Recommendations for board assembly of Infineon transistor outline type packages

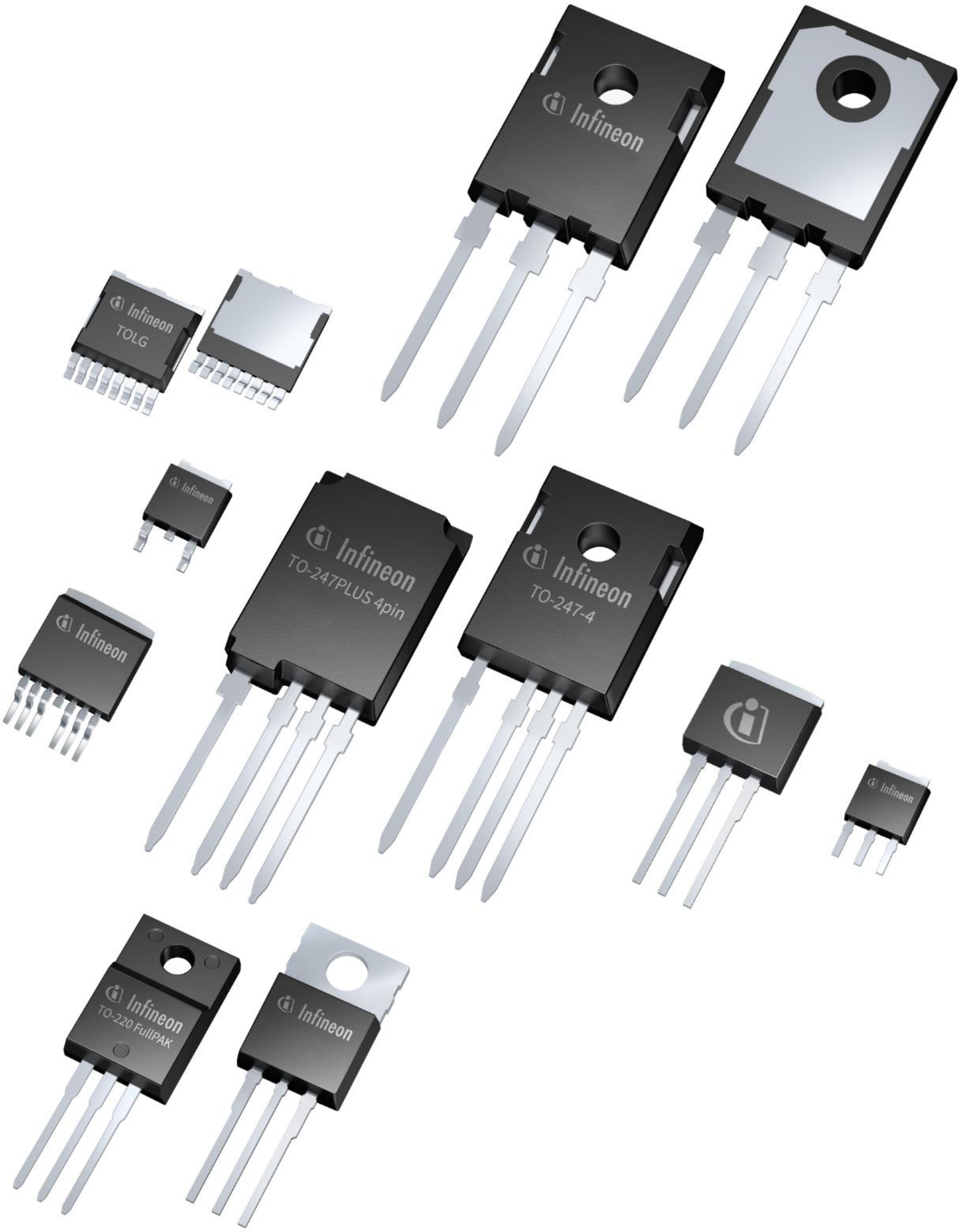

Table of contents

Table of contents 2
Acronyms and abbreviations 3
1 Package description 4
1.1 TO SMD package type 4
1.2 TO THD package type 5
1.3 HSOG package type 6
1.4 Package features and general handling guidelines 6
2 Printed circuit board 10
2.1 Routing 10
2.2 Pad design 10
2.3 Via-in-pad design 11
3 Mounting of surface-mount devices 13
3.1 Solder paste stencil 13
3.2 Solder paste 13
3.3 Component placement of SMD 14
3.4 Reflow soldering 14
3.5 Wave soldering of SMD 15
4 Mounting of through-hole devices 16
4.1 Pre-mounting processes and mounting materials 16
4.2 Component placement of THD 19
4.3 Heat sink mounting 19
4.4 Soldering of THD 23
4.5 Alternative mounting methods 24
5 Cleaning 25
6 Inspection 26
6.1 Optical solder joint inspection 26
6.2 X-ray solder joint inspection 27
7 Rework 28
8 References 29
Revision history 30

Acronyms and abbreviations

AOI	automated optical inspection
AXI	automated X-ray inspection
CSAM	c-scanning acoustic microscopy
DSO	dual small outline package
ESD	electrostatic discharge
HDSOP	heatspreader dual small outline package
HSOG	heatsink small outline gullwing
IGBT	insulated-gate bipolar transistor
I/O	input/output
K/W	Kelvin per Watts
MOSFET	metal-oxide-semiconductor field-effect transistor
MSL	moisture-sensitivity level
NSMD	non-solder mask defined
PG	plastic green
PCB	printed circuit board
SAC	tin silver copper (SnAgCu)
Si	silicon
SiC	silicon carbide
SMD	colder mask defined pad
SMD	surface-mount device
SMT	surface-mount technology
Sn	matte tin plating
TCoB	temperature cycling on board
THD	through-hole device
THT	through-hole technology
TO	transistor outline package
TOLG	transistor outline leaded with gullwing package
TOLT	transistor outline leaded top-side cooling package

1 Package description

This document provides information about the board assembly of Infineon transistor outline (TO) and heatsink small outline gullwing (HSOG) packages by surface mount technology (SMT) and through-hole technology (THT). TO packages are available as surface-mount devices (SMD) and through-hole devices (THD).

The I/O leads of the single-ended SMD packages are bent outwards from the package mold body side forming a distinct “foot” and “heel” geometry. That “gullwing” shaped lead geometry can be mounted on the same board surface together with the heat sink pad using fully automated processes.

The leads of THD are inserted in drilled holes of the board prior to soldering. Depending on the specific process and technology, this may require certain pre-mount processing steps. The heat sink of THD package can either be mounted on the board or can be equipped with an additional heatspreader.

The packages are optimized for silicon (Si) and silicon carbide (SiC) metal-oxide-semiconductor field-effect transistor (MOSFET), insulated-gate bipolar transistor (IGBT), and Schottky diode devices for automotive and industrial power applications and provide a high heat dissipation.

This document does not discuss dual small outline (DSO) or heatspreader dual small outline package (HDSOP) from so called transistor outline leaded top-side cooling (TOLT) package family. These package families are described in separate documents.

1.1 TO SMD package type

Infineon TO SMD packages are available as DPAK and D2PAK with with a mold body size of approx. 6 to 9 mm. Infineon designations are following the standardized names TO-252 and TO-263 as given by JEDEC. **Figure 1** shows examples of the TO SMD package family.

- PG-TO252 DPAK
- PG-TO263 D2PAK

PG = plastic green
TO = transistor outline

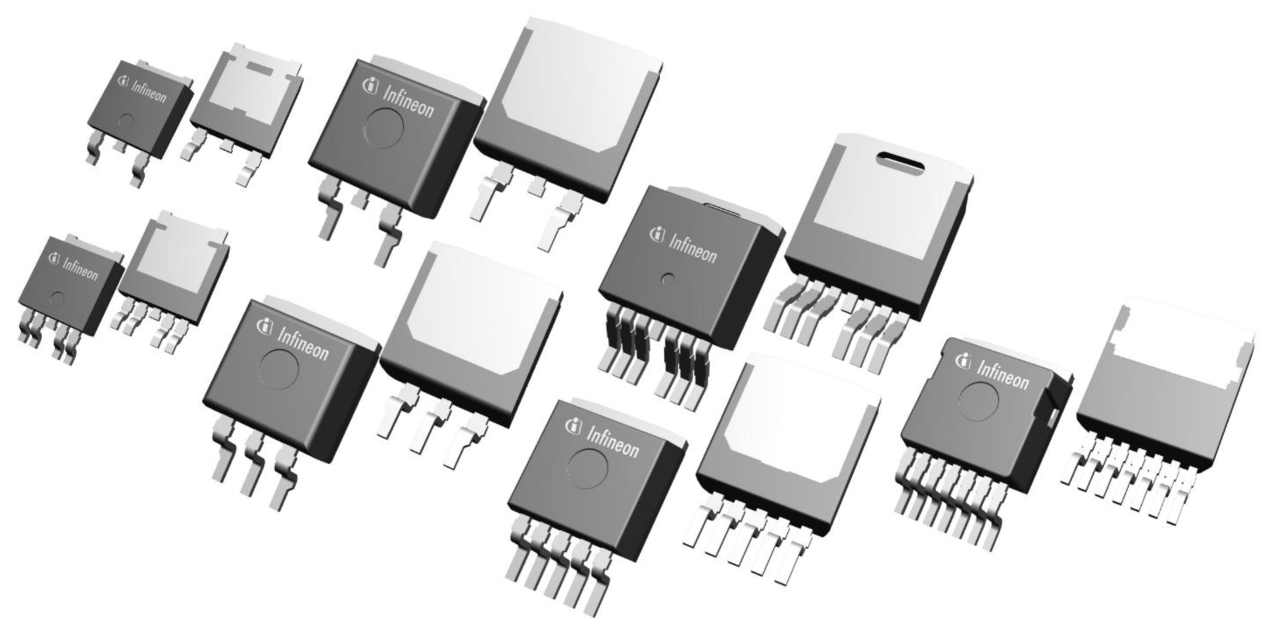

Figure 1 **Examples of surface-mount TO packages.**

1.2 TO THD package type

Infineon TO SMD packages are available in various versions such as IPAK, I²PAK and else. Fully isolated package versions without open metal lug are called “FullPAK”. There are package versions with screw mounting hole and without it featuring an increased heatsink area (“Plus” version). Some packages feature leads with extended or reduced length (“Long lead” and “Short lead” versions). Infineon designations are following the standardized names from TO-220 to TO-262 as given by JEDEC. **Figure 2** shows examples of the TO THD package families.

- PG-TO220
- PG-TO220 FullPAK
- PG-TO247
- PG-TO247 Long lead
- PG-TO247 Plus
- PG-TO251 IPAK
- PG-TO262 I²PAK
- PG-TO273 Super 220

PG = plastic green
TO = transistor outline

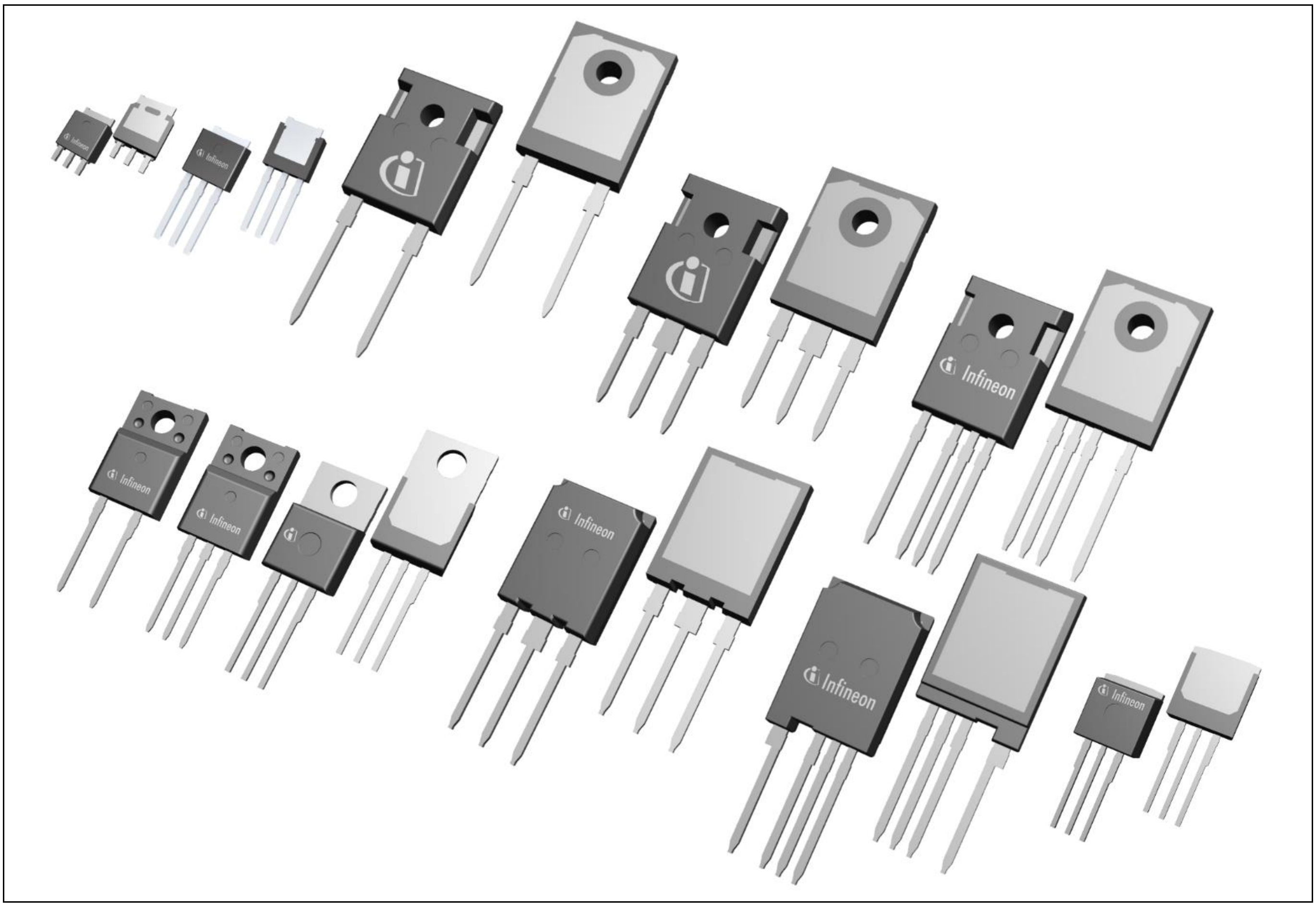

Figure 2 **Examples of through-hole TO packages.**

1.3 HSOG package type

Infineon heatsink small outline gullwing (HSOG) packages present a space optimized version of the TO263 (D2PAK) and are also called TO-Leaded with gullwing (TOLG). The tab thickness is reduced to decrease the junction to case thermal resistance while the height shrink increases the power density. The gullwing lead shape is preserved in contrast to leadless TO solutions in order to achieve high performance during temperature cycling on board (TCoB). **Figure 3** shows a HSOG package.

- PG-HSOG

PG = plastic green
H = heatsink
SO = small outline
G = gullwing

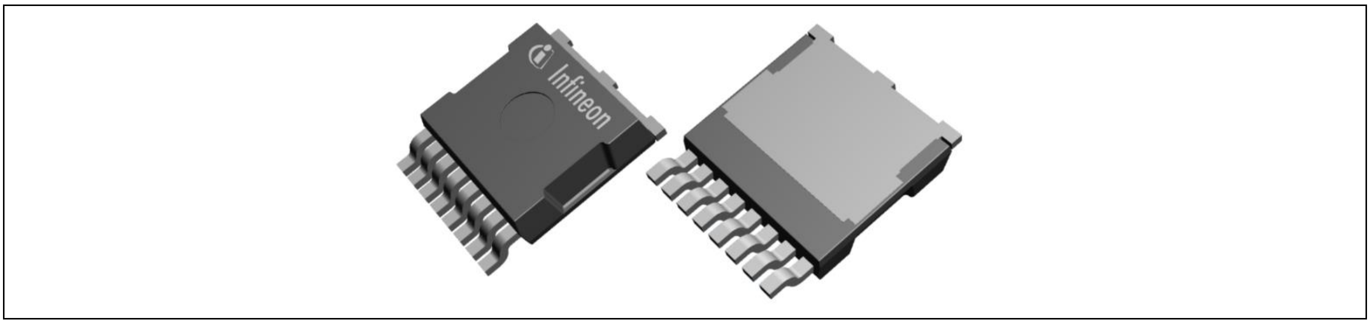

Figure 3 **Example of surface-mount HSOG package.**

1.4 Package features and general handling guidelines

General handling guidelines

Semiconductor devices are sensitive to excessive electrostatic discharge (ESD), 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 further information about component handling, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

Internal construction

The components of the TO SMD and HSOG package families feature plastic-encapsulated chips on a copper leadframe using single-ended perimeter leads. An example of the inner setup of TO SMD or HSOG packages is shown in **Figure 4**.

The TO THD components provide long strait leads for through-hole insertion. Large heat sink pads can protrude the mold body outline or can terminate within it. An example of the inner setup of TO THT packages is shown in **Figure 5**.

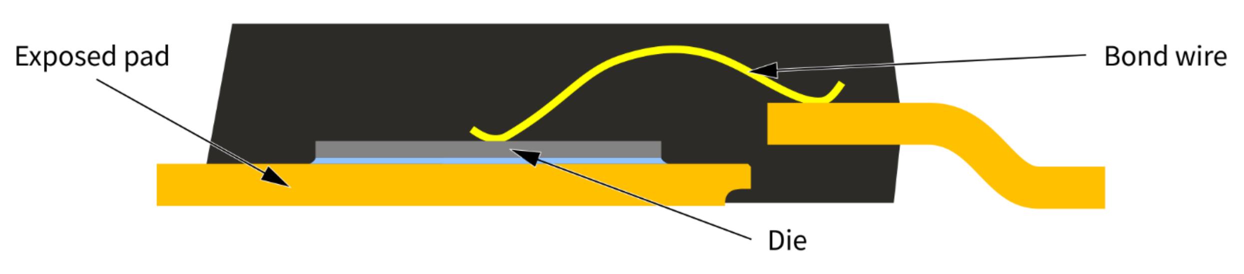

Figure 4 Schematic showing the inner setup of a TO SMD package.

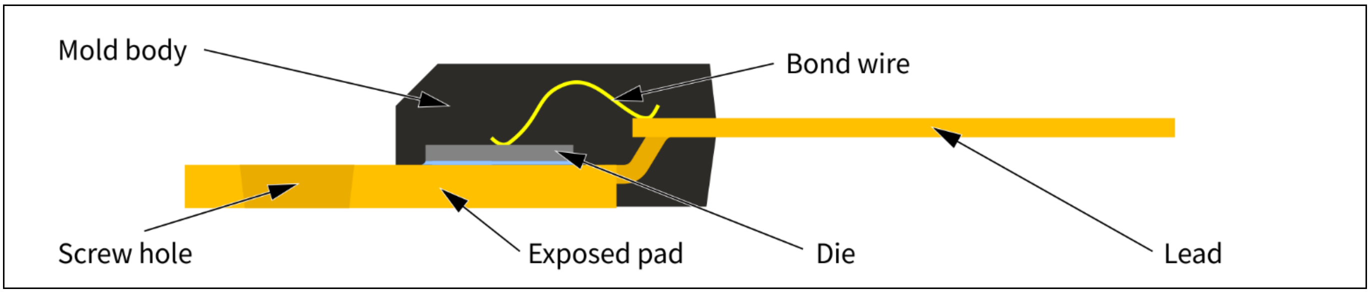

Figure 5 Schematic showing the inner setup of a TO THT package.

Thermal performance

The overall thermal performance of a package with a heat sink is characterized by a junction-to-ambient thermal resistance R_{thja}. The R_{thja} can be calculated as shown in **Figure 6** and **Equation 1**.

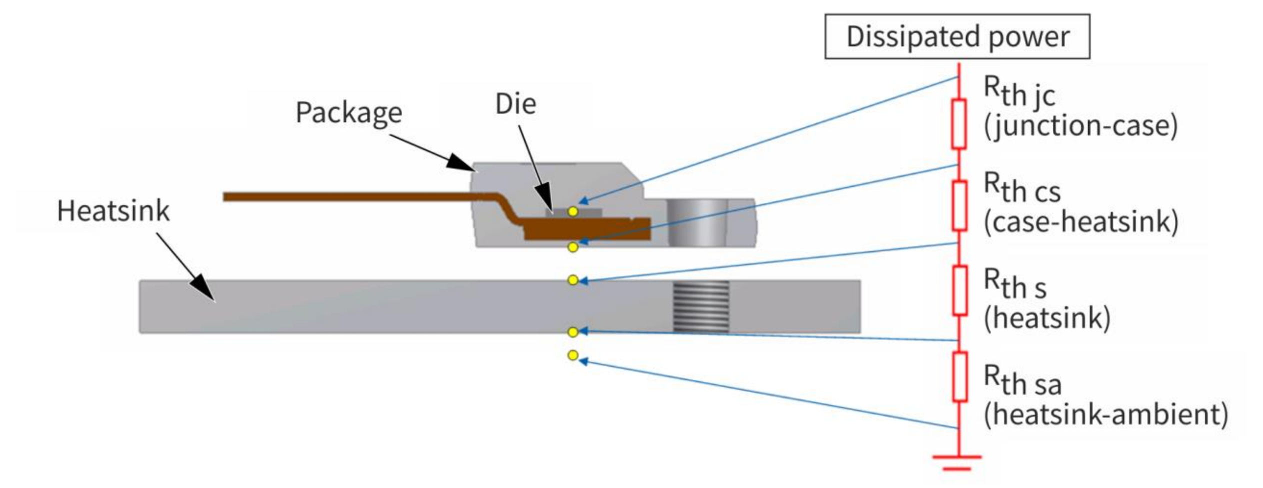

Figure 6 Thermal model of a TO package on a heatsink.

$$R_{th\,ja} = R_{th\,jc} + R_{th\,cs} + R_{th\,s} + R_{th\,sa} \; (1)$$

$R_{th\,ja}$ *thermal resistance junction to ambient (K/W)*

$R_{th\,jc}$ *thermal resistance junction to case (K/W) – specified in relevant datasheet*

$R_{th\,cs}$ *thermal resistance case to sink (K/W)*

$R_{th\,s}$ *thermal resistance sink (K/W)*

$R_{th\,sa}$ *thermal resistance sink to ambient (K/W)*

When mounting a package on an external heat spreader, it is important to consider the interface resistance R_{thcs}. In real application cases, R_{thcs} will exceed the ideal value of 0. That is caused by small air gaps between the package and the heat spreader that can have the following reasons:

- Surface roughness and waviness of package heat sink and external heat spreader
- Slight tilt between package heat sink and external heat spreader

For insulated packages such as the PG-TO220 FullPAK, the use of thermal grease is recommended to fill the air gap between the package and the heat sink. Measurements have shown that the usage of thermal grease reduces the interface resistance by 1.2 to 1.5 K/W.

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

For further information about thermal performance, please refer to the *Thermal performance of surface mount semiconductor package*s document that is available on the Infineon web page [2]. Please also feel free to contact your local sales, application, or quality engineer.

Termination design

The gullwing lead is considered to be one of the most reliable terminations for SMD. To form gullwing terminations, the leads are bent outwards at the tip. These bent input/output (I/O) lead foot and heel areas form the seating plane that is then soldered to the PCB. The leads as well as the heat sink pad feature a solderable surface for board mounting. The distance between the I/O lead seating plane and the exposed pad landing area at the package bottom is defined as the package stand-off. The single-ended gullwing lead row of TO SMD will result in a non-symmetric stand-off between the lead landing area and the heat sink pad.

The solder joints are mainly formed underneath the package. The tips of such terminations often feature bare copper (e.g. cut edges) and are therefore not intended to wet with solder by design according to IPC-A-610 [7].

Figure 7 Cross-section of a HSOG package lead with proper solder connection to board.

Termination plating

Infineon TO SMD and HSOG packages with gullwing leads feature a matte tin (Sn) surface finish that is applied to the base metal by a post-mold process. The solder connection is then made to the underlaying copper or nickel.

2 Printed circuit board

This section addresses specifically the surface-mount TO SMD and HSOG packages.

2.1 Routing

Printed circuit board design and construction are key factors for achieving solder joints with high reliability. Packages with exposed pads should not be placed opposite to each other on either side of a PCB when doing double-sided mounting. This will stiffen the assembly and cause solder joints to fatigue earlier than in a design in which the components are offset. Furthermore, the board stiffness itself has a significant influence on the reliability of the solder joint interconnect if the system is used in critical temperature-cycling conditions

2.2 Pad design

The quality and reliability of interconnect solder joints to the board are affected by:

- Pad type (solder mask defined, SMD or non-solder mask defined, NSMD)
- Specific pad dimensions
- Pad finish (also called metallization or final finish)
- Via layout and technology

For optimal heat dissipation in high-current applications, the SMD pad type is preferred beause it allows for large copper areas under the solder mask layer of the PCB. Beside the power application aspect, the SMD design type can also be beneficial in terms of routing flexibility. Mixing different pad definition types in one footprint is not recommended due to the typical alignment tolerances between the solder mask and the copper layer.

It is generally possible to layout the board pad with the same dimensions as the exposed die pad of the package. If the exposed die pad protrudes on one or more sides of the package body, it can be helpful to increase the board pad slightly in this area to improve self-centering of the component during reflow soldering. **Figure 8** shows examples of pad and stencil designs for a PG-TO263-7-2 package without and with vias in pad.

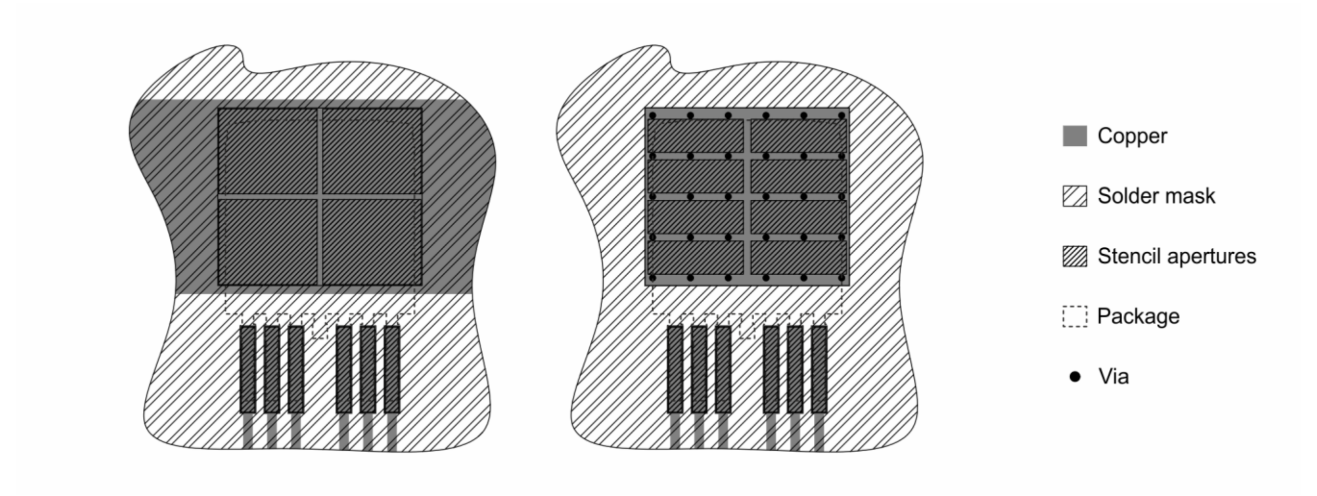

Figure 8 **Examples of a pad and stencil design for a PG-TO263-7-2 package without (left) and with (right) vias in pad.**

For reliable solder joint formation of a gullwing lead the PCB pad should feature a certain backward extension to the lead heel. According to the IPC-A-610 the minimum solder wetting height shall reach an extended line projected from the lead tip top corner in parallel to the PCB pad plane [6]. As a rule of thumb often the line parallel to the lead top plane is used to take into account the lower bend angle. The PCB pad and therefore the solder paste print should have a distinct distance to the package mold in order to avoid an unclean solder process as it is indicated by e.g. solder spatters.

An optimal PCB design depends on the specific application as well as on the specific design guidelines of the chosen board manufacturer.

For further information about PCB pad design, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

2.3 Via-in-pad design

Thermal and electrical connections to the inner and/or bottom copper planes of the PCB are usually created by plated through-hole vias in the board. The heat is then transferred from the chip over the package die pad and the solder joint to the thermal pad on the board and further through the PCB by the thermal vias.

The diameter and the number of vias in the thermal pad depend on the specific thermal requirements of the final product, the power consumption of the product, the application, and the construction of the PCB.
A typical hole diameter for thermal vias is 0.2 - 0.5 mm. An array with 1.0 - 1.2 mm pitch can be a reasonable starting point for further design optimization. The implementation of thermal vias has several impacts on the board assembly as outlined below. A constant increase of number of vias does not necessarily translate into a constant decrease of the thermal resistance of the entire assembly set-up. Thermal and electrical analysis and/or testing together with a proper board assembly design procedure are recommended to determine the optimal number of vias needed.

One of the primary exposed pad design objectives, besides the thermal management, should be to avoid the penetration of the vias by solder. Consequences of such solder wicking can be a decreased stand-off between the PCB and the package, an increased void formation ultimately resulting in an insufficient solder joint area, or surplus solder on the opposite side of the PCB.

A first approach for risk reduction should be the prevention of a direct print of solder paste on the via opening. The stencil for large area prints such as on die pads is usually segmented. It is a good practice to position the vias under the stencil sheet beams as shown in **Figure 8**. With such an approach, a good solder joint on a die pad can be formed using vias that remain open on both sides of the board.

Despite this precautionary stencil design approach the solder can move into the via, driven by the wetting forces. If the solder then protrudes to the opposite side of the PCB, it may interfere with a second solder paste print process. To minimize the effect, wettable dummy areas surrounding the via opeing on the opposite side of the board can catch the surplus solder to avoid beading and solder lumping.

In case the solder variance in volume below the die pad is too high due to the wetting of vias, they can be closed by “tenting.” This process includes covering the vias by a solder mask (e.g. dry-film solder mask). If the via tenting is done only on the opposite side of the board, the voiding rate will increase significantly. Another method to close vias is called “plugging” (filling with epoxy), followed by overplating. Very small vias (100 µm in diameter or smaller) should be filled with copper and overplated. In both cases, the specification of a planar filling is necessary to avoid cavities that will trap gases, forming voids during reflow soldering.

In case it is not necessary to provide a direct connection from the solder pad under the exposed die pad to the inner layers of the PCB, the vias can be placed next to the footprint near the package and covered with solder mask.

For further information about vias in pad, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3 Mounting of surface-mount devices

The following factors have to be taken into account to achieve the best assembly quality for a given application:

- PCB design
- Footprint and stencil layout
- Solder paste formulation
- Solder paste application and inspection
- Component placement
- Reflow soldering process, especially the reflow profile

3.1 Solder paste stencil

In SMT 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. While an excessive solder paste volume will cause solder bridging, an insufficient solder paste volume can lead to reduced solder spreading between all contact surfaces. To ensure a uniform and sufficiently high solder paste transfer to the PCB, laser-cut (mostly made from stainless steel) are preferred.

In most cases, the thickness of a stencil has to be matched to the needs of all components on the PCB. For typical TO SMD packages, stencils with a thickness of 150 µm are recommended. Extending the solder pads together with the print can provide a sufficiently high solder volume to overcome the typical single-ended package stand-off.

A typical stencil aperture print reduction for TO leads is approx. 90% of the landing pad size. The solder paste volume in apertures larger than approximately 5 mm may be scooped out depending on the specific squeegee pressure and rigidity. Such apertures are necessary for many die pad prints and should be segmented into smaller areas. When reducing the die pad print, potential vias in the pad can be considered as outlined in the relevant section above (see **Figure 8**).

For individual design adaptations to reach the optimum amount of solder, the stencil thickness, the PCB pad finish, quality and solder masking, the via layout, and the solder paste type should be considered. In every case, application-specific experiments are recommended.

Further details and specific stencil aperture recommendations can be found in the package data base that is available on the Infineon web page [1]. Please choose a specific package when searching the data base, which will then show an example of the stencil aperture layout for each package.

For further information about solder stencil design, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3.2 Solder paste

Pb-free solder pastes typically contain some type of SnAgCu alloy (SAC solder with typically 1-4% Ag and <1% Cu). The most common alloy is SAC305 (3.0% Ag and 0.5% Cu). The average alloy particle size must be suitable for printing the solder stencil aperture dimensions. Using Type 3 or Type 4 paste is recommended for the board mounting of TO SMD, depending on the specific stencil aperture size and therefore solder paste transfer efficiency.

The solder alloy particles are dispersed in a blend of liquid flux and chemical additives (approx. 50% by volume or 10% by weight), forming a creamy paste. The flux and chemical solvents have various functions such as adjusting the viscosity of the paste for stencil printing or removing contaminants and oxides on the surface.

The solder paste solvents have to evaporate during reflow soldering, while residues of the flux will remain on the joint. The capacity of the flux additive for removing oxides is given by its activation level, which also affects the potential need for removing the flux residuals after the assembly. For leadless packages in which the solder joint is formed mainly on the package bottom side, a “no clean” paste is recommended to avoid subsequent cleaning steps underneath the package. Small gaps make cleaning highly difficult if not impossible. Certain precautions have to be taken if any kinds of flux residues remain on the board prior to any kind of coating. For power packages, leakage currents and the potential for shorting below components have to be considered when choosing the specific flux type (e.g. halide-free vs. zero halides).

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

3.3 Component placement of SMD

The pick-up nozzle should be chosen according to the package size. The nozzle should be slightly smaller than the package mold body.

For TO SMD packages, most of the time the nozzle position needs to be in the center of the mold body and not in the center of the overall package outline. That is due to the fact that TO SMD packages typically are single-end leaded.

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. An automated pick-and-place machine is recommended to obtain reliable solder joints.

Component placement accuracies of +/-50 µm and less 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 prior to the mounting process.

For further information about component placement, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3.4 Reflow soldering

For PCB assembly of the TO SMD packages, the widely used method of reflow soldering in a forced convection oven is recommended. Soldering in a nitrogen atmosphere can generally improve the solder joint quality but is not necessary to create a reliable joint.

Typical most PCB assemblies contain various package types and sizes. The the reflow settings have to match the allowed profile range of all the components and materials. Components with large thermal masses do not heat up at the same speed as lightweight components. In addition, the position and the surrounding of the component on the PCB, as well as the PCB thickness, can influence the solder joint temperature significantly. It is therefore recommended to measure the temperatures at critical positions by thermocouples.

Power packages where leakage currents and shorting below the component have to be considered should be soldered with decreased flux spreading. Therefore, it is recommended to optimize the reflow profile in such a way that excessive flux or solder spattering is avoided. The soldering profile should be in accordance with the recommendations of the solder paste manufacturer to achieve optimal solder joint quality.

Minimum reflow conditions

The lower temperatures and durations of an optimal reflow profile must stay above those of the solderability qualification. The solderability of the terminations of Infineon components is tested according to the standards IEC 60068-2-58 and J-STD-002 [2][4].

Maximum reflow conditions and cycles

Components that are moisture-sensitivity level (MSL) classified by Infineon have been tested by three reflow runs in accordance with the J-STD-020 standard, including a double-sided reflow and one rework cycle. The maximum temperatures must not be exceeded during board assembly. Please refer to the product barcode label on the packing material that states this maximum reflow temperature according to the J-STD-020 [5] standard as well as the MSL according to the J-STD-033 standard [6].

Typical Infineon TO SMD packages are generally suited for mounting on double-sided PCBs. Solder joints of components on the first PCB side will reflow again in the second step. In the peak zone of the reflow profile (i.e. where the solder is liquid), the components are only held in place 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 will force the components closer to the PCB surface. This shape will be frozen during cooling and therefore will 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. That can be prevented by applying an appropriate SMD adhesive prior to the reflow.

For further information about reflow soldering, please refer to *the General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

3.5 Wave soldering of SMD

In general, Infineon TO SMD packages are not designed to withstand a wave soldering process in which the whole package body will be immersed in the molten solder. The application of an appropriate SMD adhesive prior to the soldering should be considered. In case wave soldering is applied to TO SMD, individual investigations on a per-product basis are necessary.

For further information about wave soldering of TO SMD packages, please contact your local sales, application, or quality engineer.

4 Mounting of through-hole devices

The following factors have to be taken into account to achieve the best quality assembly for a given application:

- Insulating material (e.g. washers, spacers)
- Screws
- Thermal grease
- Mounting torque
- PCB and heat sink
- Attachment holes for heat sink screw mounting
- Soldering method, especially temperature profile

4.1 Pre-mounting processes and mounting materials

Through-hole packaged parts are mostly supplied with the leads projecting straight out of the mold body. Many practical power circuits use bulky heat sinks in contact with the device tabs to enhance thermal performance. This may preclude the straight-leaded orientation arrangement of the standard part. Consequently it is common to change the lead length and direction to make a more convenient electrical connection on an adjacent printed circuit board. This section is intended to address the frequently asked package related question of how the leads of TO THD may be safely bent without endangering part reliability.

Lead cutting

The leads of THDs can be cut before mounting or after mounting and soldering. In case of cutting after soldering, the leads are embedded in the solder joint. Severe mechanical mishandling during the cutting process could cause solder joint failure, but generally does not harm the component body. When cutting prior to mounting, the leads have to be fixed by a clamp that should be similar to a clamp used for lead bending as outlined below.

Lead bending

In order to insert the terminals of THDs in the holes of the PCB, the terminals may have to be bent to match the pitch of the respective PCB holes for various reasons.

The original straight alignment of the leads (gate, drain, source of a transistor package) may not provide an adequate electrical clearance and creepage distance between the track pads. By offsetting the centre leg this problem may be avoided.

For some mechanical arrangements the heat sink and the lead termination point may be subjected to relative movement. In these cases where these forces are unavoidable, it is desirable to introduce a stress relieving lead bend in order to reposition this stress. With a simple bend of this sort, the stresses which would normally have degraded either the lead-to-body joint or the lead-to-PCB joint, will be absorbed along the lead length.

The different concepts are shown in **Figure 9**.

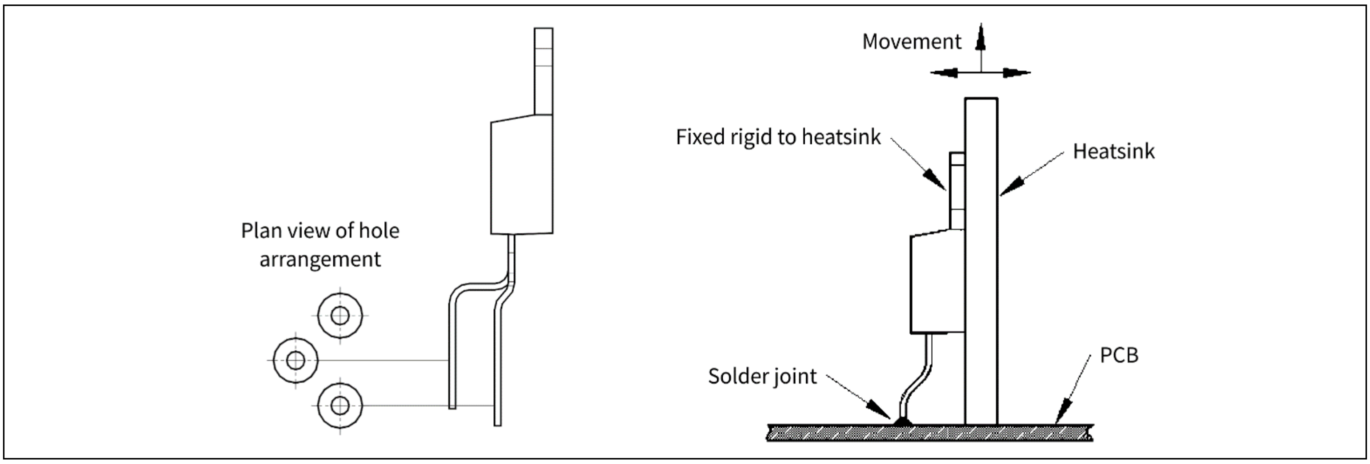

Figure 9 **Lead bending for mounting hole offset compensation (left) and heatsink to lead arrangement for compensation of relative movement (right).**

When bending the leads it is very important to comply with the following guidelines:

Clamping

- Clamping, bending and cutting need to be done sequentially in that order as shown in **Figure 10**; a subsequent process is not allowed to be initiated before the preceding process is finished.
- A properly designed clamping tool supports a reproducible bending shape. In general, the larger the clamp area the more reliable the quality of the lead forms.
- The clamp shall be designed to avoid excessive mechanical forces such as pulling and shearing between the leads and the package body. Between the point of bending and the package edge, tensile stress during the bending process needs to be relieved. Avoid slippage due to weak clamping, as well as weakening of the lead due to overly strong clamping.
- Under no circumstances must the plastic body be held or restricted while lead forming as this has the potential to mechanically damage the package-particularly at the metal-to plastic interface. In case of a package containing an isolated backside (e.g. FullPAK packages, TO247 Advanced Isolation) it has to be taken into consideration not to damage the isolation in any of the tools used for pre-mounting processes.
- In case the package has an increased lead width next to the package body (cut dam bar), the bending distance X needs to be higher than the length of area with the increased lead width L1 as shown in **Figure 11**. If the there is no increased lead with by a cut dam bar the minimum distance between the package body and the first bend should be 2.5 mm as shown in **Figure 10**.
- Bending the leads parallel to the lead plane or immediately adjacent to the package as shown in **Figure 10** is not allowed.

Radius

- The general rule of thumb is to create a bending radius R that is one to two times the material thickness T as shown in **Figure 11**. It is recommended that the bend radius should never be less than the thickness of the lead material.
- The minimum bend radius R needs to be 0.5 mm.

Force

- The maximum force applied to the leads between the clamp and the point where the bending force is applied should not exceed a certain limit. This maximum force is mainly dependent on the cross-sectional area of the lead. A typical maximum force is 20 N.

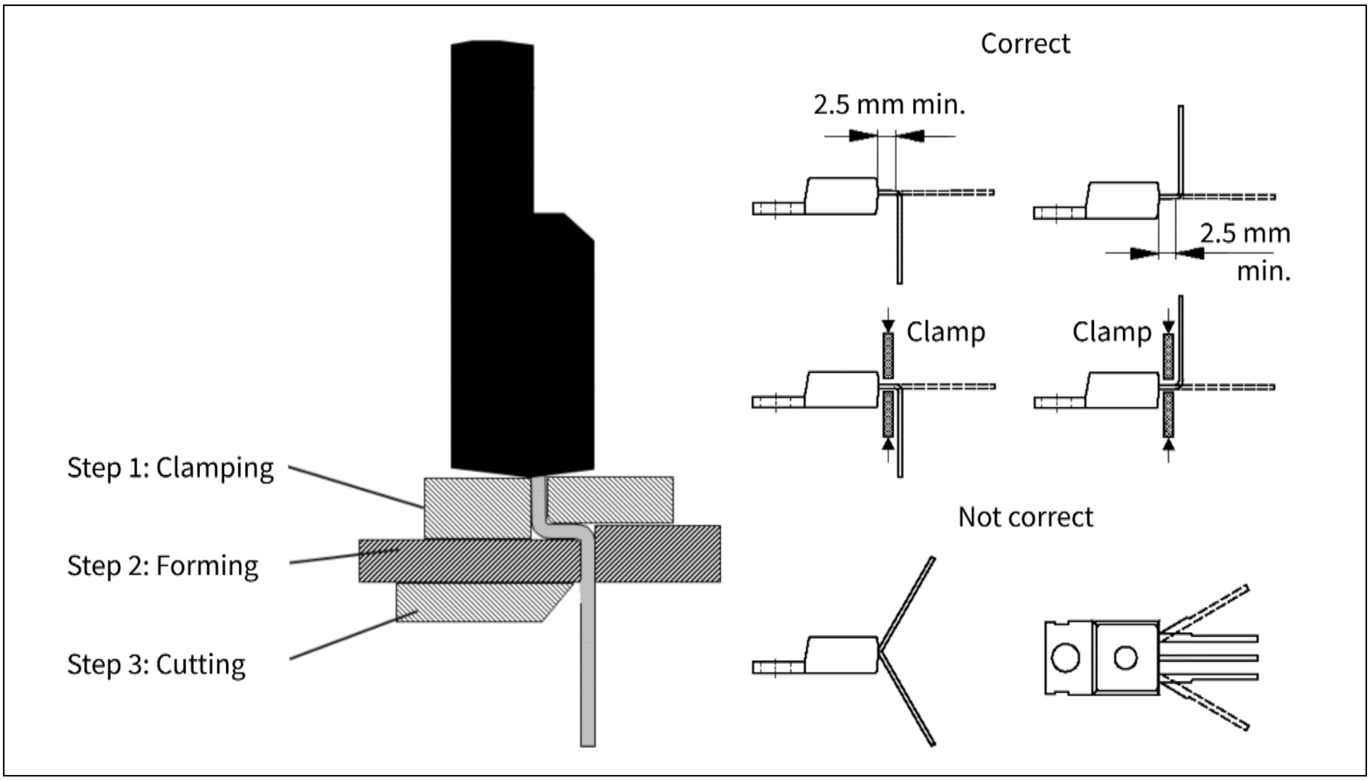

Figure 10 Lead clamping, forming, and cutting tool (left) and examples of correct and incorrect lead clamping and bending of THD packages (right).

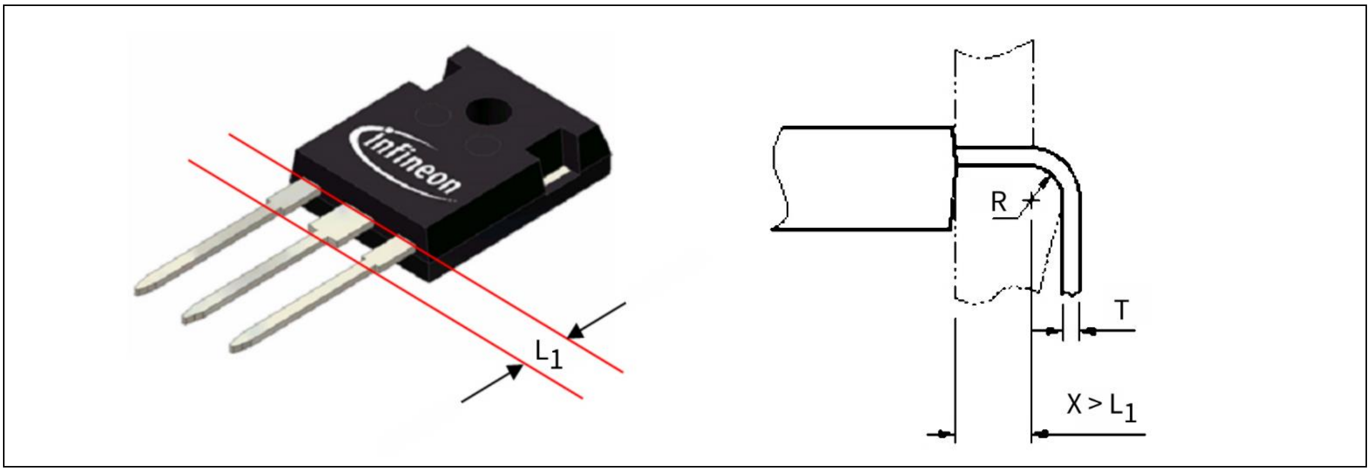

Figure 11 Parameters for recommended minimum clamping distance and radius for lead bending.

Manual bending

Manual bending is generally not recommended. In case it is considered, the guidelines above must be followed.

For further information about TO THD pre-mounting processes, please refer to the individual product *Application Note* documents and contact your local sales, application, or quality engineer.

4.2 Component placement of THD

THDs are inserted either with special automatic equipment or manually. During this insertion, special care has to be taken that excessive deformation or violent bending is avoided. The diameter of the drill holes in the PCB has to be appropriate for the tolerances of component leads, drill hole position placement accuracy, and properties of the solder alloy used.

4.3 Heat sink mounting

For special packages with high power dissipation, heat sinks can be mounted before or after soldering the leads.

Insulating material

Commonly used insulating materials are mica and rubber. Mica is much harder than rubber. For softer materials, the mounting torque in particular needs to be controlled very carefully in order to avoid high mechanical stress and damage to the package as shown in **Figure 12**.

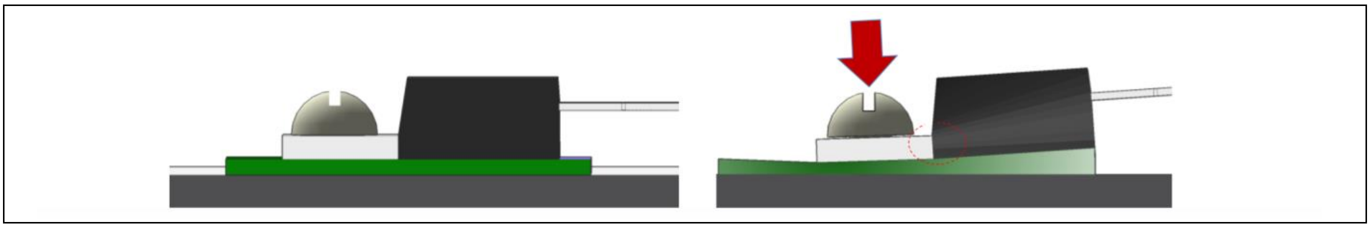

Figure 12 **Schematic view of the influence of the insulating material (left mica, right rubber) on the generation of mechanical stress.**

Thermal grease

A specific amount of grease must be applied. The amount should be determined during customer process evaluation. When the amount is correct, a very small amount of grease should appear around the perimeter of each surface as the assembly is slowly torqued to the recommended value. Examination of a demounted assembly should reveal even wetting across each surface.

Heat sink requirements

The contact areas of the package and the heat sink must be free of any particles and damage as well as any other contamination. The following surface conditions are recommended:

Heat sink roughness	$R_Z \leq 10$ µm
Heat sink flatness	≤ 10 µm (reference length 15 mm)

Attachment holes

Drill holes for heat sink screw mounting should be as small as possible. Using drill holes that are too big will reduce the effectiveness of the heat sink.

The flatness and effectiveness of the heat sink will also be reduced by intrusions and/or burrs around the hole. These are caused by the punching or drilling process and should be less than the specified heat sink roughness and flatness.

If intrusions or large drill holes are unavoidable, a fitting square washer should be used to get a good and flat contact between heat sink and package.

Heat sink screw mounting

Screw mounting is a traditional assembly method using a screw, nut and washer, following these guidelines:

- Self-tapping screws should not be used.
- It is recommended that a plane washer is inserted between the screw head and the mounting tab. Care must be taken to ensure that the washer does not damage the plastic body of the package during the mounting process.

The screw should be properly tightened in a controlled process to ensure that the package makes good contact with the heat sink.

Figure 13 illustrates the correct mounting components for a TO220 and a TO220 FullPAK. The same concept is applicable to TO247 packages.

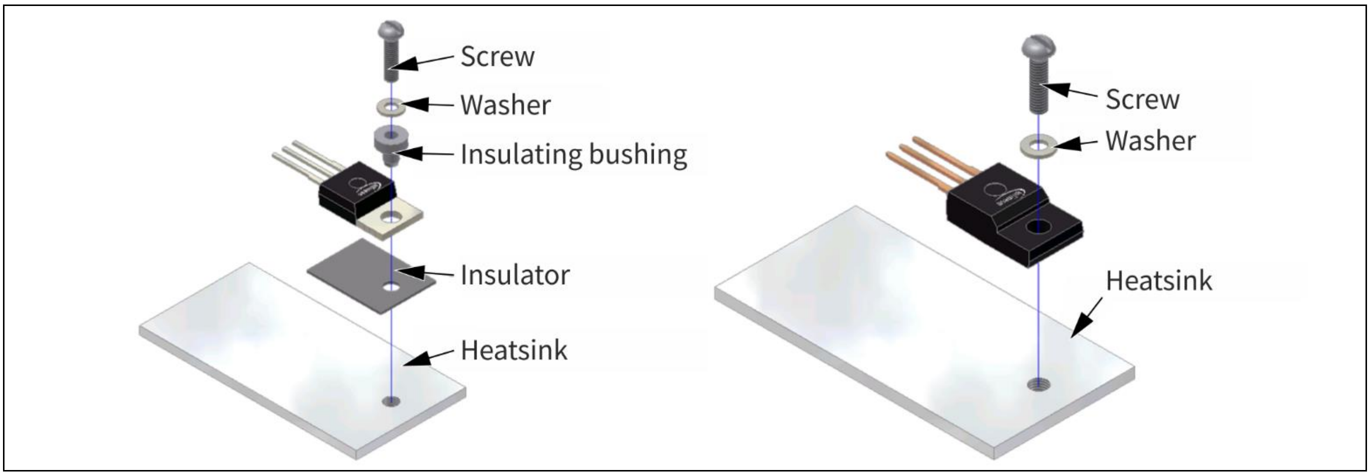

Figure 13 Screw-mounting concept, examples TO220 (left) and TO220 FullPAK (right).

Table 1 Package specific screw mounting concepts. The concepts are independent from the number of leads of the respective package.

Package	Insulator	Thermal grease	Insulating bushing	Washer
TO220	Yes	No	Yes	Yes
TO220 FullPAK	No	Yes	No	Yes
TO247	Yes	No	Yes	Yes
TO247 Advanced Isolation	No	No	No	Yes

Mounting torque

The thermal contact resistance between the two surfaces is maximized by increasing the contact pressure between the package and its heat sink. Increasing the mounting torque in the fastening screw, or using a clip with a high spring constant will result in lower contact resistance areas that provide solid conduction heat flow paths compared to air conduction through the air gap.

Applying the proper mounting torque is the key factor in obtaining an adequate contact pressure along the contact surfaces of the package and the heat sink to minimize the contact thermal resistance. If the mounting torque is too low, the contact thermal resistance increases due to poor thermal connection under insufficient contact pressure. If mounting torque is too high, the package head and mounting tab are deformed so that the package may be lifted away from the heat sink as shown in **Figure 14**, also increasing the contact thermal resistance. Hence, an appropriate range of mounting torque values must be used to ensure minimal thermal resistance. This prevents package destruction or changes in thermal and electrical device characteristics.

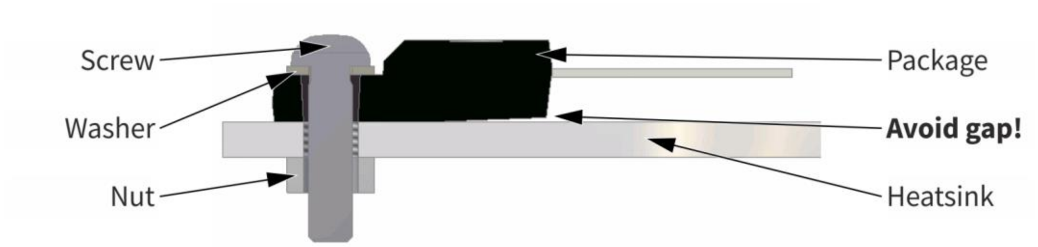

Figure 14 **Example of a heat sink assembly with screw mounting.**

Improper mounting can damage a die. Molding compound can delaminate due to external mechanical stress, e.g. mounting on warped heat sink a shown in **Figure 15**.

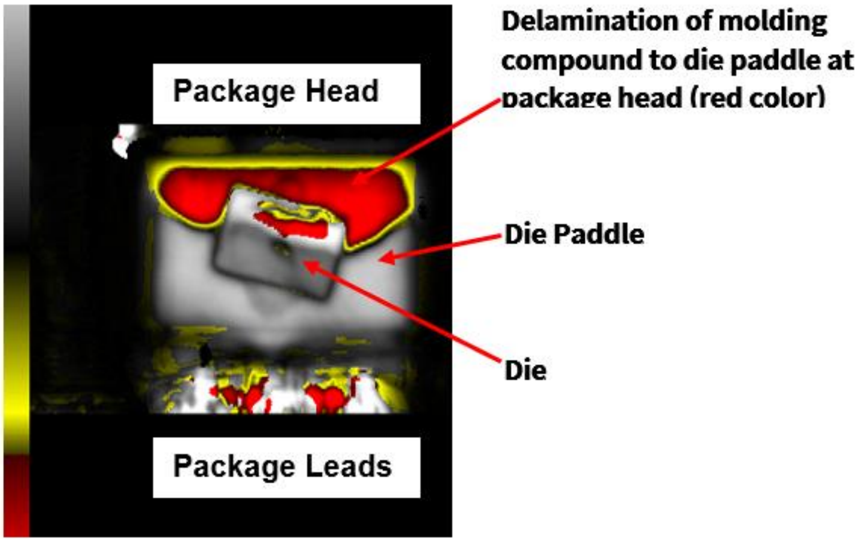

Figure 15 **C-scanning acoustic microscopy (CSAM) image showing the delamination of molding compound to die paddle interface at the package head (TO220).**

A maximum torque used for heat sink mounting of 1.0 Nm is recommended. Higher mounting torque does not directly result in lower R_{th} values as can be seen in the graph from **Figure 16** but may at the worst cause package damage depending on the insulation material used.

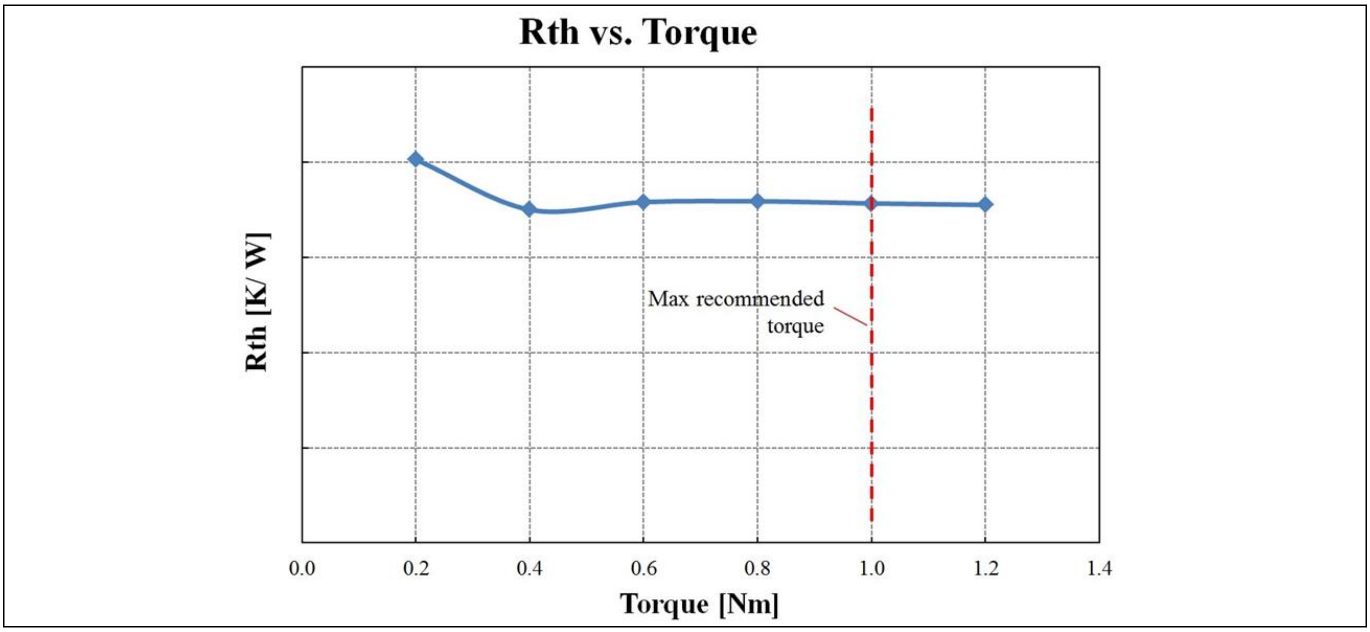

Figure 16 Effect of torque on the R_{th} value between package and heat sink.

Double sided heat sink mounting

Double-sided heat sink mounting is not recommended. In case of unsuitable mechanical heat sink properties, high mechanical stress may occur, resulting in package damage as shown in **Figure 17**.

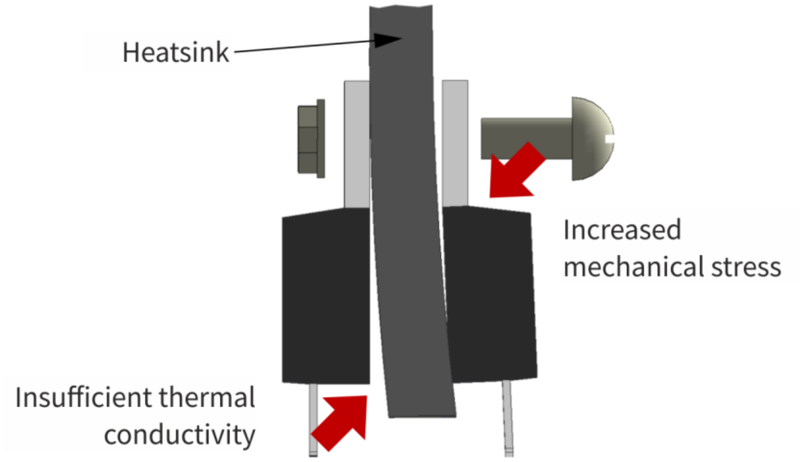

Figure 17 Negative effects of double-sided assembly on improper heat sink material.

Heat sink clip mounting

The clip mounting method has become popular because it is simple and reliable. The process is fast and appropriate for mass production assembly. The screw mounting is applied at one end of the package during fastening and can cause uneven contact pressure and bad thermal contact, whereas the clip is mounted on the package center and therefore results in more uniform contact pressure as shown in **Figure 18**.

In case clips are used, the contact area between the plastic case and the clip is important. Clips should be round or smooth in the contact area to avoid concentrated loads on the plastic body of the package.

For some applications it is necessary to have the package electrically isolated from the heat sink or PCB.

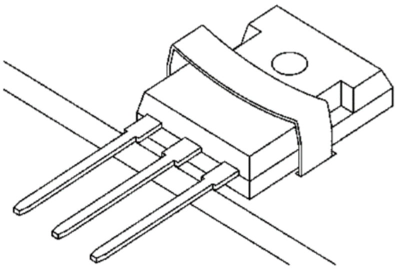

Figure 18 **Schematic drawing of TO247 package with clip mounting.**

4.4 Soldering of THD

THDs are designed for wave soldering and not for reflow soldering. As the number of THDs on each board has been decreasing in some applications wave soldering, however, becomes less cost effective. The few remaining THDs (mostly connectors and special components) are soldered with selective or wave soldering, or with the pin-in-paste technique plus reflow soldering.

TO THD are often mounted on a carrier or submount that has special features which enable it to serve as a good heat sink. In this case, special processes are needed to attach the component.

Wave soldering and selective soldering of THDs

These processes use a tank holding molten solder. The components are inserted into the PCB and the loaded PCB is passed across a pumped wave or cascade of solder. The solder wets the exposed metallic areas of the board (those not protected with solder mask), creating a reliable mechanical and electrical connection.

In the case of TO THD, only the leads that reach through the drilled holes in the PCB are directly in contact with the hot solder. The body of the package only gets heated by the hot leads. As a result, the package body is cooler than in case of reflow soldering, and the temperature gradient between leads and body and inside the package is steeper compared to reflow soldering.

The heat resistance of wave-solderable TO THD is tested according to JESD22-B106 and IEC 60068-2-20 (typically 260°C for 10 s). This gives the maximum acceptable temperature and time for selective/wave soldering (e.g. for dual wave: max. time in wave 1 + max. time in wave 2 < 10 s) [8][9].

Note: *Immersing the whole TO THD package body into the molten solder is not recommended. This would result in a temperature shock of the package body, which is not rated for this stress.*

There are various types of wave soldering machines available. The basic components and operating principles of these machines are the same. A standard wave solder machine has three zones: the fluxing zone, the preheating zone, and the soldering zone. An additional fourth zone, cleaning, may be used depending on the type of flux applied. Dual-wave soldering is the most commonly used wave soldering method.

The peak temperatures, ramp rates and times depend on the materials used and on the wave soldering equipment. The first wave has a turbulent flow and therefore guarantees wetting of nearly all shapes of leads and board pads, but also results in a large number of solder bridges. These solder bridges must then be removed by the second, laminar wave. When using Pb-free solder alloys, a nitrogen atmosphere is recommended.

Selective soldering is typically used when only a few THD components have to be mounted. Generally this is done after the other components are already soldered by reflow soldering. These components must be protected while undergoing selective soldering. This protection can be achieved either by using special fixtures and deflectors for the PCB or/and by creating a small wave shape in the machine by using special wave-guiding tubes or covers.

4.5 Alternative mounting methods

Further techniques beside the previously described wave and reflow soldering methods might be used in special applications. Examples include selective soldering, laser welding and laser soldering, hot bar soldering and manual soldering with a soldering iron and hot air guns.

Some general guidelines should be followed for a wide variety of soldering techniques that cannot be tested for every component:

- The maximum temperature of the package body and leads must not exceed the maximum allowed temperature for reflow or wave soldering.
- The maximum allowed time at high temperatures must not exceed the maximum allowed time for reflow or wave soldering.
- If heat is applied to the leads, the maximum temperatures in the package and of the package body must not exceed the maximum allowed temperatures during reflow or wave soldering.

If extended contact and heating times are unavoidable, the resulting temperature on various leads near the package body should be measured and must not exceed the temperature and duration experienced during wave or reflow soldering.

Pin-in-paste

Another technique used to solder THDs is to print solder paste onto a PCB near or over drill holes through which the leads are then inserted. The reflow of the solder paste is done together with soldering the surface mount components and therefore they have to go through the reflow temperature profile. Consequently, the temperature is nearly the same for the whole package in contrast to wave soldering, and the time for which the peak temperature is applied to the package is much longer compared to wave soldering.

Heat sink mounting by reflow soldering

In special applications the heat sink of high-power THD can be mounted to the board by solder paste printing, pick-and-place, and reflow soldering. In this case, the product undergoes a reflow profile cylcle.

Note: *TO THD are qualified for wave soldering and not for reflow soldering. Therefore it is not recommended to use reflow soldering for heat sink mounting of TO THD.*

For further information about alternative mounting techniques, please refer to the individual product *Application Note* documents and contact your local sales, application, or quality engineer.

5 Cleaning

After the soldering process, some flux residues may remain on the board, especially near the solder joints. Generally, cleaning beneath a SMD component is difficult due to the small gap between the component body and the PCB. Therefore, a “no-clean” flux is recommended whose residues usually do not have to be removed after the soldering process.

In case the solder joints have to be cleaned, the cleaning method (e.g. ultrasonic, spray, or vapor cleaning) and cleaning solution have to be selected while taking into account the type of package, the flux used in the solder paste (rosin/resin-based, water-soluble, etc.) as well as the environmental and safety aspects. Even small residues of the cleaning solution should be removed or dried out very thoroughly.

For recommended cleaning solutions, please contact the solder paste or flux manufacturer.

6 Inspection

6.1 Optical solder joint inspection

Irrespective of the specific geometry of a TO SMD gullwing lead, the solder joint is generally considered to be of good quality when the heel region is wetted up to a certain height and additionally the sidewall is sufficiently covered by solder. **Figure 19** and **Figure 20** shows optical side-views of two properly soldered packages with gullwing leads of different geometries. The tips of the gullwing leads have bare copper (e.g. cut edges) that is not intended to wet by design according to the IPC-A-610 [7].

The visual inspection of the solder joints of the outer, gullwing-shaped terminations with conventional automated optical inspection (AOI) systems is a standard procedure. According to IPC-A-610 the top side of a gullwing lead as well as the cut edges at the toe are not taking part in the solder joint formation [7].

For engineering tasks, cross-sectioning can provide detailed information about the solder joint quality. Due to its destructive character, cross-sectioning for monitoring purposes is naturally not practical.

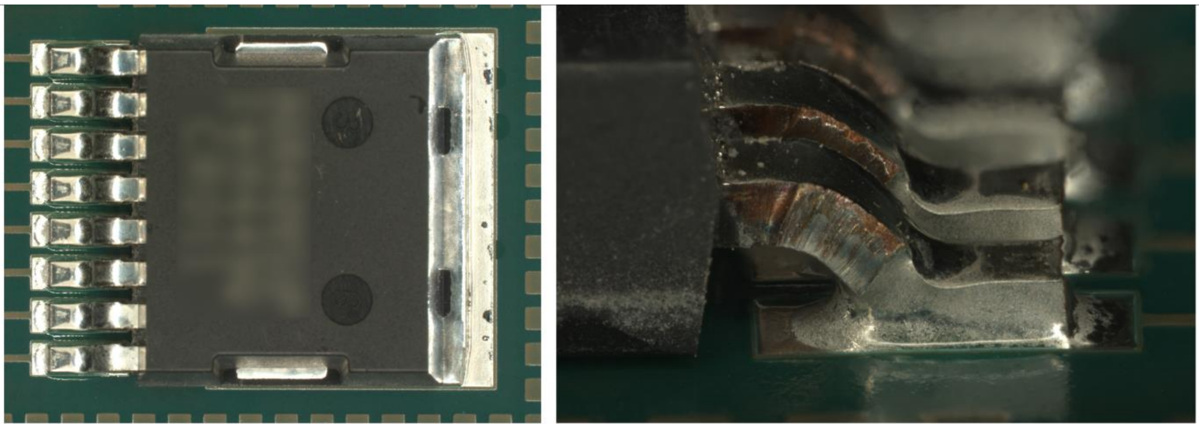

Figure 19 Example of a properly mounted HSOG-8 package.

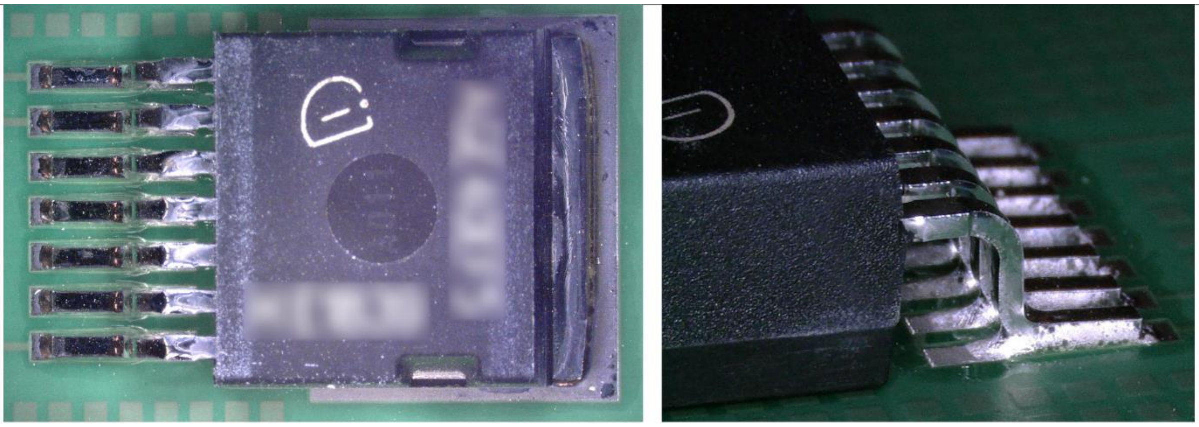

Figure 20 Example of a properly mounted TO263-7 package.

In order to reach an optimal TO THD solder joint, it has to be assured that a metallized via is filled properly. This cannot be detected by visual inspection, only by X-ray and/or examining cross sections.

For further information about the acceptability of electronic assemblies inspected optically, please also refer to the IPC-A-610 document [7].

6.2 X-ray solder joint inspection

Automated X-ray inspection (AXI) systems are appropriate for efficient inline control of component parts that cannot be inspected properly by optical systems (such as exposed pads). AXI systems are available as 2D and 3D solutions. They usually consist of an X-ray camera and the hardware and software needed for inspection, controlling, analyzing, and data transferring 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 21 shows typical X-ray photographs of HSOG-8 and TO263-7 SMD components. The top and inner layers of the boards as well as vias in the pad are visible. Large exposed pads may tend to increased voiding because they do not provide a sufficient ratio between volume and surface necessary for proper outgassing of the organic paste compounds during reflow. Generally, the extent of voiding depends on the board pad size, the via and stencil layout, the solder paste, and the reflow profile. For thermal evaluations, the entire thermal path must be considered as well as all boundary conditions such as the application environment or the electrical use of the component

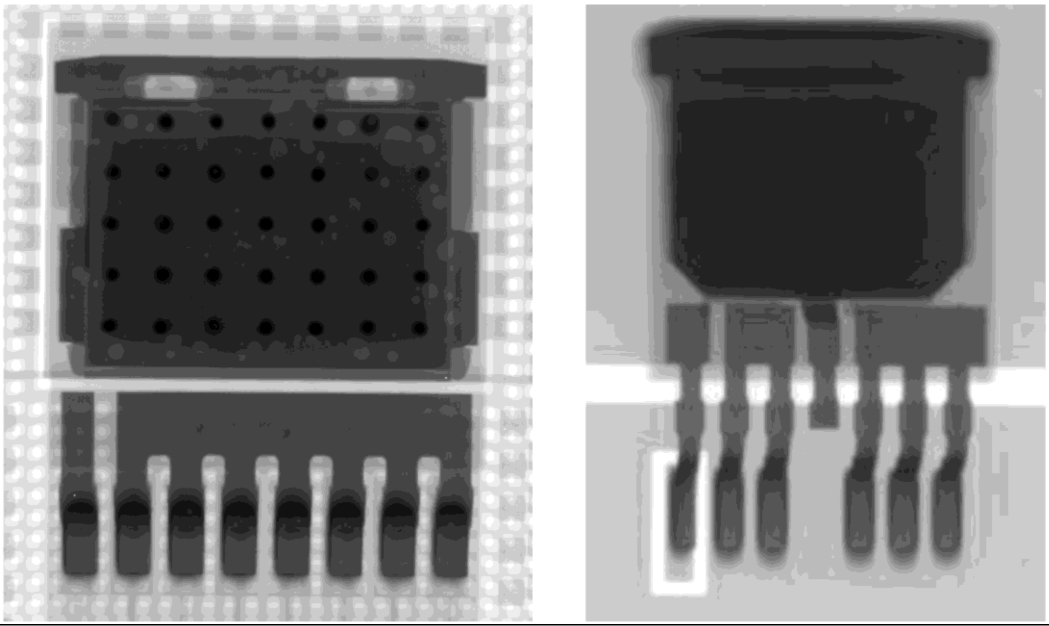

Figure 21 Typical X-ray image of a soldered HSOG-8 (left) and of a TO263-7 SMD package (right).

7 Rework

Infineon TO package solder joints are generally reworkable. The reuse of de-soldered components is not recommended. The de-soldered components should be replaced by new ones.

When reworking TO THD devices their resistance to solder shock according to JESD22-B106 must be respected [8].

A rework process of SMD packages is commonly done on special rework equipment. There are various systems available that meet the requirements for reworking SMD packages. All handling guidelines discussed in this document have to be respected. Special focus should be on the following items:

- Due to the decreased automation level given by the general rework approach, even higher care compared to standard assembly must be taken. Tools that do not damage the component mechanically have to be chosen. Mechanical forces that do not necessarily cause visible external damage can still cause internal damage that reduces the component's reliability. A proper handling system with vacuum nozzle may be the gentlest process and is therefore recommended. However, the impact of rework tools has to be assessed properly. In general, more manual handling increases the effort for documentation, training, and monitoring of the rework process(es).
- During rework, special care must be taken concerning the proper moisture level of the SMD component according to the J-STD-033. Drying the PCB and the component prior to rework might be necessary. A proper drying procedure for SMD packages is described in the international J-STD-033 standard [6]. Please also refer to the recommendations of your PCB manufacturer and take all specific needs of components, PCB, and other materials into account.
- Whatever heating system is used (hot air, infrared, hot plate, etc.), the applied temperature profile at the component must never exceed the maximum temperature according to the J-STD-020 standard. Depending on the specific heating profile used during rework, components adjacent to the mounting location might also experience a further "reflow run" in terms of the J-STD-020 standard [5]. Internal investigations have shown that the temperature profile must be recorded.

If a device is suspected to be defective and a failure analysis is planned, Infineon usually expects customers to desolder the component prior to return to Infineon. The component shall be returned in a proper condition according to the original package outlines.

In some special cases such as solder joint inspection Infineon may request that the PCB or part of the PCB with the component still attached should be sent to Infineon.

Note: *Before returning a device for failure analysis at Infineon, please clarify the return condition of the suspected component (i.e. onboard or desoldered) with the Infineon Application Engineer or Customer Quality Manager who supports your company.*

For further information about component rework on PCB, please refer to the *General Recommendations for Board Assembly of Infineon Packages* document that is available on the Infineon web page [1]. Please also feel free to contact your local sales, application, or quality engineer.

8 References

[1] Infineon: Packages. https://www.infineon.com/packages.

[2] Infineon: Thermal performance of surface mount semiconductor packages. AN_201708_PL52_023, V 1.0, 2018-03-07. https://www.infineon.com.

[3] International Electrotechnical Commission: IEC 60068-2-58. Environmental testing – Part 2-58: Tests – Test Td: Test methods for solderability, resistance to dissolution of metallization and to soldering heat of surface mounting devices (SMD).

[4] Electronic Components Industry Association, Assembly and Joining Processes and JEDEC Solid State Technology Association Committee: EIA/IPC/JEDEC J-STD-002. Solderability Tests for Component Leads, Terminations, Lugs, Terminals and Wires.

[5] JEDEC Solid State Technology Association: IPC/JEDEC J-STD-020. Moisture/Reflow Sensitivity Classification for Nonhermetic Surface Mount Devices.

[6] JEDEC Solid State Technology Association: IPC/JEDEC J-STD-033. Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices.

[7] Association Connecting Electronics Industries: IPC-A-610. Acceptability of Electronic Assemblies.

[8] JEDEC Solid State Technology Association: JESD22-B106. Resistance to Solder Shock for Through-Hole Mounted Devices.

[9] International Electrotechnical Commission: IEC 60068-2-20. Environmental testing – Part 2-20: Tests – Test T: Test methods for solderability and resistance to soldering heat of devices with leads

Revision history

Page or reference	Major changes since the last revision
Entire document	Transfer to new template including update of document structure.

Trademarks

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

Edition 2021-10-11

Published by

Infineon Technologies AG

81726 Munich, Germany

© 2021 Infineon Technologies AG.
All Rights Reserved.

Do you have a question about this document?

Email: erratum@infineon.com

Document reference

IMPORTANT NOTICE

The information contained in this application note is given as a hint for the implementation of the product only and shall in no event be regarded as a description or warranty of a certain functionality, condition or quality of the product. Before implementation of the product, the recipient of this application note must verify any function and other technical information given herein in the real application. Infineon Technologies hereby disclaims any and all warranties and liabilities of any kind (including without limitation warranties of non-infringement of intellectual property rights of any third party) with respect to any and all information given in this application note.

The data contained in this document is exclusively intended for technically trained staff. It is the responsibility of customer's technical departments to evaluate the suitability of the product for the intended application and the completeness of the product information given in this document with respect to such application.

For further information on the product, technology, delivery terms and conditions and prices please contact your nearest Infineon Technologies office (**www.infineon.com**).

WARNINGS

Due to technical requirements products may contain dangerous substances. For information on the types in question please contact your nearest Infineon Technologies office.

Except as otherwise explicitly approved by Infineon Technologies in a written document signed by authorized representatives of Infineon Technologies, Infineon Technologies' products may not be used in any applications where a failure of the product or any consequences of the use thereof can reasonably be expected to result in personal injury.