

TO-Leadless: A new Package for High Current High Reliability Applications

IFAT PMM APS SE DC
Ralf Walter

Edition 2011-02-02
Published by
Infineon Technologies Austria AG
9500 Villach, Austria
© Infineon Technologies Austria AG 2011.
All Rights Reserved.

Attention please!

THE INFORMATION GIVEN IN THIS APPLICATION NOTE IS GIVEN AS A HINT FOR THE IMPLEMENTATION OF THE INFINEON TECHNOLOGIES COMPONENT ONLY AND SHALL NOT BE REGARDED AS ANY DESCRIPTION OR WARRANTY OF A CERTAIN FUNCTIONALITY, CONDITION OR QUALITY OF THE INFINEON TECHNOLOGIES COMPONENT. THE RECIPIENT OF THIS APPLICATION NOTE MUST VERIFY ANY FUNCTION DESCRIBED HEREIN IN THE REAL APPLICATION. INFINEON TECHNOLOGIES HEREBY DISCLAIMS ANY AND ALL WARRANTIES AND LIABILITIES OF ANY KIND (INCLUDING WITHOUT LIMITATION WARRANTIES OF NON-INFRINGEMENT OF INTELLECTUAL PROPERTY RIGHTS OF ANY THIRD PARTY) WITH RESPECT TO ANY AND ALL INFORMATION GIVEN IN THIS APPLICATION NOTE.

Information

For further information on technology, delivery terms and conditions and prices please contact 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.

AN 2013-05

Revision History: 13-05-10, V1.1

Subjects: TO-Leadless: A new Package for High Current High Reliability Application

Authors: Ralf Walter

We Listen to Your Comments

Any information within this document that you feel is wrong, unclear or missing at all? Your feedback will help us to continuously improve the quality of this document. Please send your proposal (including a reference to this document) to: Ralf.Walter@infineon.com

Table of contents

| | | |
|----------|---|-----------|
| 1 | Introduction | 5 |
| 2 | Mechanical dimensions | 5 |
| 2.1 | Space reduction | 5 |
| 2.2 | High Continuous Current | 6 |
| 2.3 | Automatic Optical Inspection (AOI) | 7 |
| 3 | Package Handling | 9 |
| 3.1 | ESD-Protective Measurement | 9 |
| 3.1.1 | ESD-Protective Measure in the Workplace | 9 |
| 3.1.2 | Equipment for Personnel | 9 |
| 3.1.3 | Production Installations and Processing Tools | 9 |
| 3.2 | Packing of Components | 10 |
| 3.3 | Moisture Sensitivity Level | 10 |
| 3.4 | Storage and Transportation Conditions | 11 |
| 3.5 | Handling Damage and Contamination | 11 |
| 3.6 | Component Solderability | 12 |
| 4 | Printed Circuit Board | 12 |
| 4.1 | Routing | 12 |
| 4.2 | PCB Pad Design | 12 |
| 4.3 | Pad Surfaces | 13 |
| 5 | PCB Assembly | 14 |
| 5.1 | Solder Stencil | 14 |
| 5.2 | Solder Paste | 15 |
| 5.3 | Component Placement | 15 |
| 5.4 | Soldering | 16 |
| 5.5 | Cleaning | 18 |
| 5.6 | Inspection | 18 |
| 6 | Rework | 19 |
| 6.1 | Tooling | 19 |
| 6.2 | Device Removal | 19 |

| | | |
|-----|-----------------------------|----|
| 6.3 | Site Redressing..... | 19 |
| 6.4 | Reassembly and Reflow | 20 |

1 Introduction

The TO-Leadless (P/PG-HSOF-8-1) is a molded package optimized for high power high reliability applications. Its small mechanical dimensions allow really compact designs and the high current capability combined with the low Thermal Resistance (R_{thJC}), resulting in lower chip temperatures enables the designer to go for higher power density and higher reliability.

All mechanical details shown in the following chapters and additionally a general recommendation how to handle Infineon's SMD devices could be found at www.infineon.com/packages.

Detailed mechanical information about the TO-Leadless (P/PG-HSOF-8-1) is available at TO-Leadless.

General information like datasheets, product brief etc. are available under www.infineon.com/toll.

2 Mechanics

2.1 Outlines

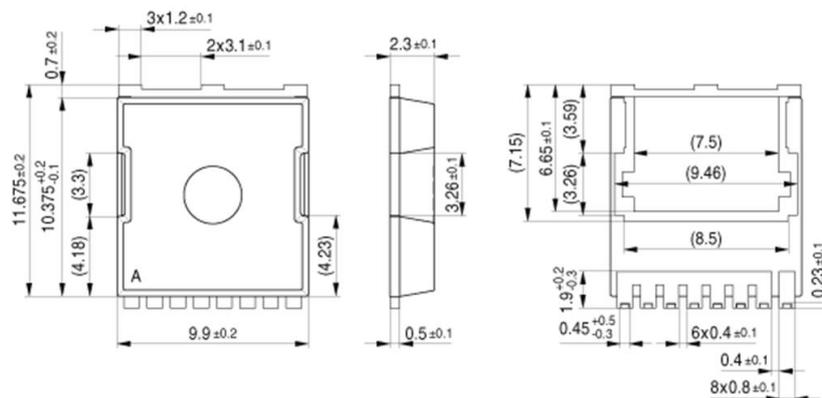


Figure 2.1: Outlines of TO-Leadless

2.2 Space reduction

Compared to the commonly used D2PAK or D2PAK 7Pin the TO-Leadless has a smaller footprint. It's only 11.7mm*9.9mm*2.3mm compared to the 15.0mm*10.0mm*4.4mm of the D2PAK (7Pin). This leads to a 30% smaller footprint and a 60% smaller space.

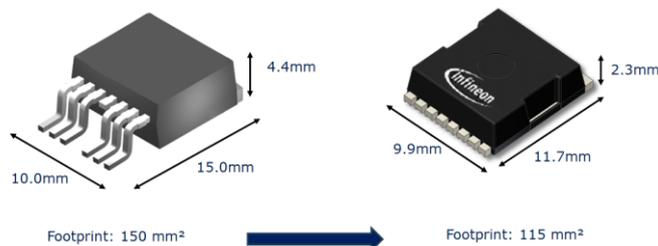


Figure 2.2: Space reduction of TO-Leadless compared to D2PAK 7Pin

2.3 High Continuous Current

The maximum continuous current of a device could be limited by different reasons:

- Silicon losses are too high with given thermal resistance R_{thJC}
- Bond wire losses due to too high current density
- Current density of solder joints

Because of the thin leadframe the thermal resistance R_{thJC} is also reduced compared to the D2PAK 7Pin. As a result the resistance R_{thJC} now is less than 0.4K/W (max.) for a 30mm² chip (D2PAK: 0.5K/W max.). The maximum continuous current capability is only in 150V limited by the silicon losses, in all other voltage classes the solder joints are the limiting factor. Figure 2.3 shows additionally the calculated Z_{thJC} for both D2PAK and the new TO-Leadless

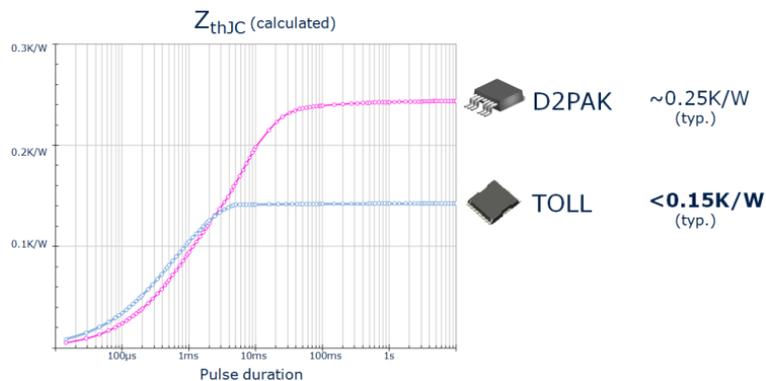


Figure 2.3: Z_{thJC} (calculated) of D2PAK 7Pin and TO-Leadless

Figure 2.4 shows a picture of the bond wires. Up to 5 bond wires with a diameter of 500µm each are used. Big solder connections to the chip also reduce the current density. Resulting in a lower temperature stress of the bond wires and the chip connections.

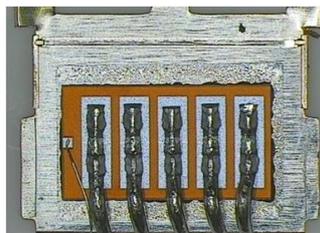


Figure 2.4: Five Bond wires, diameter 500µm each and massive solder connections

Depending on the package another possible bottleneck limits the maximum possible continuous current. Especially at higher current density combined with an increased temperature an effect called “electromigration” could weaken the solder connections. As a result the reliability of the complete device could be influenced negatively. The TO-Leadless offers a solderable area of more than 12mm² (D2PAK 7Pin: <8mm²), decreasing the current density by more than one third, reducing the temperature stress and the risk of electromigration. Figure 2.5 shows the comparison between these two power packages.

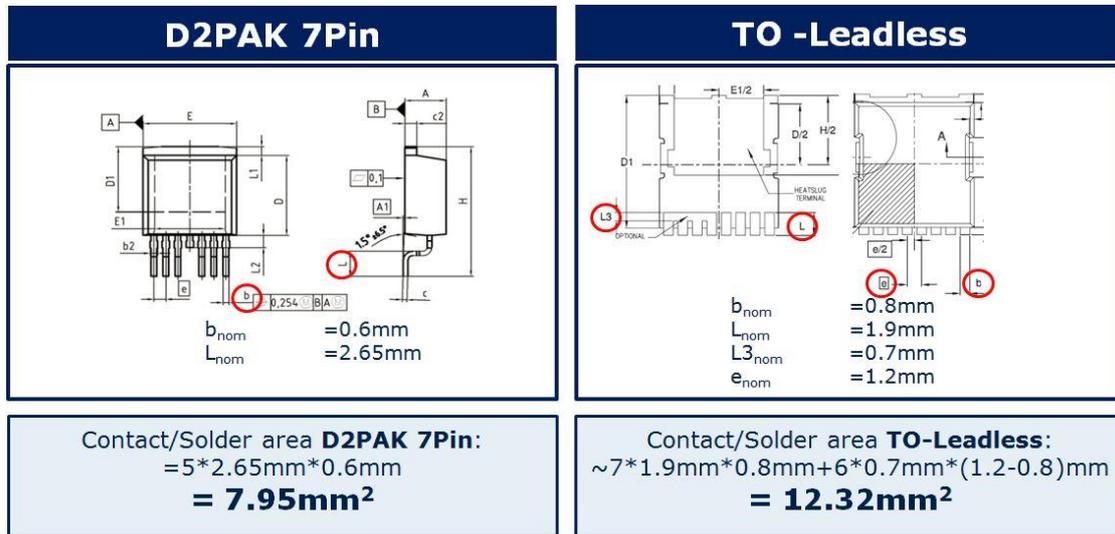


Figure 2.5: Increased solderable area reduces current density

2.4 Automatic Optical Inspection (AOI)

Leadless packages like CanPAK™ or SuperSO8 don't allow an AOI because the solder joints are (partly) hidden under the package. On the bottom side of the gate and source contacts trapezoidal grooves lead to a visible solder joint, avoiding the necessity of an expensive X-Ray inspection.

These grooves are the result of the trim-and-form process before plating and cutting.

Figure 2.6 shows on the left the view from the front side. On the right side (flipped view) the grooves are shown.

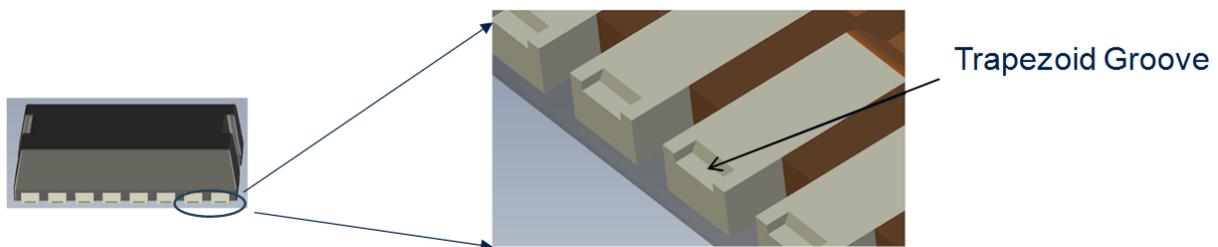


Figure 2.6: Tinned trapezoidal grooves on the tips of gate and source

After soldering it's easy to identify a good solder joint using standard AOI. In Figure 2.7 a typical result of a solder process is highlighted. A cut through a source connection shows the solder (light yellow). The groove is filled completely and additionally a solder meniscus is visible on the left side outside the part, allowing an assessment of the solder connection.

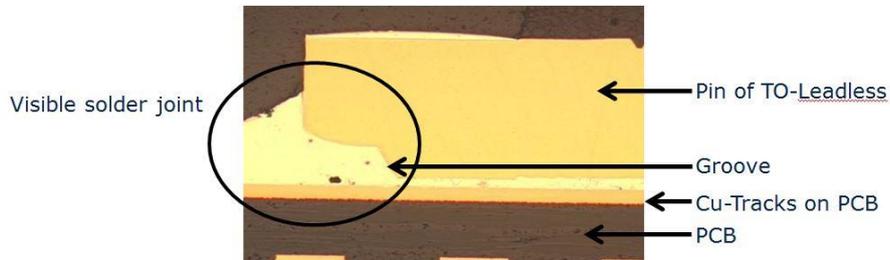


Figure 2.7: Visible solder meniscus allows a simple and unexpensive AOI

Shape and amount of the visible solder meniscus can be checked by AOI, avoiding reliability problems due to bad solder joints.

3 Package Handling

3.1 ESD-Protective Measurement

Semiconductors are normally Electro-Static Discharge Sensitive (ESDS) devices 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 ESDSs. A few hints are provided below on handling and processing.

3.1.1 ESD-Protective Measure 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

3.1.2 Equipment for Personnel

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

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

3.1.3 Production Installations and Processing Tools

- Machine and tool parts made of dissipative or metallic materials
- No materials having thin insulating layers or 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.

3.2 Packing of Components

Different packings such as fixtures for feeding components in an automatic pick&place machine (tape&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.

Generally the following list of standards dealing with packing should be considered if applicable for a given device and packing:

IFX packings 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.
- 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 Surface Mount Devices (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 www.infineon.com/packages

Other References:

- ANSI/EIA-481-Standards Proposal No. 5048, Proposed Revision of ANSI/EIA-481-B “8mm through 200mm Embossed Carrier Taping and 8mm & 12mm 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)

3.3 Moisture Sensitivity Level

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-resistant bag, and only remove them immediately prior to assembly to the Printed Circuit Board (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* defines eight different MSLs (see Table 3.1). Please refer to the “Moisture Sensitivity Caution Label” on the packing material, which contains information about the moisture sensitivity level 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.

| 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 |

Table 3.1: Moisture Sensitivity Levels (acc. to IPC/JEDEC J-STD-033, RH=Relative Humidity)

If moisture-sensitive components have been exposed to ambient air for longer than the specified time according to their MSLs, or the humidity indicator card indicates too much moisture after opening a Moisture Barrier Bag (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, tapes,...) may not withstand higher baking temperatures. Please refer to imprints/labels on the respective packing to determine allowable maximum temperature.

For Pb-free components, two MSLs can be given: One for a lower reflow peak temperature (Pb-containing process) and one for a higher reflow peak temperature (Pb-free). Each one is valid for the respective application.

3.4 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

| Product | Condition for storage |
|------------------------------------|-------------------------|
| Wafer/die | N2 or MBB (IEC 62258-3) |
| Component – moisture-sensitive | MBB (JEDEC J-STD-033*) |
| Component - not moisture-sensitive | 1K2 (IEC 60721-3-1) |

Table 3.2: General Storage Conditions – Overview (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.

3.5 Handling Damage and Contamination

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

TO-Leadless components in the packing are ready to use.

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 to conductive particles)

3.6 Component Solderability

The sufficiently thick and wettable metal surfaces (final plating) or solder depots/balls of most semiconductor packages assure good solderability, even after a long storage time.

Suitable methods for the assessment of solderability can be derived from JESD22B 102 or IEC60068-2-58.

TO-Leadless components are compatible with Pb-containing and Pb-free soldering.

4 Printed Circuit Board

4.1 Routing

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

4.2 PCB Pad Design

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

- „Solder-Mask Defined“ (SMD) pad: The copper pad is larger than the solder-mask opening above this pad. Thus the wettable area is defined by the opening in the solder mask.



Figure 4.1: Solder-Mask-Defined (SMD) Pad

- „Non-Solder-Mask Defined“ (NSMD) pad: Around each copper pad there is solder-mask clearance. It is necessary to specify the dimensions and tolerances of the solder mask clearance so that no overlapping of the solder pad by solder mask occurs (depending on PCB manufacturers' tolerances, 75 μm is a widely used value).

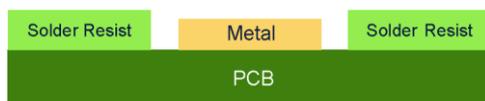


Figure 4.2: Non-Solder-Mask-Defined (NSMD) Pad

| Finish | Typical Layer Thickness [µm] | Properties | Concerns |
|---|------------------------------|--|---|
| HASL (SnAg) (Hot Air Solder Leveling) | > 5 | Low cost, widely used, well known in fabrication | Uneven surface, formation of humps, flatness of single pads has to be good for fine-pitch applications |
| Electroless Sn | 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 Ag | 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) | Typical 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 |

Figure 4.4: Typical PCB pad finishes

5 PCB Assembly

5.1 Solder Stencil

The solder paste is applied onto the PCB metal pads by stencil printing. The volume of the printed solder paste is determined by the stencil aperture and the stencil thickness. Too much solder paste will cause solder bridging, whereas too little solder paste can lead to insufficient solder wetting between all contact surfaces. In most cases the thickness of a stencil has to be matched to the needs of all components on the PCB. To ensure a uniform and high solder paste transfer to the PCB, laser-cut stencils are suitable and cost-effective.

The stencil thickness directly affects the amount of solder that is available to form the solder joints for source and gate contacts and the solder volume of source/gate pads has to be within a certain range to avoid electrical opens or shorts. This implies that the stencil apertures for source/gate pads have to be adjusted to the stencil thickness. A thick stencil results in a reduced aperture, a thin stencil in an enlarged aperture. Internal investigation showed good results with 150-µm stencil thickness, and the following Figure 5.1 shows the given aperture sizes.

When using stencils thinner than 100 µm, it is only possible to bridge the gap by overprinting of source/gate pads, which may result in an increased failure rate due to electrical shorts. Apertures for especially small gate pads for stencils thicker than 175 µm will barely release the solder paste.

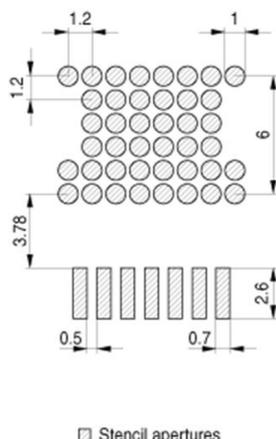


Figure 5.1: Recommended Stencil Design

5.2 Solder Paste

Solder paste consists of solder alloy and a flux system. Normally the volume is split into about 50% alloy and 50% flux and solvents. In term of mass, this means approximately 90 wt% alloy and 10 wt% flux system and solvents. The flux system has to remove oxides and contamination from the solder joints during the soldering process. The capacity for removing oxides and contamination is given by the respective activation level. The contained solvent adjusts the viscosity needed for the solder paste application process. The solvent has to evaporate during reflow soldering.

The paste must be suitable for printing the solder stencil aperture dimensions; Type 3 paste is recommended.

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

5.3 Component Placement

For the placement of TO-Leadless the following important machine specifications have to be considered:

- Placement accuracies of $\pm 50 \mu\text{m}$ are recommended. Tests have shown that greater inaccuracies are tolerable but not necessarily desirable.
- Placement forces of 1.5 to 2.5N or over-travel during placing of 50 to 100 μm are recommended.

Further details about the placement of TO-Leadless are described in the following paragraph.

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

Component placement accuracies of $\pm 50 \mu\text{m}$ 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 only located on the edge of the PCB for the entire PCB or additionally on individual mounting positions (local fiducials). These fiducials are detected by a vision system immediately before the mounting process and help to avoid displacement due to deviations in PCB geometry.

Recognition of the packages is performed by a special vision system, enabling the complete package to be centered correctly.

The following factors are important:

- Especially on large boards, local fiducials close to the device can compensate for PCB tolerances.
- The lead recognition capabilities of the placement system should be used rather than the outline centering. Outline centering can only be used for packages where the tolerances between pad and outline are small compared to the placement accuracy needed.
- To ensure the identification of the packages by the vision system, adequate lighting as well as the correct choice of measuring modes is necessary. The correct settings can be taken from 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 and may result in insufficient sticking of the component on the solder paste, which may then lead to shifted or dropped devices. Placement forces of 1.5 to 2.5 N or over-travel (= going further down with the component after the machine has registered the first touch down onto solder paste) during placing of 50 to 100 μm are good starting points.

A pick-up nozzle suitable for the package body size should be used. The nozzle should be slightly smaller than the package body. A bigger nozzle may lead to an irregular force distribution, thereby increasing forces at the edges of the package body in particular. On the other hand, a nozzle that is too small may lead to increased forces in the package center. Package bodies that are divided into different areas that have different heights require special care when choosing the nozzle. Nozzle shape and size are probably more critical in these cases.

5.4 Soldering

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

- Forced convection (max. qualified profile given by the JEDEC MSL classification)
- Vapor phase
- Infrared (with restrictions)

Typical temperature profiles are suitable for board assembly of the TO-Leadless.

During the reflow process, each solder joint has to be exposed to temperatures above the solder melting point or "liquidus" for a sufficient time to get the optimum solder joint quality, whereas overheating the PCB with its components has to be avoided. 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. The recommended type of process is forced convection reflow. Using a nitrogen atmosphere can generally improve solder joint quality, but is normally not necessary for soldering tin-lead metal alloys.

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. Figure 5.2 shows a general forced convection reflow profile for soldering TO-Leadless. Table 5.1 shows an example of the key data of such a solder profile that has been used for the Sn-Pb and for the Pb-free alloy listed above. Individual parameters are influenced by various facts, not only by the package. It is essential to follow the solder paste manufacturer's application notes, too. 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. Consider that components with large thermal masses do not heat up at the same speed as lightweight components, the position and the surrounding of the package on the PCB as well as the PCB thickness can also influence the solder-joint temperature significantly. Therefore, these reflow profiles should serve as guidelines, but have to be further adjusted to each actual application.

Because the thermal impact of reflow is critical for Pb-free solder pastes, linear temperature profiles can be applied to achieve a shorter reflow time in total. When reducing the soaking time, it is very important to ensure a homogeneous temperature distribution on the PCB; in this case, a convection oven is recommended.

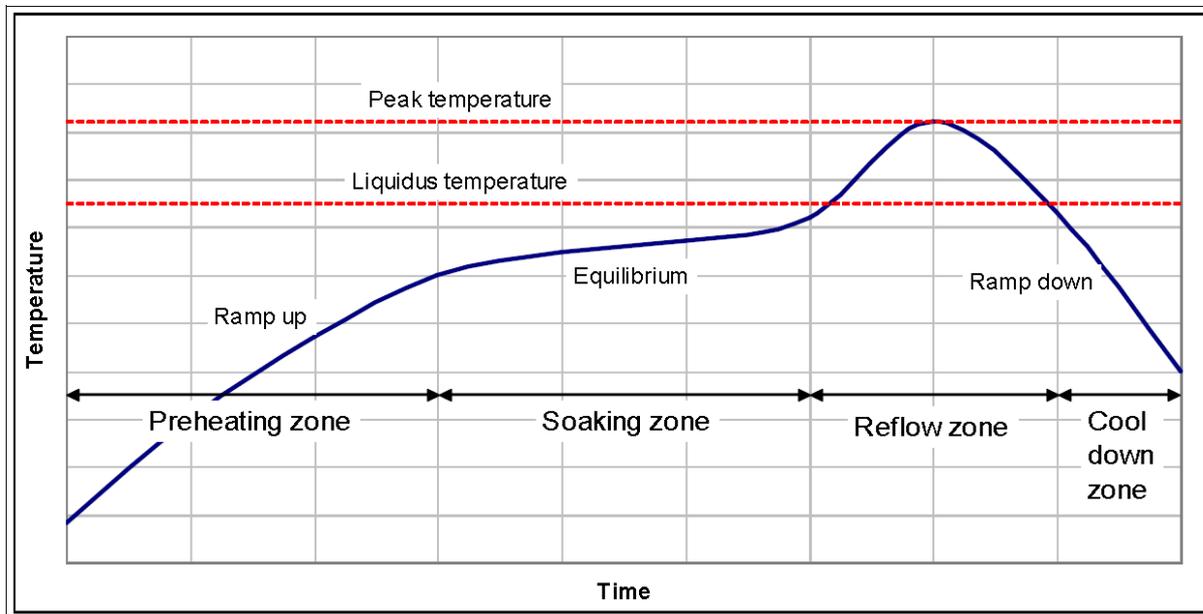


Figure 5.2: General forced-convection reflow solder profile

| Parameter | Tin-lead alloy (SnPb or SnPbAg) | Pb-free (SnAgCu) | Main influences come from |
|--|------------------------------------|---------------------|----------------------------|
| Preheating rate | 2.5K/s | 2.5K/s | Flux system (solder paste) |
| Soaking temperature | 140-170°C | 140-170°C | Flux system (solder paste) |
| Soaking time | 80s | 80s | Flux system (solder paste) |
| Peak temperature | 225°C | 245°C | Alloy (solder paste) |
| Reflow time above melting point (liquidus) | 60s | 60s | Alloy (solder paste) |
| Cool-down rate | 2.5K/s | 2.5K/s | |

Table 5.1: Example of the key data of a forced-convection reflow solder profile

Double-Sided Assembly

TO-Leadless generally suitable for mounting on double-sided PCBs. First, the board assembly is done on one side of the PCB (including soldering). Afterwards, the second side of the PCB is assembled.

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). This shape will be frozen at temperatures below the melting point of solder and therefore result in a higher stand-off on the bottom side after the reflow process. Heavy vibrations in a reflow oven may cause devices to drop off the PCB.

Underfill application

The board-level reliability of TO-Leadless is robust enough to do without underfill. Nonetheless if underfill should be applied, we recommend assessing the electrical and (thermo-)mechanical behaviour of assembled and underfilled components over the lifetime of the electronic device.

5.5 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.

Generally processes and materials which are used to avoid corrosion and following migration (e.g. coating) may have to be adjusted very thoroughly when using TO-Leadless.

If solder joints have to be cleaned, the cleaning method (e.g. ultrasonic, spray or vapour cleaning) and solution will depend on the packages to be cleaned, the flux used in the solder paste (rosin-based, water-soluble, etc.), and environmental and safety aspects. Even small residues of the cleaning solution should be removed/dried very thoroughly. It is recommended to contact the solder paste manufacturer for recommended cleaning solutions and to measure the residues after cleaning. Please also take into account that the amount of residues may depend on PCBs surface conditions, line output, cleaning solution life time, etc.

Infineon Technologies has tested TO-Leadless under harsh conditions like 85°C and 85% r.h. while applying voltage between gate and source pads (so-called H3TRB test acc. JESD22-A101C). After 1000 hours the leakage currents are still within the data sheet specification and after desoldering the components no hints for silver migration can be found

5.6 Inspection

A visual inspection of the solder joints with conventional Automatic Optical Inspection (AOI) systems is possible due to the grooves at the source and gate pads solder joints. Figure 5.3 shows a cut through a source connection, showing the solder joint at a source connection.

For the acceptance of electronic assemblies inspected optically, please refer also to the IPC-A-610 standard.

Automatic X-ray Inspection (AXI) systems are appropriate for efficient inline control. AXI systems are available as 2D and 3D solutions. They usually consist of an X-ray camera and the hardware and software needed for inspection, controlling, analysing, and data transfer routines. These reliable systems enable the user to detect soldering defects such as poor soldering, bridging, voiding and missing parts. However, other defects such as broken solder joints are not easily detectable by X-ray.

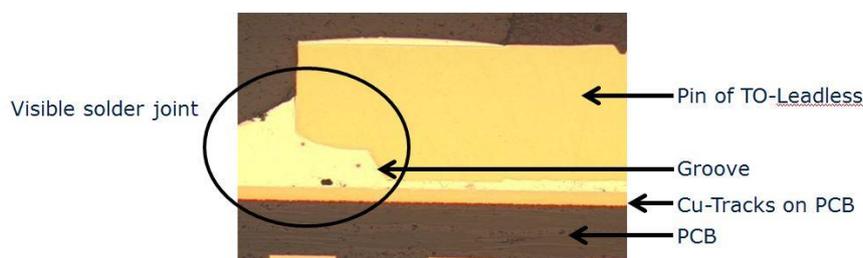


Figure 5.3: Visible solder meniscus allows a simple and unexpensive AOI

Pb-free solder joints look different from tin-lead (SnPb) solder joints. SnPb solder joints typically have a bright and shiny surface. Lead-free (SnAgCu) solder joints typically do not have this bright surface. Pb-free solder joints are often dull and grainy. These surface properties are caused by the irregular solidification of the solder, as the solder alloys are not exactly eutectic (like the 63Sn37Pb solder alloy). This means that SnAgCu-solders do not have a melting point but a melting range of several degrees. Although Pb-free solder joints have this dull surface, this does not mean that Pb-free joints are of lower quality or weaker than the SnPb joints. It is therefore necessary to teach the inspection staff what these Pb-free joints look like, and/or to adjust optical inspection systems to handle Pb-free solder joints.

6 Rework

If a defective component is observed after board assembly, the device can be removed and replaced by a new one. Repair of components' individual solder joints is not possible. Repair of underfilled components is not recommended. Damage caused by mechanically removing underfill from PCB and components may result in reduced reliability. Reusing components (especially underfilled components) is not recommended. Please take care that during rework no corrosive acting substances are applied on, underneath, or near to TO-Leadless which may lead to silver migration.

6.1 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 $\pm 100 \mu\text{m}$. The system should have the capability of removing solder residues from PCB pads (special vacuum tools).

6.2 Device Removal

If a component is suspected to be defective and will be sent back to the supplier, please do not remove this component from the PCB, but send the PCB to Infineon Technologies. This guarantees that no further defects are introduced to the device, because this may hinder the failure analysis at the supplier's facility. This procedure is mandatory for underfilled devices. For non-underfilled components, it is possible to remove the device gently from the PCB prior to sending it back. Please follow these precautions:

- **Moisture:** Depending on the component's MSL, the package may have 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 Section 2.3). 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 the package peak temperature is not higher and temperature ramps are not steeper than for the standard assembly reflow process.
- **Mechanics:** Be careful not to apply high mechanical forces for removal. Otherwise failure analysis of the package can be impossible or the PCB can be damaged. Pipettes (implemented on most rework systems) can be used for large packages; tweezers may be more practical for small packages.

6.3 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-called "sticky" flux is used for this purpose). No-clean flux and solder paste is mandatory.

6.4 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.