Recommendations for Board Assembly of Infineon CanPAK^{TM*}

IMM PSD LV
Kilger Thomas

Published by Infineon Technologies AG
http://www.infineon.com

CanPAKTM products use DirectFET® technology licensed from International Rectifier Corporation DirectFET^{TM} is a registered trademark of International Rectifier Corporation

Edition 28.02.2011

Published by
Infineon Technologies AG
81726 München, Germany

Legal Disclaimer
The information given in this document shall in no event be regarded as a guarantee of conditions or characteristics. With respect to any examples or hints given herein, any typical values stated herein and/or any information regarding the application of the device, Infineon Technologies hereby disclaims any and all warranties and liabilities of any kind, including without limitation, warranties of non-infringement of intellectual property rights of any third party.

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

Warnings
Due to technical requirements, components may contain dangerous substances. For information on the types in question, please contact the nearest Infineon Technologies Office. Infineon Technologies components may be used in life-support devices or systems only with the express written approval of Infineon Technologies, if a failure of such components can reasonably be expected to cause the failure of that life-support device or system or to affect the safety or effectiveness of that device or system. Life support devices or systems are intended to be implanted in the human body or to support and/or maintain and sustain and/or protect human life. If they fail, it is reasonable to assume that the health of the user or other persons may be endangered.

1 Package Description

The CanPAK™ is a near chipscale metal-shielded package with an exposed die surface on the bottom (see **Figure 1**). The metal can forms the drain connection of the package and has two connections that are soldered to the PCB during the board-mounting process. The can acts as a heat spreader and also protects the die from mechanical damage. The drain contact of the die is connected to the interior of the can with a conductive adhesive. The die surface is protected by a passivation layer to avoid mechanical damage; this layer also defines the wettable surfaces of source and gate pads. The pads are pre-soldered to aid the board mounting soldering process. There are no unnecessary packaging elements to contribute to higher thermal and electrical inductance, and resistance. Therefore the CanPAK™'s power capabilities exceed those of other comparably sized packages.

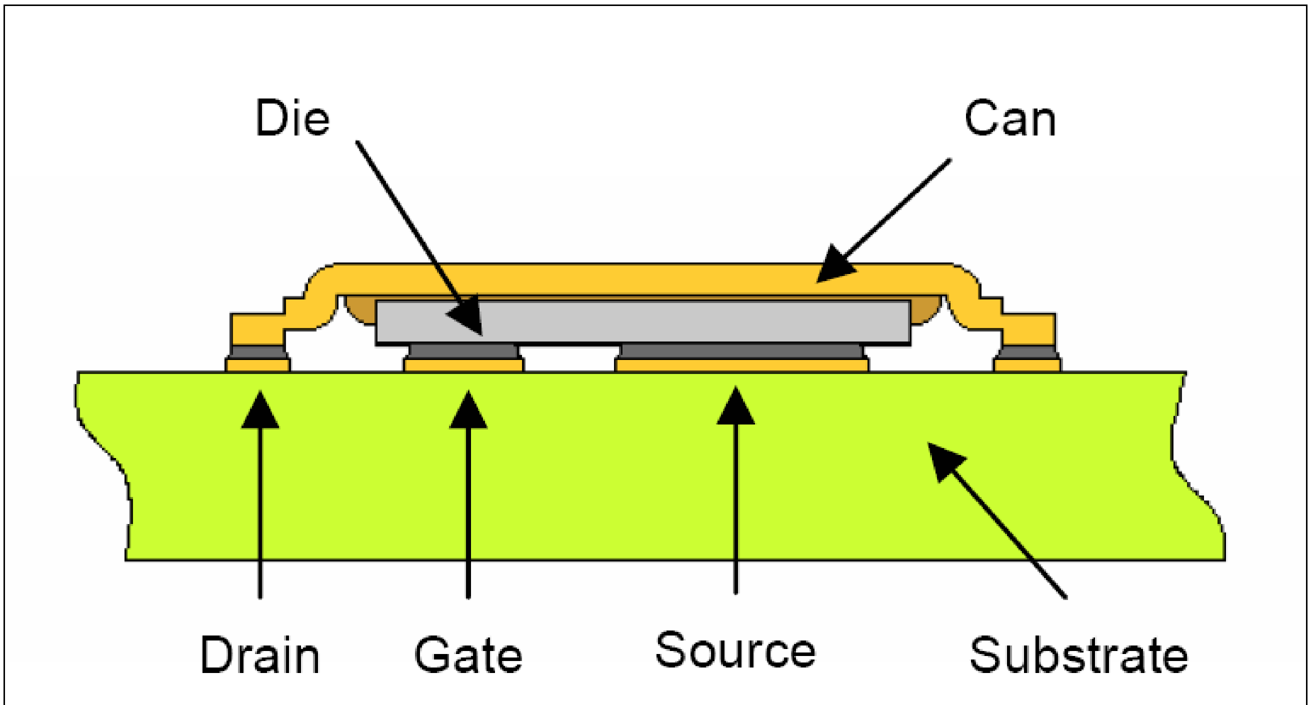


Figure 1: Schematic cross section through CanPAK™ soldered on PCB

Table 1 shows a list of Infineon's CanPAK™ and the corresponding name according JEDEC specification.

Table 1: CanPAK™ vs. MG-WDSON (name acc. JEDEC specification: MG-WDSON Metal Green Very Very thin Dual Small Outline Non-leaded package)

CanPAK™	SJ	SQ	ST	MN	MP	MX	MZ
MG-WDSON	-2-2	-2-3	-2-4	-2-5	-2-6	-2-8	-2-9

2 Package Handling

2.1 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.

2.1.1 ESD-Protective Measures in the Workplace

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

2.1.2 Equipment for Personnel

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

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

2.1.3 Production Installations and Processing Tools

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

2.2 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 strorage 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 ⇨ Packing Information (Internet)

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)

*) please differ packing from package. Packings are covering devices. Packages are different devices in terms of the external appearance (not dealing with electrical parameters).

2.3 Moisture-sensitive Components (MSL Classification)

For moisture-sensitive packages, it is necessary to control the moisture content of the components. Penetration of moisture into the package molding compound is generally caused by exposure to ambient air. In many cases, moisture absorption leads to moisture concentrations in the component that are high enough to damage the package during the reflow process. Thus it is necessary to dry moisture-sensitive components, seal them in a moisture-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 2). 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.

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

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

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.

2.4 Storage and Transportation Conditions

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

These standards should be taken into account:

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

Table 3: General storage conditions – overview

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)

(MBB = Moisture Barrier Bag)

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

Internet Links to Standards Institutes:

American National Standards Institutes (ANSI)

Electronics Industries Alliance (EIA)

Association Connecting Electronics Industries (IPC)

2.5 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.
CanPAK™ 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)

2.6 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.
CanPAK™ components have two different finishes. The metal can is plated copper. The source and gate pads have small solder depots of Pb-free composition (SnAgCu). CanPAK™ components are compatible with Pb-containing and Pb-free soldering.

3 Printed Circuit Board

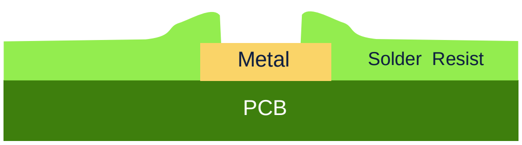
3.1 Routing

The PCB design and construction are key factors for achieving highly reliable solder joints. For example, CanPAK™ 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.

3.2 PCB Pad Design

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

- „Solder-Mask Defined“ (SMD) pad: 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.



- „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).

The metal can of CanPAK™ packages (drain) has to be soldered to the PCB on both sides of the package to conduct a large amount of heat into the PCB to achieve high thermal performance. In high-current applications or those having high thermal dissipation, source pads also require the largest possible contact area to the PCB.
SMD pads are the preferred solution to get the largest possible contact areas for drain and source. To increase the conductivity, the copper areas should be maximized (depending on application, PCB manufacturer capability, etc.).

The following figures show the recommended PCB pad designs including appropriate dimensions for CanPAK™ SJ, CanPAK™ SQ, CanPAK™ ST, CanPAK™ MN, CanPAK™ MP, CanPAK™ MX, and CanPAK™ MZ, respectively.

Please note that the recommendations can only give dimensions for the solder-mask openings. Generally the copper dimensions depend on the capability of the board manufacturer. For high current applications, the copper dimensions for drain and source pads should be as big as possible to enlarge the conductor cross-sections.

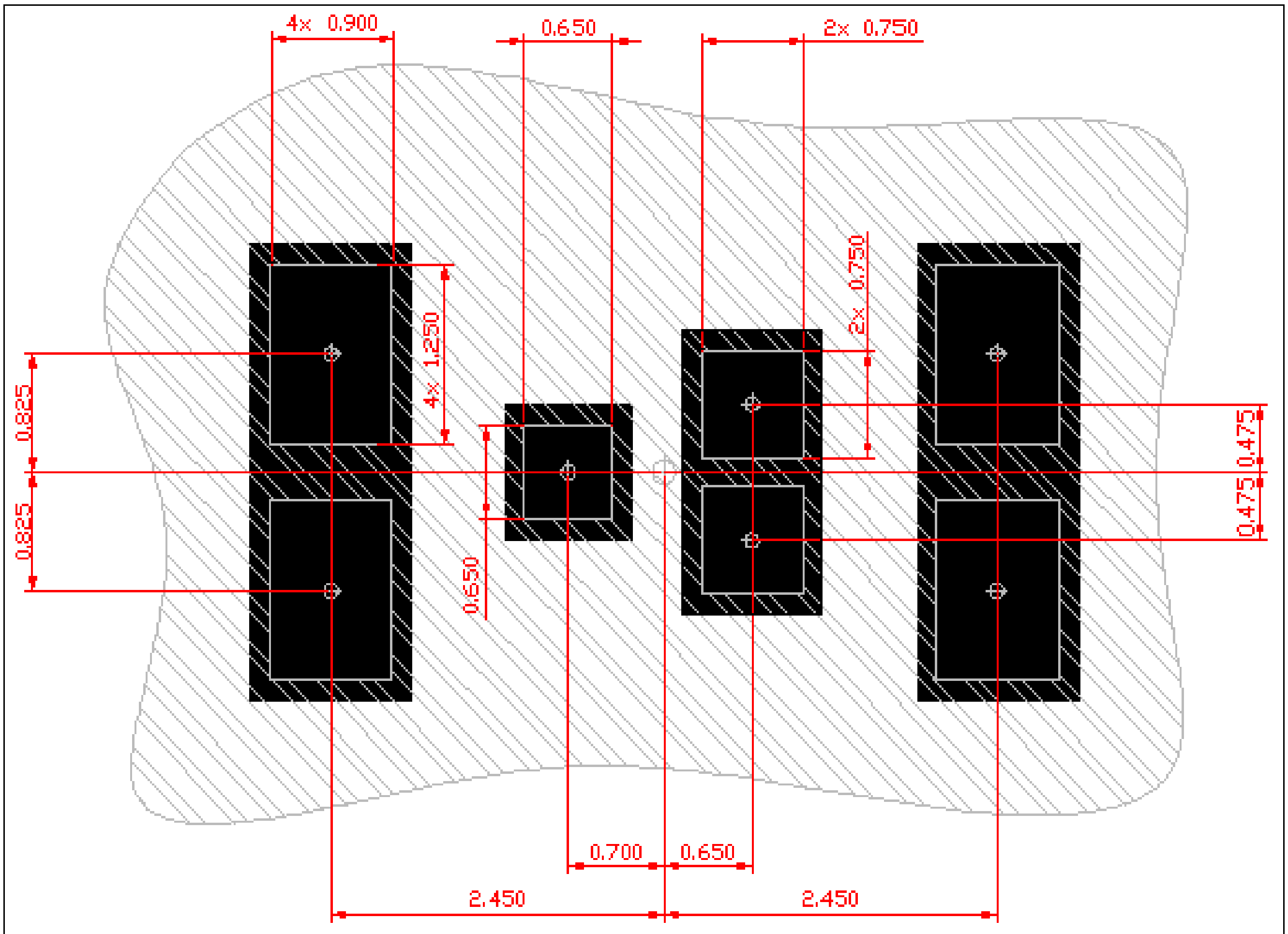


Figure 2: Recommended PCB pad design for CanPAK™ SJ

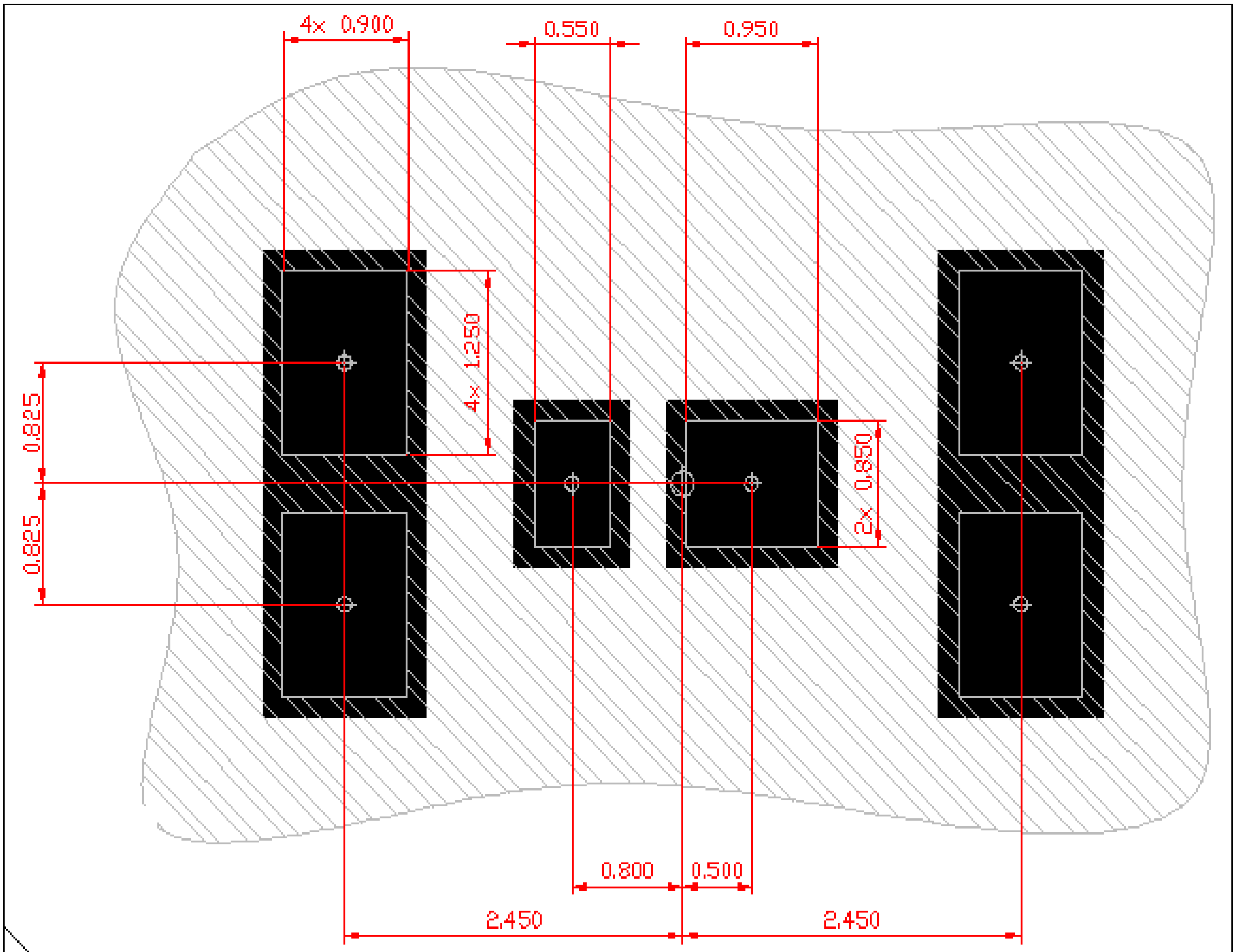


Figure 3: Recommended PCB pad design for CanPAK™ SQ

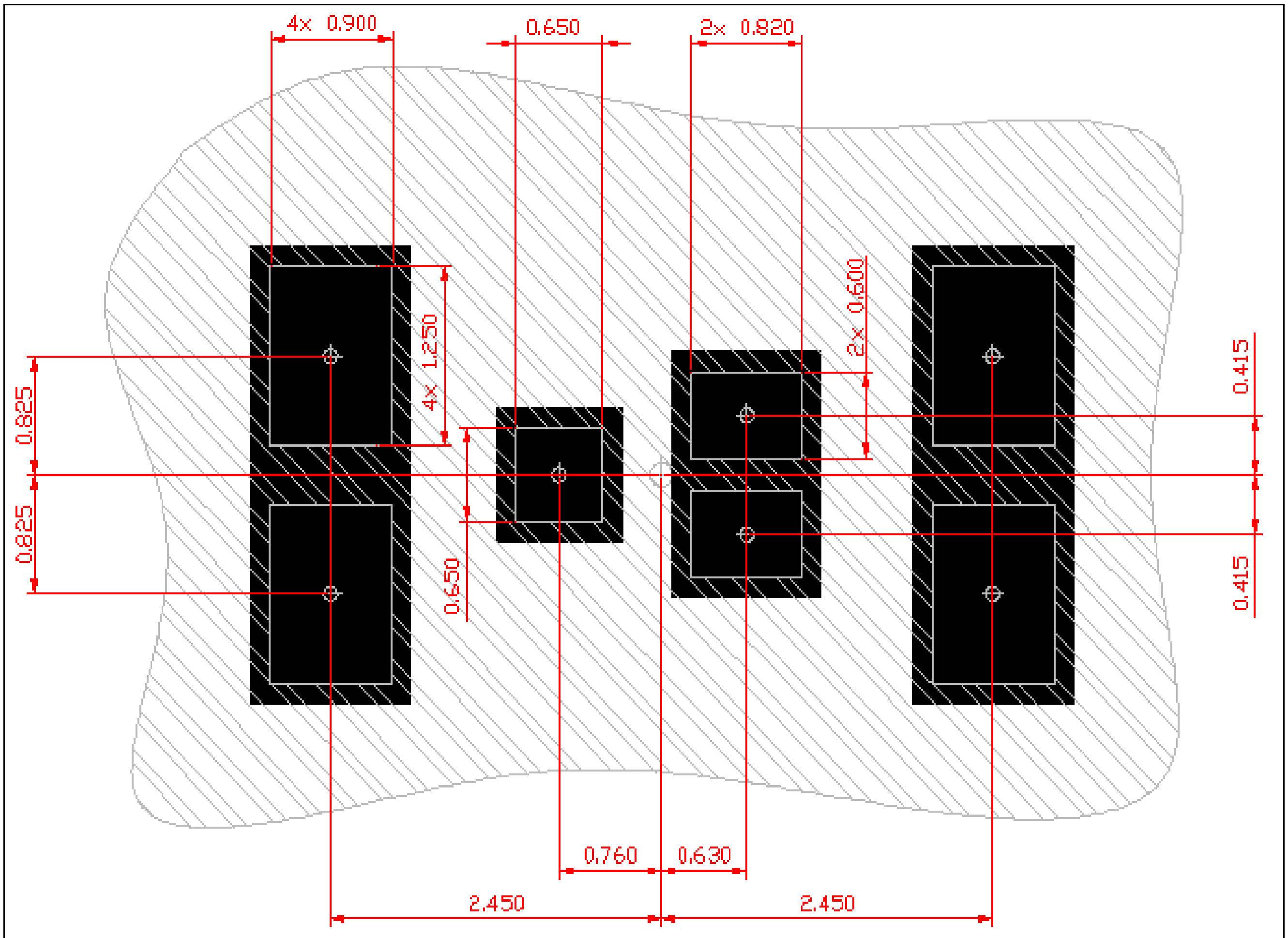


Figure 4: Recommended PCB pad design for CanPAK™ ST

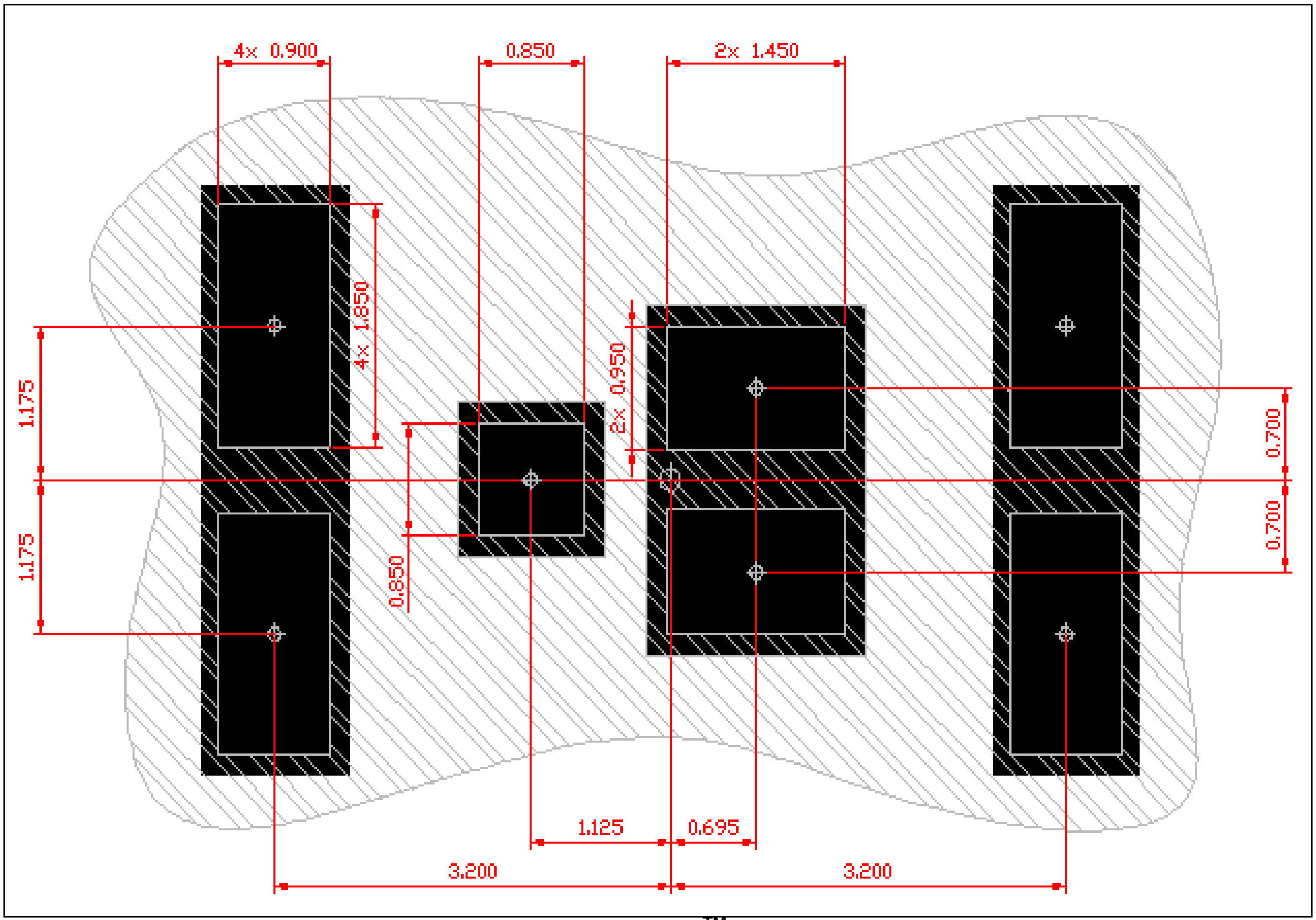


Figure 5: Recommended PCB pad design for CanPAK™ MN

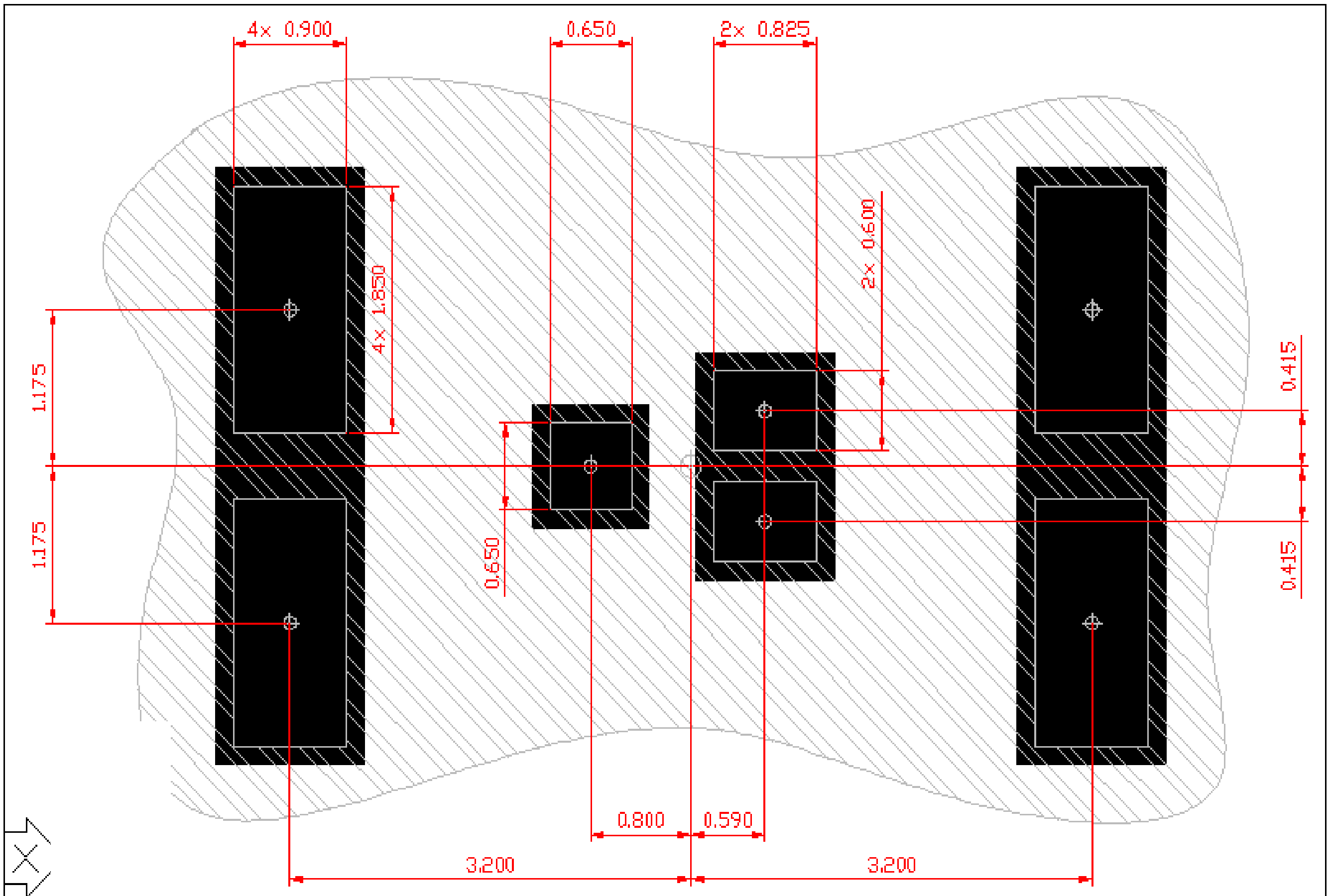


Figure 6: Recommended PCB pad design for CanPAK™ MP

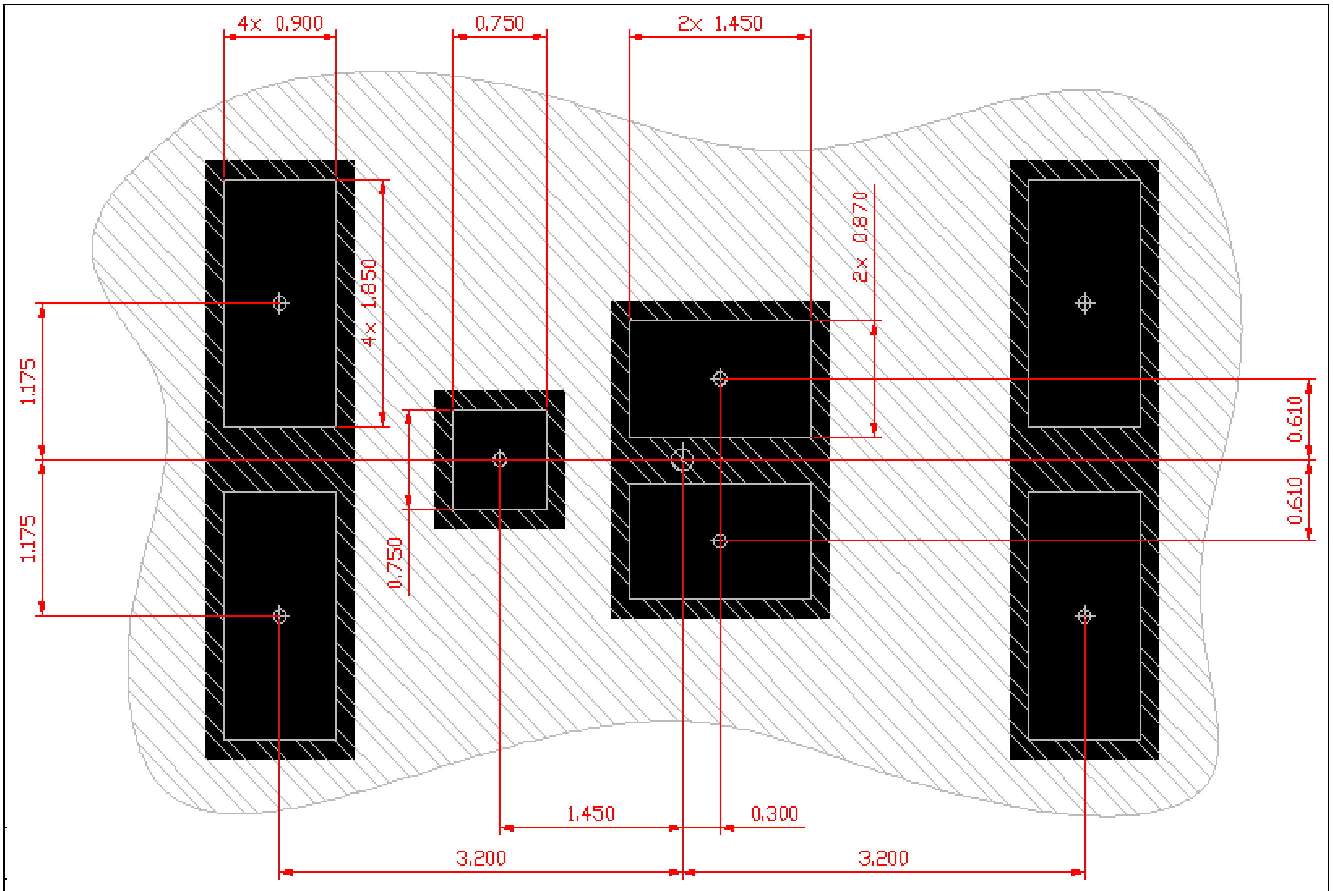


Figure 7: Recommended PCB pad design for CanPAK™ MX

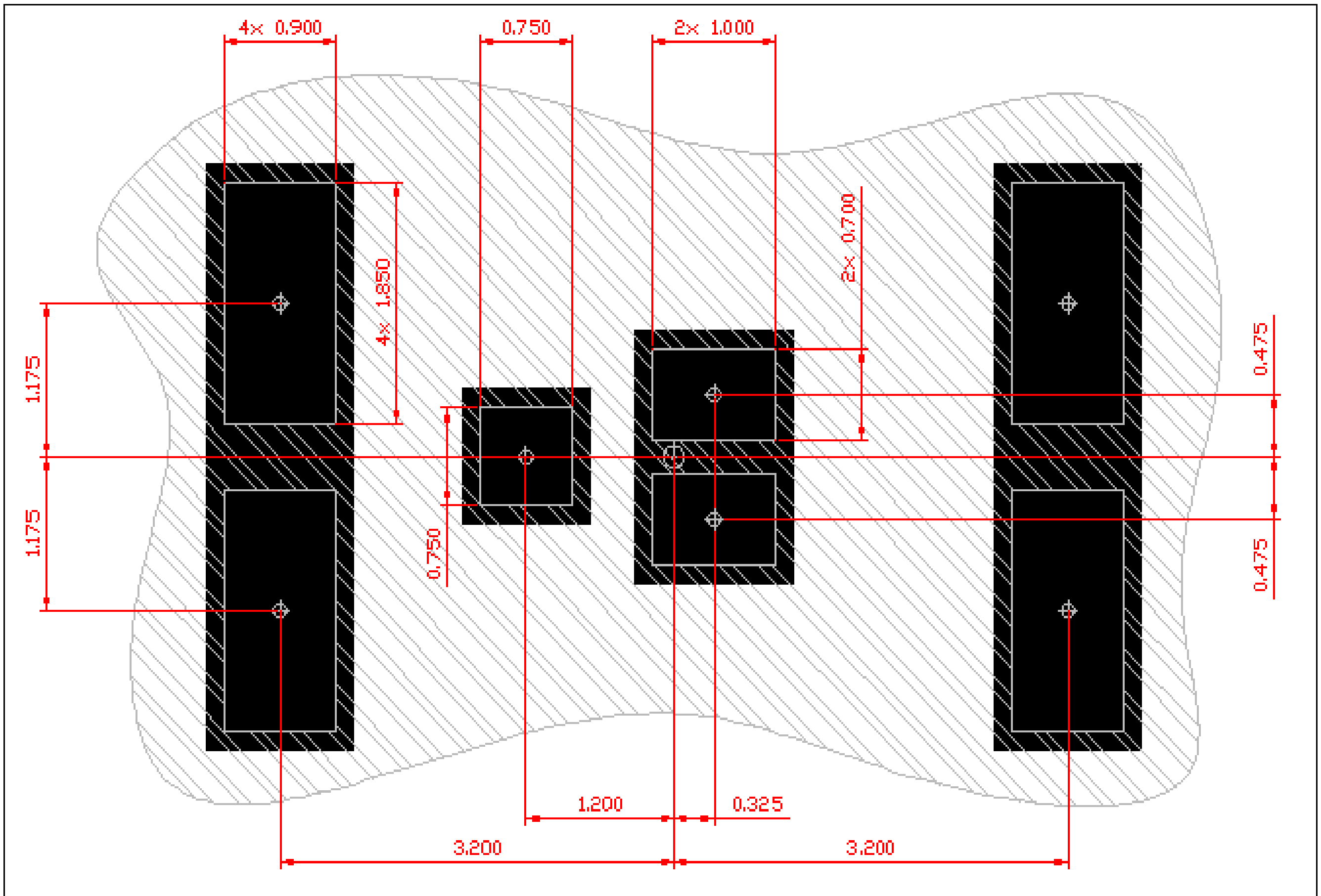


Figure 8: Recommended PCB pad design for CanPAK™ MZ

To connect drain and source pads thermally and electrically directly to inner and/or bottom copper planes of the board, plated through-hole vias are used. They help to distribute the heat into the board area, which spreads from the chip directly through source contacts or by the metal-can, in case of drain. Large copper areas surrounding the solder-mask openings of source and drain pads provide enough space for vias. Locating vias too near to or in the area of open solder mask leads to solder wicking and could finally result in soldering problems and/or reduced reliability.
Thermal and electrical analysis and/or testing are recommended to determine the minimum number of vias needed for a specific application.

3.3 Pad Surfaces

The solder pads have to be easy for the the solder paste to wet. In general, all finishes are well-proven for Surface Mount Technology Assembly (SMTA). Using a Hot Air Solder Leveling (HASL) finish (Pb-free or Pb-containing HASL), a certain unevenness has to be taken into account. Other platings are completely "flat" (e.g. Cu-

OrganicSolderabilityPreservative, electroless Sn or NiAu) and therefore are preferred when fine-pitch components are used on the PCB (please refer to Table 4).
From a package point of view, it is difficult to recommend a certain PCB pad finish that will always meet all requirements. The choice of finish also depends strongly on board design, pad geometry, all components on the board, and process conditions, and must meet the specific needs of the customer.
Infineon's internal tests have shown that Cu-OSP and NiAu are quite effective platings. Due to the higher cost of NiAu, Cu-OSP is recommended for mass production.

Table 4: Typical PCB pad finishes

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

4 PCB Assembly

4.1 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 design of the package has the die surface deliberately recessed in relation to the plane of the drain contacts. This avoids squeezing the solder because the metal can will always settle down to the board pad surface during soldering (side-wall wetting forces the component down). Furthermore, the pre-applied solder on gate and source pads allows a wide range of solder paste variation. Nevertheless, 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 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 figures show the given aperture sizes for CanPAK™ SJ, CanPAK™ SQ, CanPAK™ ST, CanPAK™ MN, CanPAK™ MP, CanPAK™ MX, and CanPAK™ MZ, respectively.

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.

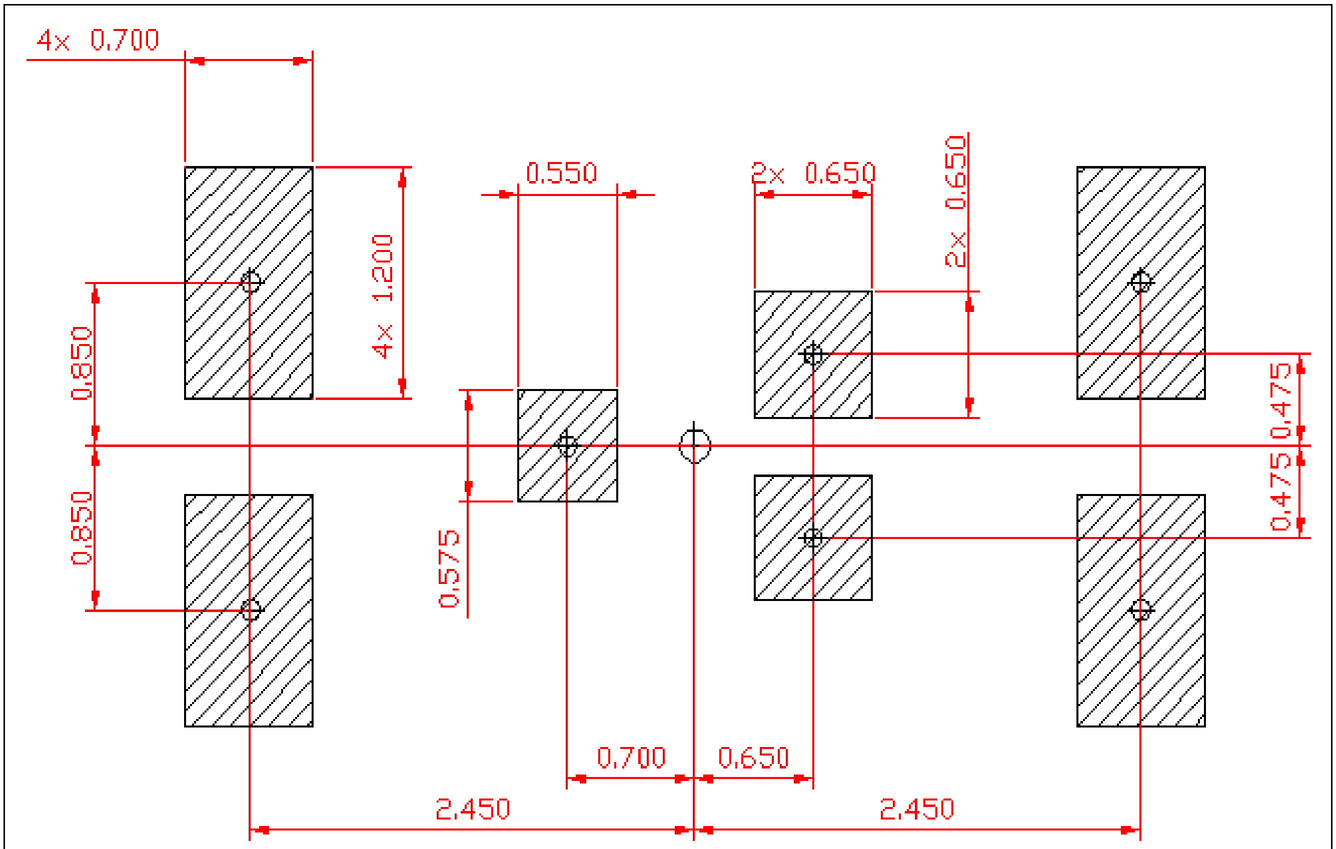


Figure 9: Recommended dimensions of stencil apertures for CanPAK™ SJ and a stencil thickness of 150 µm

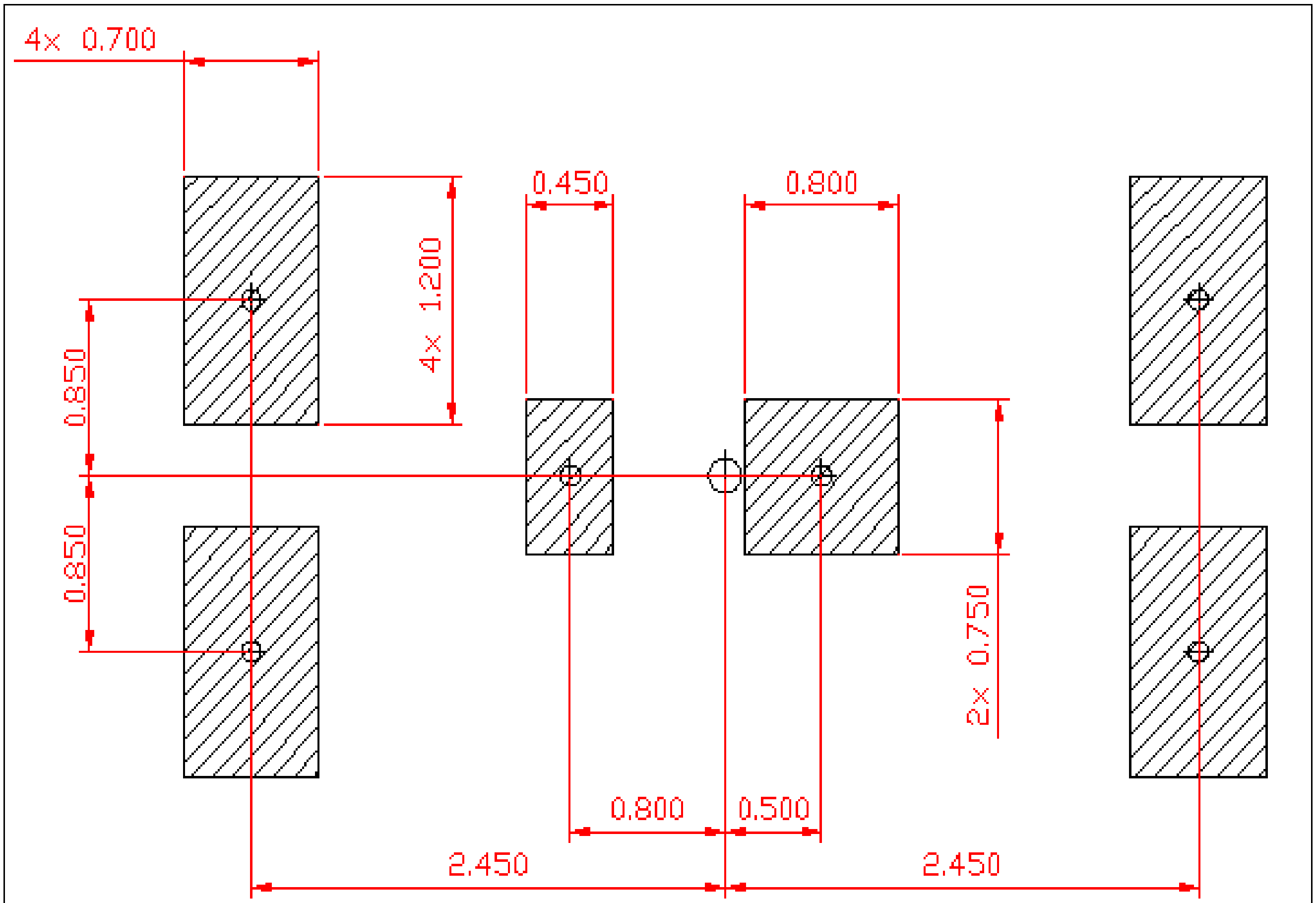


Figure 10: Recommended dimensions of stencil apertures for CanPAK™ SQ and a stencil thickness of 150 µm

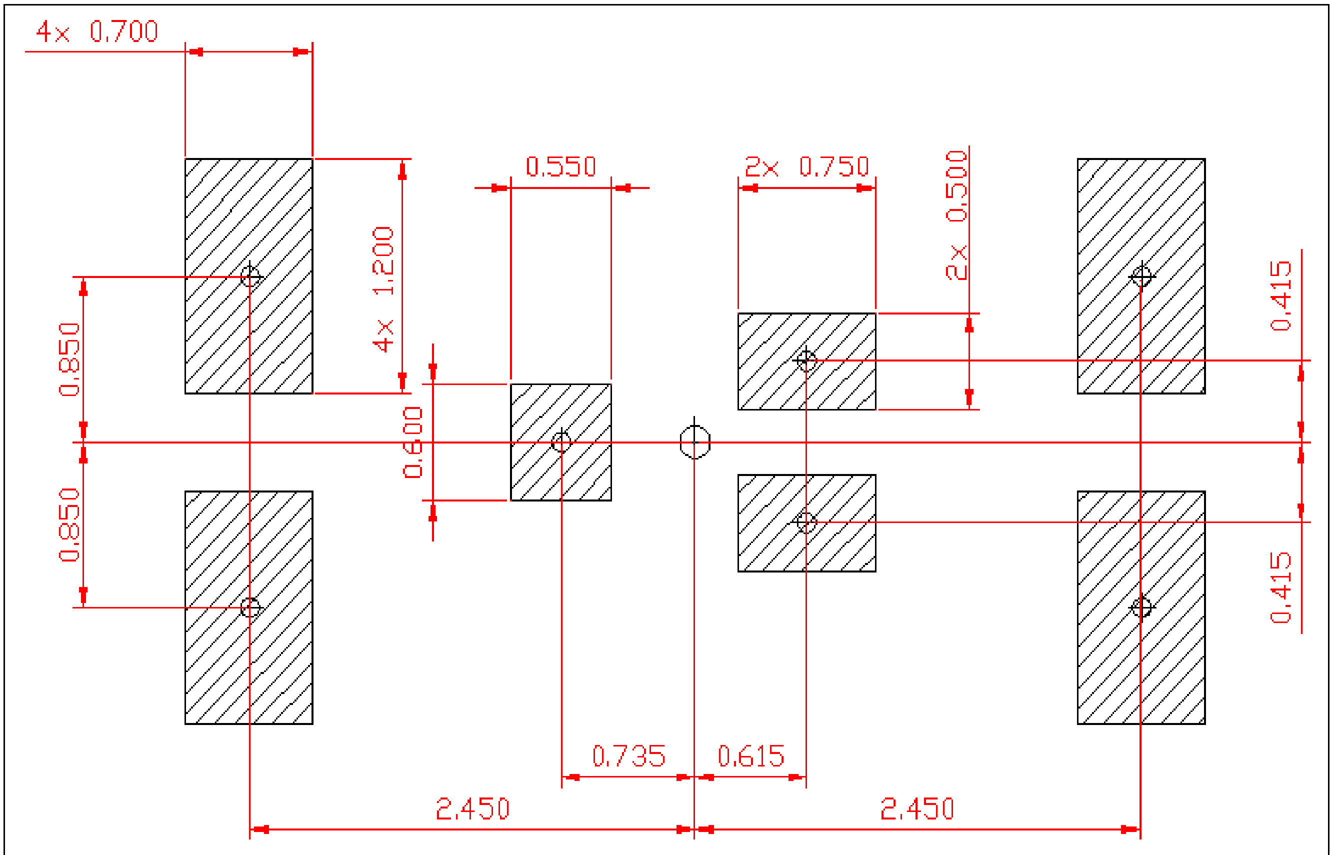


Figure 11: Recommended dimensions of stencil apertures for CanPAK™ ST and a stencil thickness of 150 µm

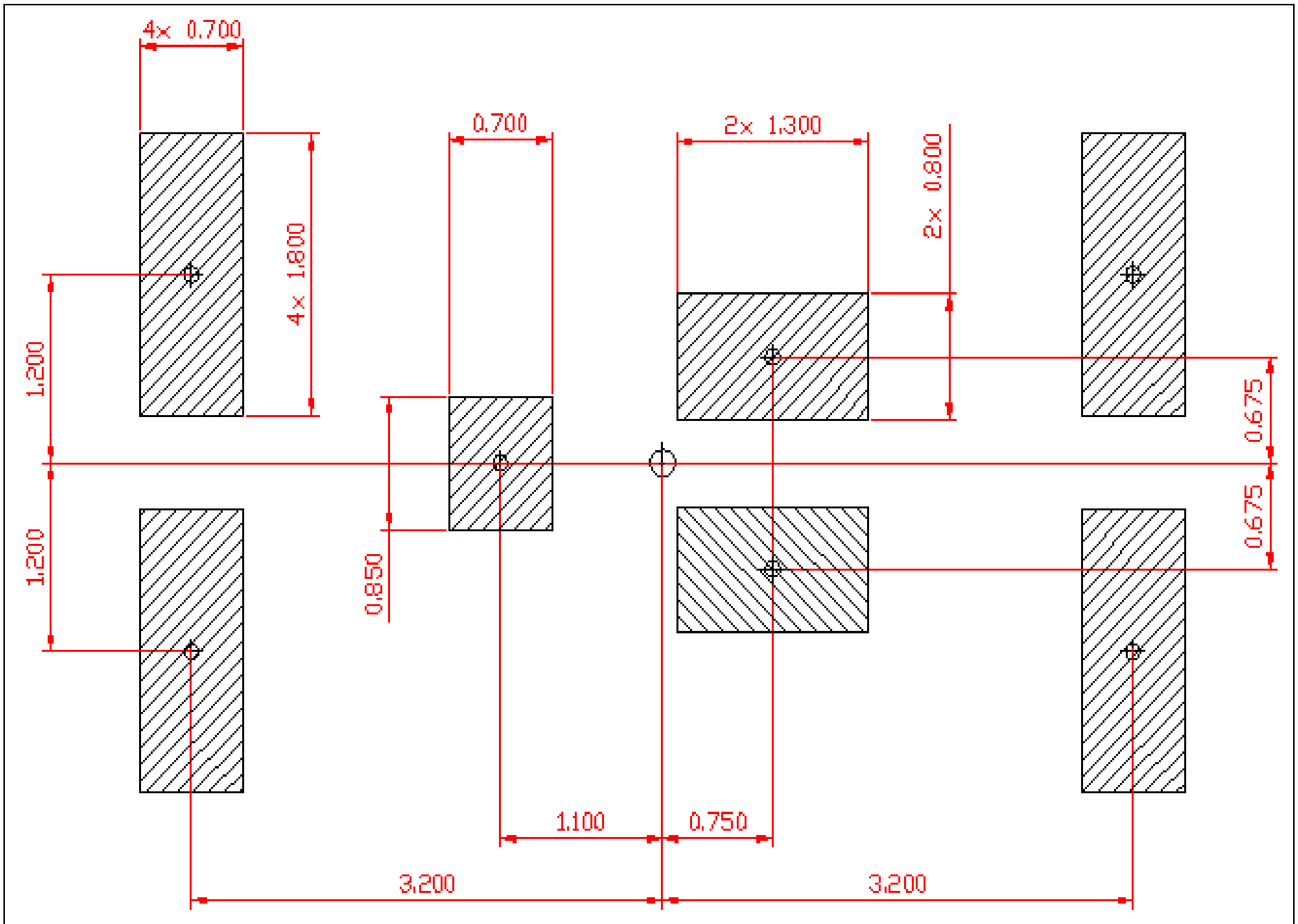


Figure 12: Recommended dimensions of stencil apertures for CanPAK™ MN and a stencil thickness of 150 µm

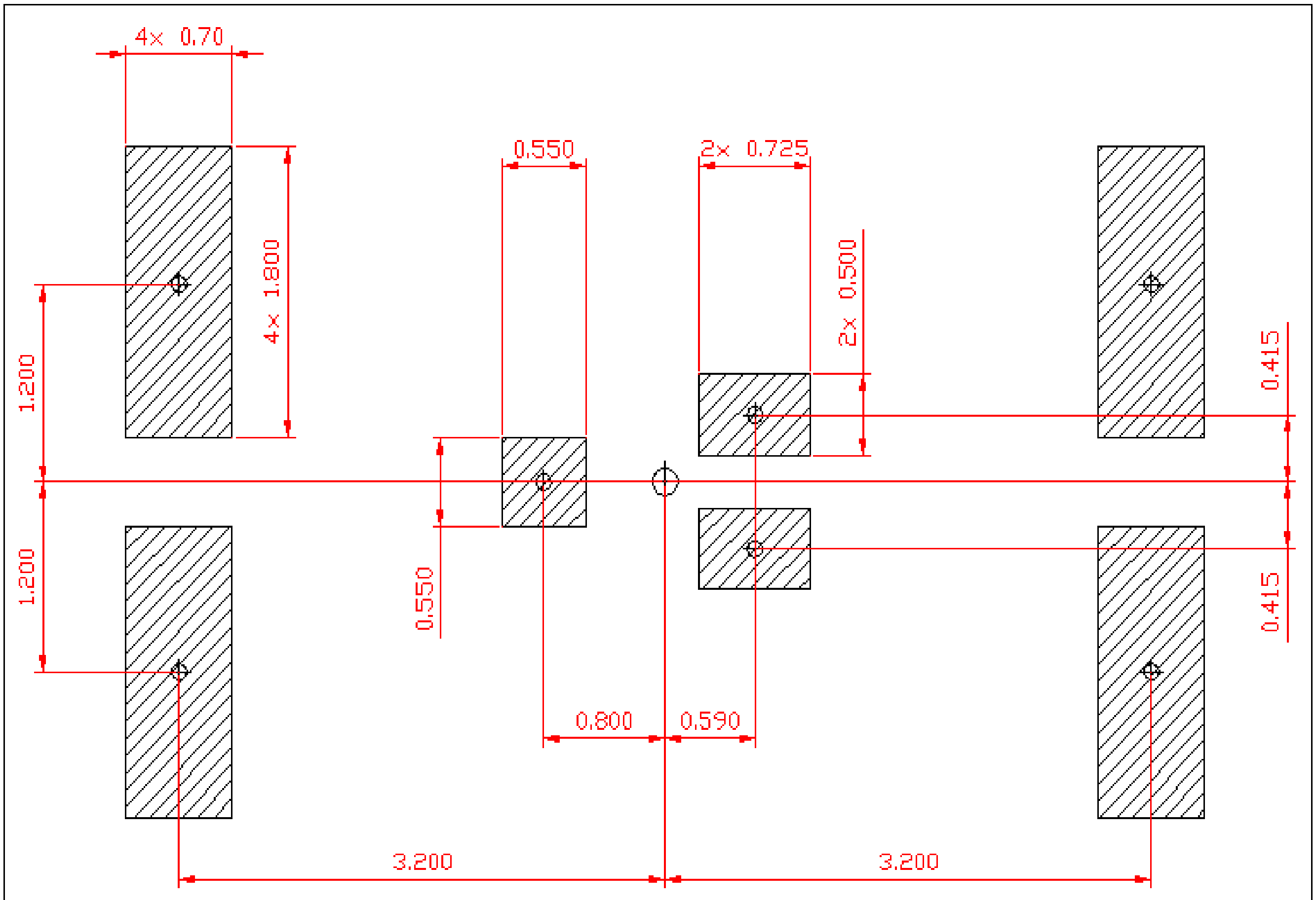


Figure 13: Recommended dimensions of stencil apertures for CanPAK™ MP and a stencil thickness of 150 µm

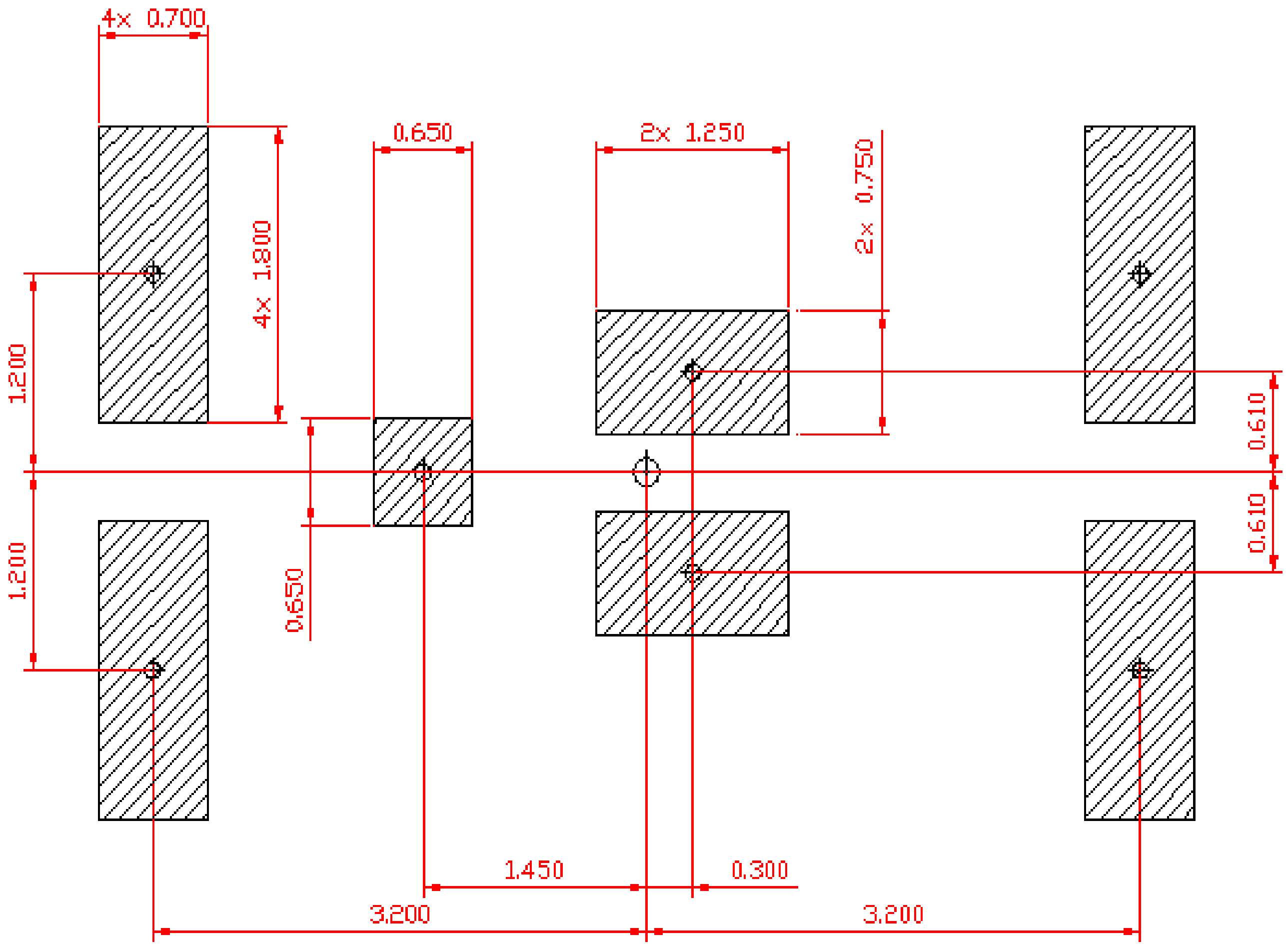


Figure 14: Recommended dimensions of stencil apertures for CanPAK™ MX and a stencil thickness of 150 µm

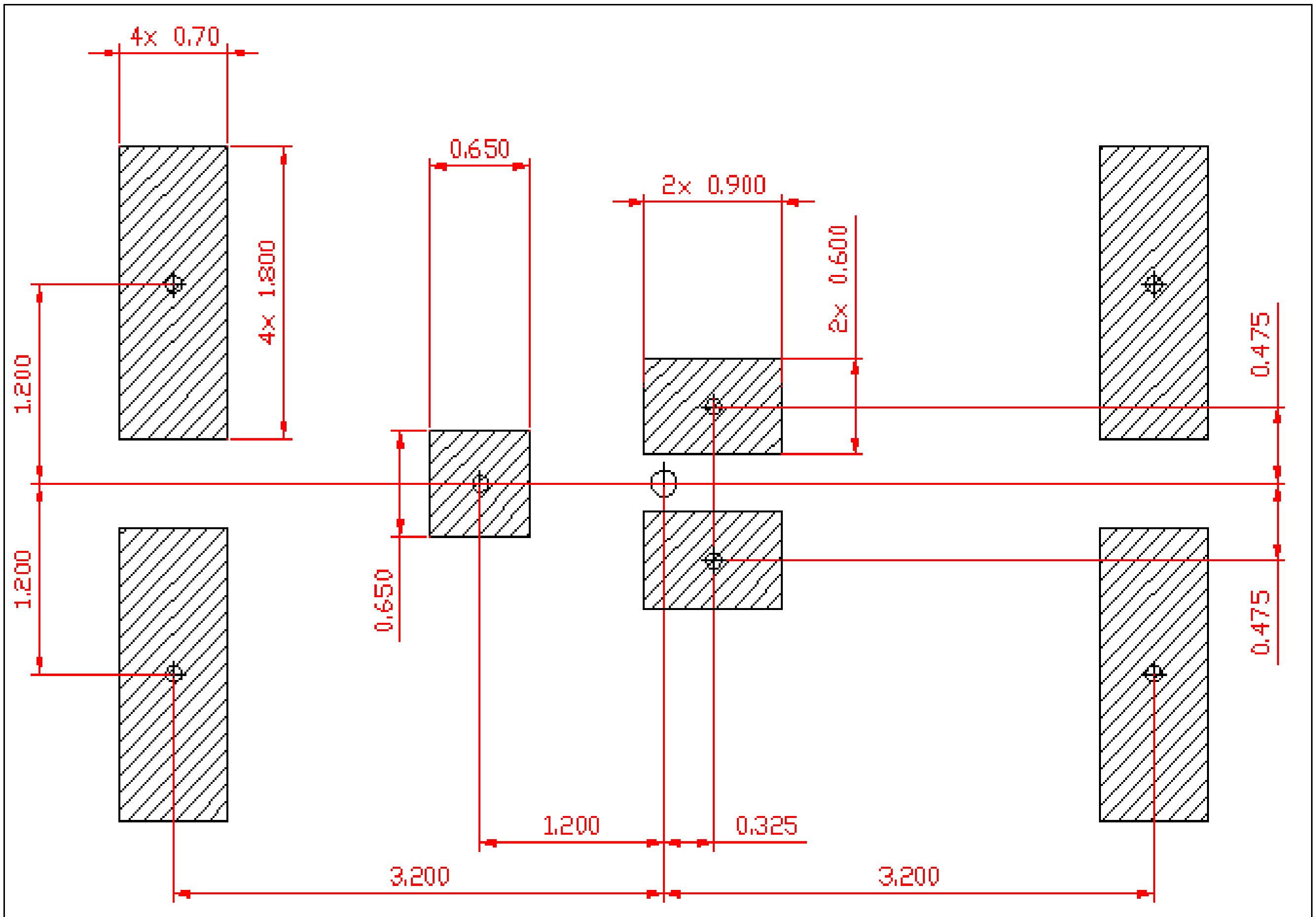


Figure 15: Recommended dimensions of stencil apertures for CanPAK™ MZ and a stencil thickness of 150 µm

4.2 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 metal alloy in Pb-containing solder pastes is typically eutectic SnPb or nearly eutectic SnPbAg. Pb-free solder pastes contain so-called SAC alloys (SnAgCu-alloys with typically 1-4% Ag and <1% Cu). A "no-clean" solder paste is mandatory for packages such as CanPAK™ where cleaning below the component is difficult and where the package contains Ag. Beside the Ag-filled conductive die adhesive and the SnAgCu-bumps of source and gate pads also the can plating and the front metallisation of the die contains silver. Together with

corrosive acting ions, humidity (→ electrolyte) and voltage, silver migration may occur. Please also refer to chapter 4.5
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.

4.3 Component Placement

For the placement of CanPAK™s the following important machine specifications have to be considered:

- Placement accuracies of +/-50 µm 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 100µm are recommended.

Further details about the placement of CanPAK™s 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 +/-50 µm are obtained with modern automatic component placement machines using vision systems. With these systems, both the PCB and the components are optically measured and the components are placed on the PCB at their programmed positions. The fiducials on the PCB are 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 maximum tolerable displacement of the components is 20% of the metal pad width on the PCB. For example, for CanPAK™ packages with minimum 0.5-mm pad width (for CanPAK™ SQ), the device-pad-to-PCB-pad misalignment has to be less than 100 µm to assure a robust mounting process (even if the self-centering effect during reflow soldering may allow much more displacement). Generally this is achievable with a wide range of placement systems.

The following factors are important:

- Especially on large boards, local fiducials close to the device can compensate for PCB tolerances.
- The lead recognition capabilites 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.

4.4 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 CanPAK™ packages. Wave soldering of CanPAK™ packages is not possible.

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 16 shows a general forced convection reflow profile for soldering CanPAK™ packages. Table 5 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, and 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.

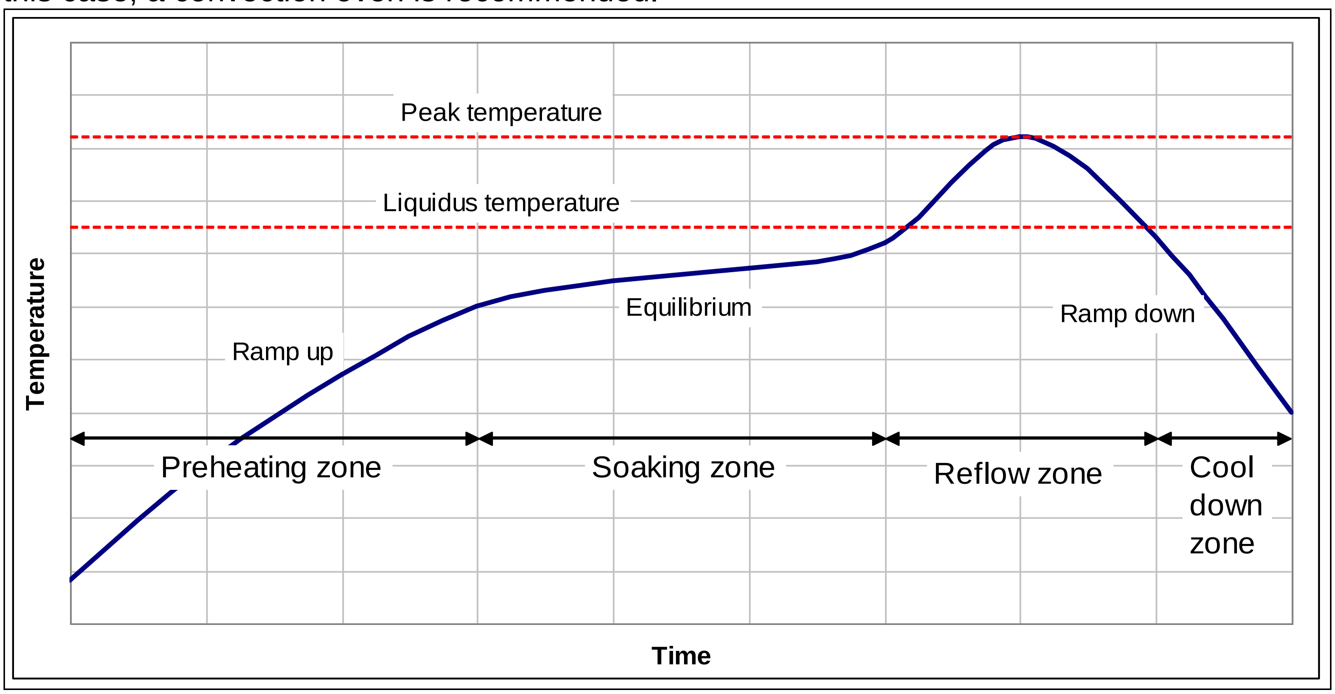


Figure 16: General forced-convection reflow solder profile

Table 5: Example of the key data of a forced-convection reflow solder profile

Parameter	Tin-lead alloy (SnPb or SnPbAg)	Pb-free alloy (SnAgCu)	Main influences coming from …
Preheating rate	2.5 K/s	2.5 K/s	Flux system (solder paste)
Soaking temperature	140-170°C	140-170°C	Flux system (solder paste)
Soaking time	80 s	80 s	Flux system (solder paste)
Peak temperature	225°C	245°C	Alloy (solder paste)

Reflow time above melting point (liquidus)	60 s	60 s	Alloy (solder paste)
Cool-down rate	2.5 K/s	2.5 K/s	

Double-Sided Assembly

CanPAK™ packages are 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 CanPAK™ packages 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.

4.5 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. For CanPAK™ such a “no-clean” solder paste is mandatory, because a reliable cleaning beneath the package in the small gap between die and PCB is very difficult.

If the customer although uses solder paste which has to be cleaned or adds other substances which contain corrosive acting ions the customer has to assure that no residues (cleaning solvent, flux, …) remain on or underneath the CanPAK™.
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 CanPAK™s.

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 CanPAK™s 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). For this the components have been soldered using no-clean solder paste onto PCBs prior to preconditioning and H3TRB testing. 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 (please refer to
Figure 17).

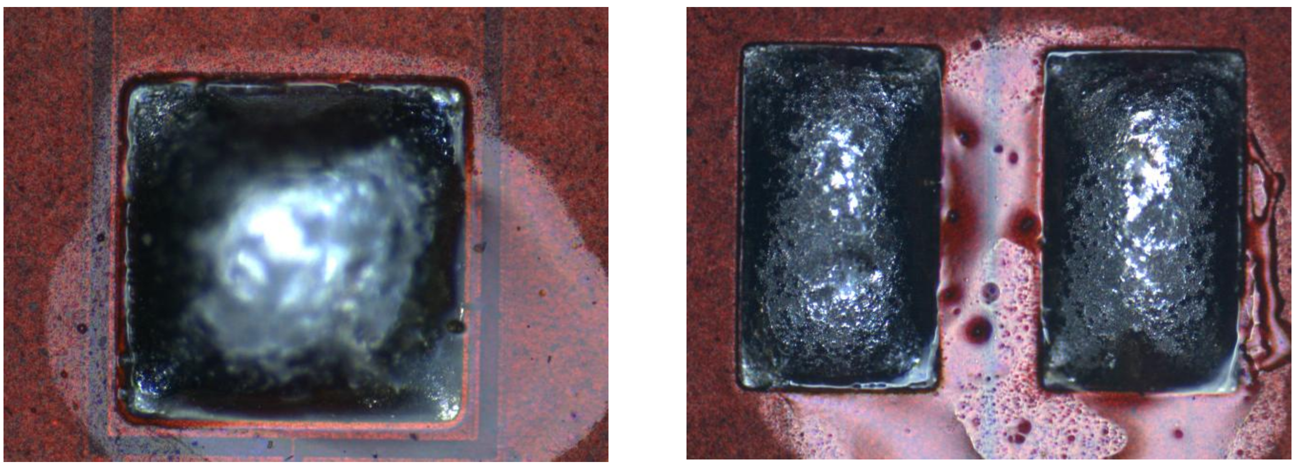


Figure 17: PCB area around the gate pad (left) and two source pads (right) after 1000 hrs H3TRB. Visible is the wide spreading flux, but no hints for silver migration or other corrosion marks can be found.

4.6 Inspection

Compared to typical SMD components that have gullwing leads, for example, the source and gate solder joints of CanPAK™ packages are formed underneath the package. A visual inspection of the solder joints with conventional Automatic Optical Inspection (AOI) systems is limited to the outer surface of the drain solder joints. **Figure 17** shows solder joints of a CanPAK™ MX and SQ.
For the acceptance of electronic assemblies inspected optically, please refer also to the IPC-A-610 standard.
CanPAK™ packages can also be inspected with endoscopes (e.g. ERSASCOPE). The optical head of the system can be moved around the package near the PCB area and can look into the small gap between PCB and package. The pictures from such an endoscopic system (**Figure 18**) are much easier to interpret than X-ray images.

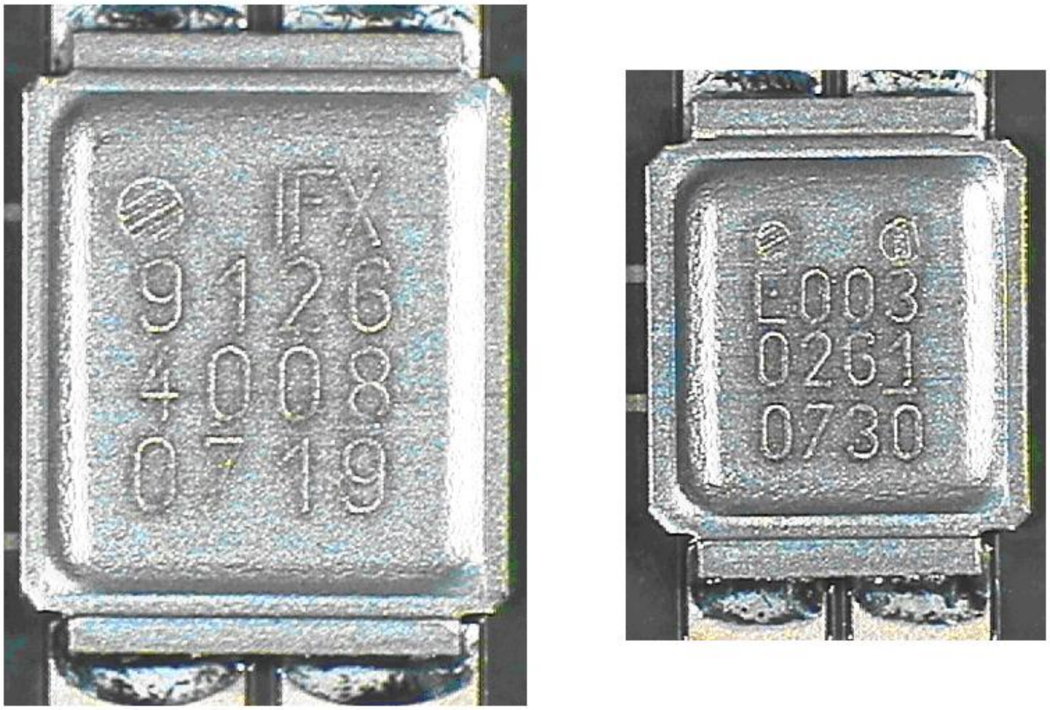


Figure 18: Typical optical photo of soldered CanPAK™ MX (left) and SQ (right). Only the drain solder joints can be seen

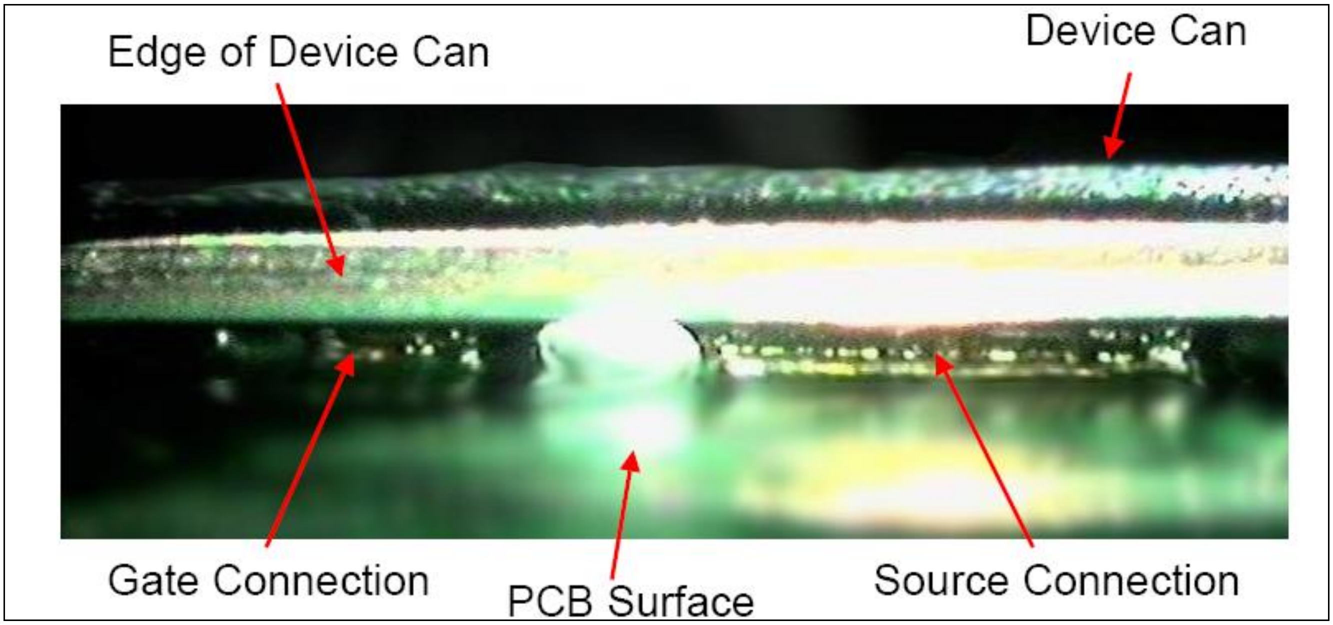


Figure 19: Optical photo of gate and source solder joints using an endosopic system

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 19** shows examples of typical X-ray photos of CanPAK™ MX and SQ packages. Solder joints of drain connections appear as light grey rectangles (metal can touches the PCB pad and results in a very thin solder joint) framed with darker areas (solder joint meniscus). Solder joints of gate and source connections appear as dark areas with rounded corners. These rounded corners are a hint for the typical “hourglass” shape of these solder joints. The silicon die and Die-Attach (DA) material are visible at the edge of the die. As a rule-of-thumb, a 25% maximum voiding rate (X-ray inspection top-down view) is a starting point.

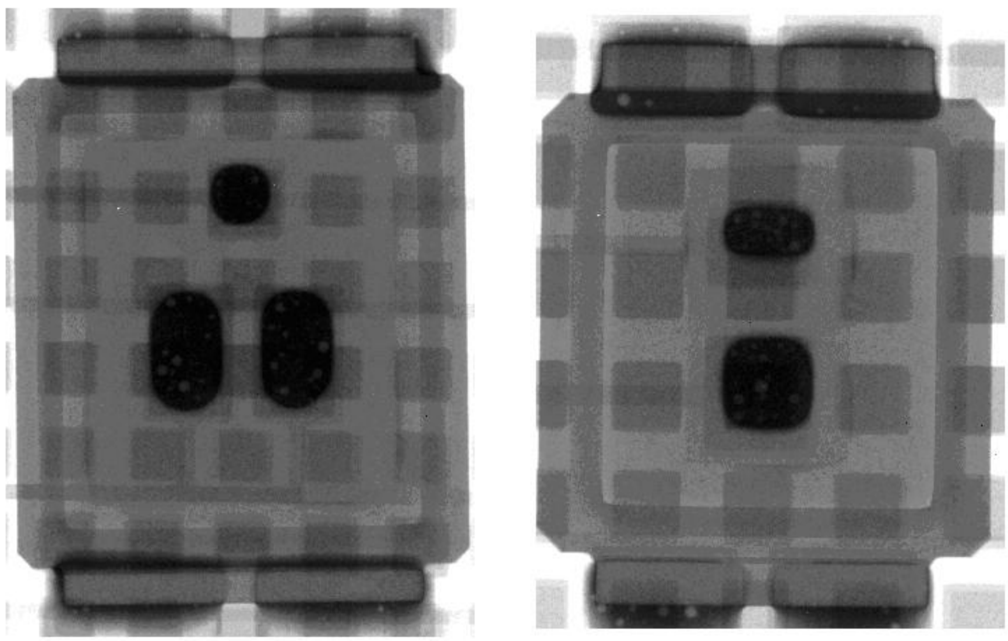


Figure 20: Typical X-ray photo of soldered CanPAK™ MX (left) and SQ (right)

Cross-sectioning of a soldered package as well as dye penetrant analysis can serve as tools for sample monitoring only, because of their destructive character. Nonetheless, these analysis methods must be used during engineering of new products at customers' production sites to get detailed information about the solder-joint quality. **Figure 20** shows typical cross-sections through solder joints of a CanPAK™ MX. The hourglass shape is typical for source and gate solder joints.

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.

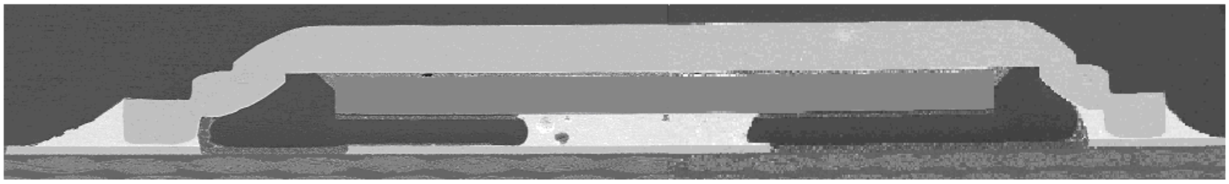


Figure 21: SEM photo of cross-section of mounted CanPAK™ MX through drain (left and right) and source connections. Gate and second source connections are located in another cross section plane.

5 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 CanPAK™s which may lead to silver migration.

5.1 Tooling

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

Heating: Hot air heat transfer to the package and PCB is strongly recommended. Temperature and air flow for heating the device should be controlled. With freely-programmable temperature profiles (e.g. by PC controller) it is possible to adapt the profiles to different package sizes and masses. PCB preheating from the underside is recommended. Infrared heating can be applied, especially for preheating the PCB from the underside, but it should be only augmenting the hot air flow from the upper side. Nitrogen can be used instead of air.

Vision system: The bottom side of the package as well as the site on the PCB should be observable. A split optic should be used for precise alignment of package to PCB. Microscope magnification and resolution should be appropriate for the pitch of the device.

Moving and additional tools: The device should be relocatable on the whole PCB area. Placement accuracy is recommended to be better than +/-100 µm. The system should have the capability of removing solder residues from PCB pads (special vacuum tools).

5.2 Device Removal

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 (see chapter 4.4).
- **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.

If any of the above instructions cannot be followed, please send the PCB together with the failed part to the supplier for further investigation.

5.3 Site Redressing

After removing the defective component, the pads on the PCB have to be cleaned of solder residues. This may be done by vacuum desoldering or wick.
If the component was underfilled, the remaining underfill on the PCB also has to be removed.
Some solvents may be necessary to clean the PCB of flux residues (and underfill residues if applicable).
Don't use steel brushes because steel residues can lead to bad solder joints. In all cases, harsh mechanical treatment may damage PCB pads and conductors.
Before placing a new component on the PCB, solder paste should be applied to each PCB pad by printing (special micro stencil) or dispensing. Another method that may lead to a decreased solder stand-off compared to non-repaired components is to apply flux only by dispensing or with a brush (often so-called "sticky" flux is used for this purpose). No-clean flux and solder paste is mandatory.

5.4 Reassembly and Reflow

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