solder 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. It is important that the maximum temperature of the device during reflow does not exceed the specified peak temperature (see also **Chapter 4.5.2**). 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 most recommended type is forced convection reflow. Nitrogen atmosphere can generally improve solder joint quality. In case of Pb-free processing at higher reflow temperatures nitrogen atmosphere may reduce oxidation and improve the solder joint quality significantly. Therefore we suggest to use N2 even though it is normally not necessary for soldering PG-TSSOP-38-4 packages.

The temperature profile of a reflow process is one of the most important factors of the soldering process. It is divided into several phases, each with a special function. The single parameters are influenced by various factors, not only by the package. It is essential to follow first the solder paste manufacturer's application notes. Additionally, most PCBs contain more than one package type and therefore the reflow profile has to be matched to all components' and materials' demands. We recommend measuring the solder joints' temperatures by thermocouples beneath the respective packages. It has to be considered that components with large thermal masses do not heat up with the same speed as lightweight components and also 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.

Wave soldering of PG-TSSOP-38-4 packages is possible but not recommended for a fine pitch package.
In any case wave soldering here requires a special footprint design and orientation of the packages on the PCB (e.g. 45° orientation towards transport direction).

4.5.1 Double-Sided Assembly

PG-TSSOP-38-4 packages are generally suitable for mounting on double-sided PCBs. That means that in a first step 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.

4.5.2 Processing of Moisture-Sensitive Components

For moisture-sensitive packages like the PG-TSSOP-38-4 it is necessary to control the moisture content of the components. The penetration of moisture into the package epoxy materials e.g. molding compounds is generally caused by exposure to the ambient air. In many cases moisture absorption leads to moisture concentrations in the component which are high enough to damage the package during the solder cycle (”popcorn effect“ due to the evaporation of the humidity). Thus it is necessary to dry moisture-sensitive components, to seal them in a moisture-resistant bag and only to remove them immediately prior to processing (soldering onto the PCB). The permissible time (from opening the moisture barrier bag until the final soldering process) that a component can remain outside the moisture barrier bag, 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 thus defines eight different MSLs (see **Table 2**). Please refer to the ”Moisture Sensitivity Caution Label“ on the packing material, which contains information about the moisture sensitivity level of our product.

IPC/JEDEC-J-STD-20 specifies the maximum reflow temperature, which shall not be exceeded during the board assembly at the customer.

Table 2 Moisture Sensitivity Levels (acc. to IPC/JEDEC J-STD-033)

Level	Floor Life (out of bag)	
	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

Internet Link to **Association Connecting Electronics Industries (IPC)**

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 after opening the moisture barrier bag (dry pack) indicates too much moisture (read within 1 minute after opening), the packages 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. Notice that packing material possibly cannot withstand the baking temperature. See imprints/ labels on the respective packing for maximum temperature.

4.6 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 do not have to be removed after the soldering process. However, if the solder joints have to be cleaned, the cleaning method (e.g. ultrasonic, spray or vapor cleaning) and solution have to be selected with consideration of the packages to be cleaned, the used flux in the solder paste (rosin-based, water-soluble, etc.), environmental and safety aspects. Removing/ drying even of small residues of the cleaning solution should also be done very thorough. Contact the solder paste manufacturer for recommended cleaning solutions. The PG-TSSOP-38-4 is capable of being cleaned with standard procedures applied for PCB cleaning.

4.7 Inspection

A visual examination of the solder joints with conventional AOI (automatic optical inspection) systems is applicable at gull-wing shaped leads. In most cases they are visible and can be judged by looking at them from the top (orthogonal view) as well as from the side (angle/inclined view). However for fine pitch components like the PG-TSSOP-38-4 it is recommended to use the angle view method, since this results in a lower pseudo-defect rate and a higher probability of detection for unsoldered pins. Depending on the type of solder paste used the residues after reflow influence the detection rate as well. Generally speaking it should taken into account that an AOI equipment is not really able to test exactly according to the IPC-A-610 quality standard. Sometimes stricter criteria are used and the final classification of the solder joints is often released by a human. E.g. the solder flow at the end of a gull-wing lead is not required according to the mentioned standard.

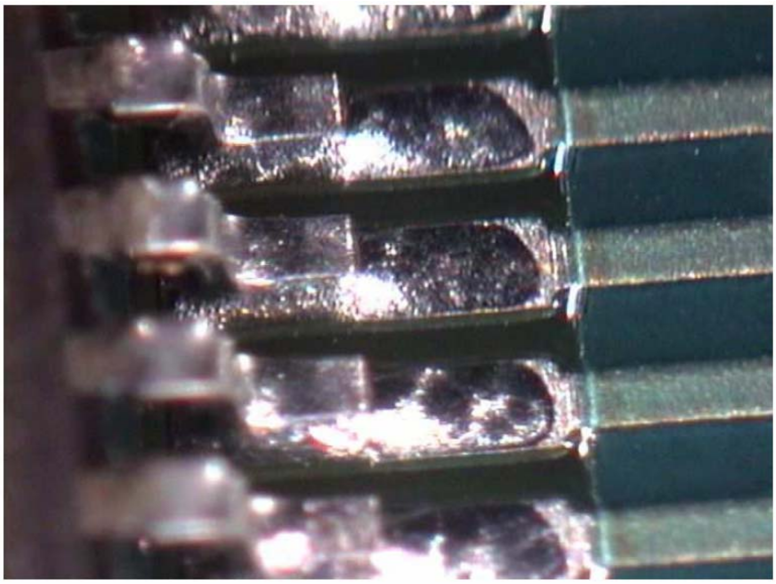


Figure 5 Typical occurrence of PG-TSSOP-38-4 solder joints (Pb-free)

Alternatively the implementation of an AXI (automatic X-ray inspection) equipment may lead in an efficient inline control system. AXI systems are available as 2D and 3D solutions. They usually consist of an X-ray camera with appropriate hard- and software needed for inspection, controlling, analyzing and data transfer routines. These systems enable the user to detect soldering defects like poor soldering, bridging, voiding and missing parts quite reliable. But other defects like broken solder joints are not easily detectable by X-ray. For acceptance criteria of electronic assemblies please refer also to the IPC-A-610 standard.

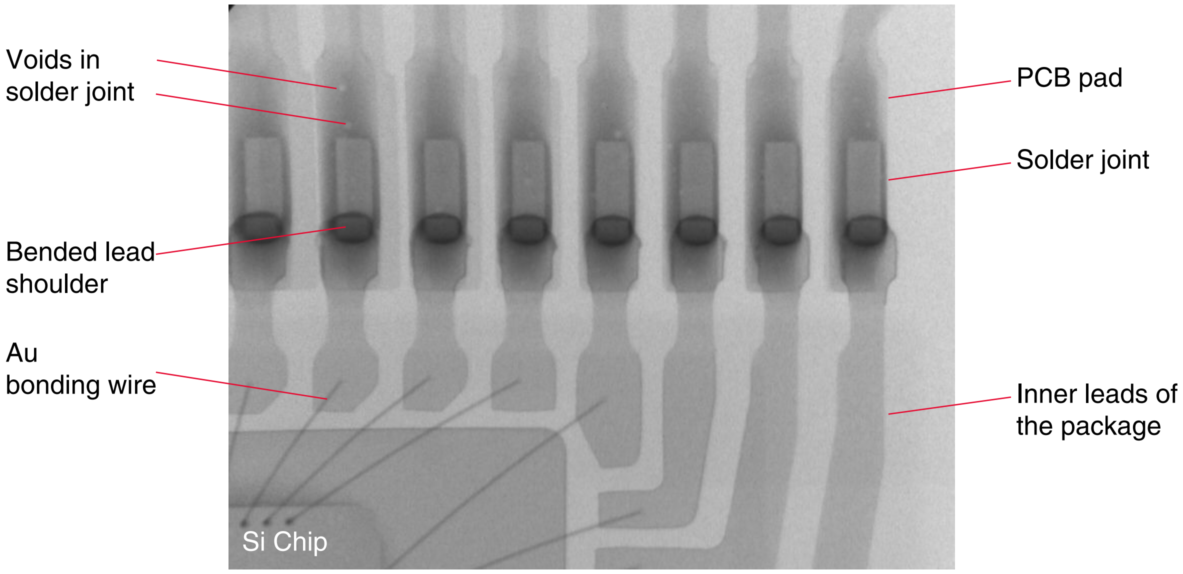


Figure 6 Typical X-ray image of a soldered PG-TSSOP-38-4.
Visible are solder joints with some small voids, bonding wires, Si chip, board pads and leadframe parts.

Cross sectioning of a soldered package as well as dye penetrant analysis can serve as tools for sample monitoring only, due to their destructive character. But they help to get an idea about the quality of the solder joint, intermetallic compounds and voids.

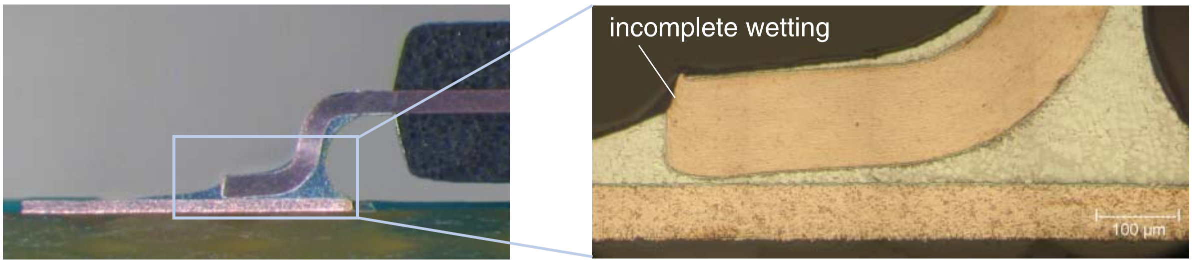


Figure 7 Cross section of a PG-TSSOP-38-4 solder joint (lead-free).
Non-/incomplete-wetted end of a lead is not a reject criterion.

In general lead-free solder joints look different than tin-lead (SnPb) once. Tin-lead joints have a bright and shiny surface, a dull surface is an indicator for an insufficient solder joint. Lead-free solder joints do not have this bright surface. Lead-free solder joints are dull and grainy. These surface properties are caused by the irregular solidification of the solder itself, as the used solder alloys are not exactly eutectic (like the 63Sn37Pb solder alloy). This means the SnAgCu-solders do not have a melting point but a melting range of some degrees. Although lead-free solder joints have this dull surface, this does not mean that lead-free joints are of lower quality or weaker than the SnPb joints. Additionally the surface of the ultra-thin pre-plated leadframe of the PG-TSSOP-38-4 with a final layer of an Au-Ag alloy on top of its leads looks a bit different. As the Au/Ag ratio is specified to a range of 45-65wt% of Au, the leads may occur more yellowish in color when being at the Au rich border line of the spec. (ref. to **Figure 8**). This characteristic make it necessary to instruct the inspection personnel how these new lead-free joints look like, and/ or to adjust optical inspection systems slightly to these solder joints.

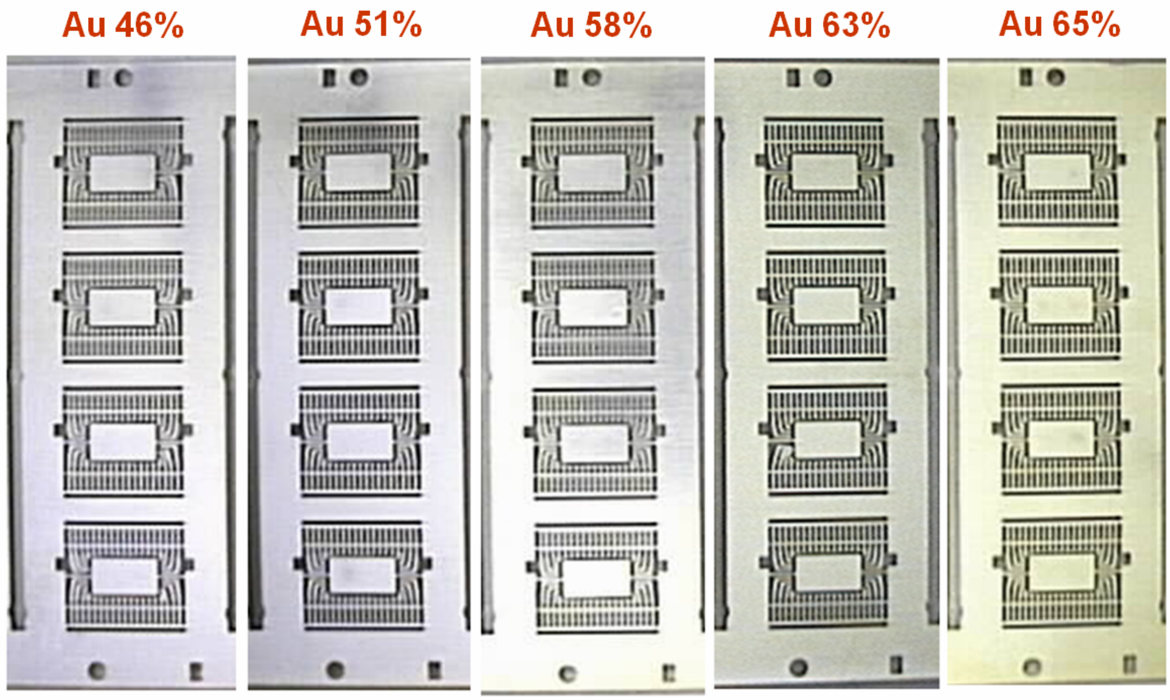


Figure 8 **Leadframes of PG-TSSOP-38-4 with different final plating within spec. limits. (Au wt% vs. color change of the surface: visible if printed in color mode only.)**

5 Rework

If a defect component is observed after board assembly the device can be removed and replaced by a new one. Repair of single solder joints is generally possible but requires proper tools for fine pitch applications while also the remarks in **Chapter 4.5** for temperature profile and exposure time to ambient atmosphere have to be respected.

5.1 Tooling

The rework process is commonly done on special rework equipment. There are a lot of systems available on the market, and for processing these packages the equipment should fulfill the following requirements:

- *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 free-programmable temperature profiles (e.g. by PC controller) it is possible to adapt the profiles to different package sizes and masses. PCB pre-heating from underside is recommended. Infrared heating can be applied, especially for pre-heating the PCB from underside, but it should be only supporting the hot air flow from the upside. Instead of air also nitrogen can be used.
- *Vision system:* The bottom side of the package as well as the site on the PCB should be observable. For precise alignment of package to PCB a split optic should be implemented. 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 ±70 µm. The system should have the capability of removing solder residues from PCB pads (special vacuum tools).

5.2 Device Removal

If it is intended to send a defect component back to the supplier, please note that during the removal of this component no further defects must be introduced to the device, because this may hinder the failure analysis at the supplier. This includes the following recommendations:

- *Moisture:* According to his moisture sensitivity level, possibly the package has to be dried before removal. If the maximum storage time out of the dry pack (see label on packing material) is exceeded after board assembly, the PCB has to be dried according to the recommendations (see **Chapter 4.5**), otherwise too much moisture may have been accumulated and damage may occur (popcorn effect).
- *Temperature profile:* During soldering process 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 (see **Chapter 4.5**).
- Mechanics: Be aware not to apply high mechanical forces for removal. Otherwise failure analysis of the package can be impossible or PCB can be damaged. For large packages pipettes can be used (implemented on most rework systems), for small packages tweezers may be more practical.

5.2.1 Site Redressing

After removing the defect component the pads on the PCB have to be cleaned from solder residues.
Do not use steel brushes because steel residues can lead to bad solder joints. Before placing a new component it is recommended to apply solder paste on each PCB pad by printing (special micro stencil) or dispensing.
No-clean solder pastes should preferably be used.

5.3 Reassembly and Reflow

After preparing the site, the new package can be placed onto the PCB. The package is positioned exactly above the PCB pads, in height just that there is no contact between the package and the PCB and the package is then dropped into the printed or dispensed solder paste depot (Zero-force-placement). During soldering process 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 (see **Chapter 4.5**). It have also to be considered that as a matter of course the newly placed package is a moisture sensitive device as well (ref. to **Chapter 4.5.2** for details).

6 List of references

IPC/EIA/JEDEC-J-STD-006	Requirements for Electronic Grade Solder Alloys and Fluxed and Non-fluxed Solid Solders for Electronic Soldering Applications
IPC/EIA/JEDEC-J-STD-001	Requirements for Soldered Electrical and Electronic assemblies
IPC A-610	Acceptability of Electronic Assemblies
IPC/EIA/JEDEC J-STD-002	Solderability tests for Component Leads, Terminations, Lugs, Terminals and Wires
IPC/JEDEC J-STD-033/ -020	Handling, Packing, Shipping and Use of Moisture / Reflow Sensitive Surface Mount devices / Moisture/reflow Sensitivity Classification for Non-hermetic Solid State Surface Mount Devices
JESD22-B102	Test Method for Solderability
IEC 60068-2-58	Test methods for solderability, resistance to dissolution of metallization and to soldering heat of surface mounting devices
IPC 7351/7355	Generic Requirements for Surface Mount Design and Land Pattern Standard

www.infineon.com

Published by Infineon Technologies AG