AN90036

Recommendations for Printed Circuit Board assembly of DSN1006-3 (SOT8026)

Rev. 1.0 — 9 August 2022 application note

Document information

Information	Content
Keywords	DSN1006-3, SOT8026, reflow soldering, surface mount, solder paste, stencil aperture, PCB, SMD, footprint, landing pattern, pick and place, Chip Scale Package, CSP
Abstract	This application note provides guidelines for board assembly of a DSN1006-3 (1.0 mm x 0.6 mm) chip scale package. Main focus is on recommendations for reflow soldering. As general information about footprint design and reflow soldering see application note AN10365 (Surface mount reflow soldering description). If not otherwise stated, all measurement units given in this document are metric units. This means that also the package nomenclature, i.e. the term “1006”, refers to metric units.

nexperia

1. Introduction

This application note provides guidelines for board mounting of a DSN1006-3 (SOT8026) which size is only 1.0 mm x 0.6 mm x 0.2 mm surface mount. This ultra-small package enables smart and other mobile devices to reduce the dimensions and increase the density of functionality.

Due to the very small size of the component Nexperia investigated the board assembly process intensively in order to offer board mounting recommendations.

This includes printing circuit board (PCB) mounting pads, the stencil apertures, solder paste and board assembly process parameters.

Using the recommended dimensions for pads and stencil as described in this document will help to achieve:

- optimum stand up height
- minimum tilt
- minimum rotation
- good board assembly process performance

Following all the advices in this document guarantees a low risk on unexpected fails. The results may differ depending on the machine capability, atmosphere, material used such as printed circuits boards or solder paste material.

2. DSN1006-3 (SOT8026) Package Details

The DSN1006-3 (SOT8026) is a Discrete Silicon No Lead package (DSN). It is applied with Tin Silver (SnAg) plated copper contacts (CuSnAg pillars) under the package (bottom terminations) similar to Discrete Flat No-leads (DFN) style packages. The DSN style package enables 100% utilization of the package area for active silicon, offering a significant performance per board area advantage compared to products in plastic molded packages.

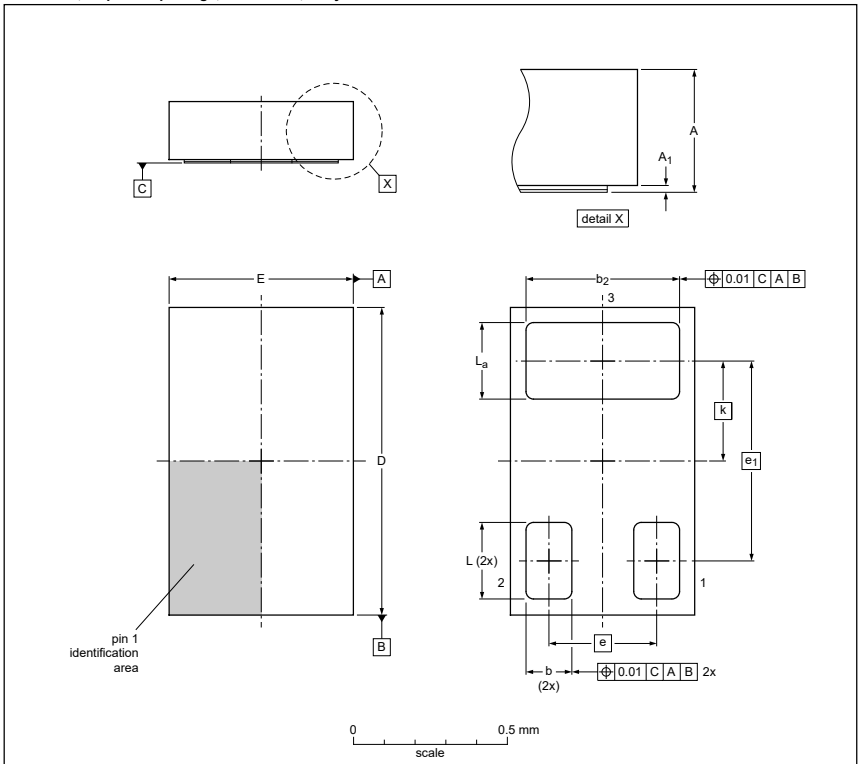
Outline dimensions of the DSN1006 have a tolerance of ±40 µm. The package comes with a low overall height of 200 µm.

Key Features:

- Ultra-small and flat package (1.0 x 0.6 x 0.2 mm^3)
- SnAg plated Cu contacts (CuSnAg pillars) for soldering on PCB

The visual appearance is shown in Fig 1 and Fig 2 shows the package dimensions.

Fig. 1. DSN1006-3 (SOT8026) Visual Appearance



Dimensions (mm are the original dimensions)

Unit(1)		A	A_1	b	b_2	D	E	e	e_1	k	L	L_a
mm	max	0.22	0.03	0.16	0.51	1.04	0.64				0.26	0.26
	nom	0.20		0.15	0.50	1.00	0.60	0.35	0.65	0.325	0.25	0.25
	min	0.18		0.14	0.49	0.96	0.56				0.24	0.24

Note
Dimension A is including coating foil thickness.

sot8026_po

Outline version	References				European projection	Issue date
	IEC	JEDEC	JEITA			
SOT8026		---				~~20-10-07~~ 20-10-09

Fig. 2. Package Dimensions DSN1006-3; 3 terminals (SOT8026)

3. PCB solder pattern

3.1. Printed Circuits Boards

Printed-Circuit Boards (PCBs) are not only used as mechanical carriers for electronic components; they also provide the electronic interconnection between these components. On these boards Cu solder pads are used to make the interconnects. The layout of the pads depend on the package design is called the footprint. By applying solder paste on the boards interconnects can be made using a solder process.

3.2. Solder pad design general options

There are two types of solder pad / solder resist designs:

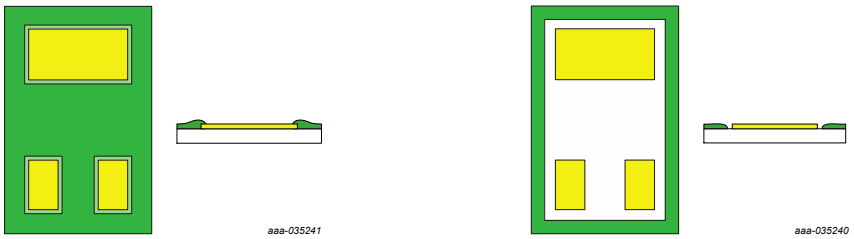
- Solder Mask Defined (SMD)
- Non-Solder Mask Defined (NSMD)

SMD is a method of designing the solder resist to partially overlap the Cu landing pads on the PCB, whereas for NSMD designs have a gap between the solder resist and the Cu landing pattern on the PCB. These two types are described in more details in the next chapter.

3.2.1. Solder Mask Defined (SMD) pad versus Non-Solder Mask Defined (NSMD) pad

If the solder mask extends onto the solder lands, the remaining solder-able area is solder mask defined (SMD). The “effective” solder pad is equal to the copper area that is not covered by solder mask. This situation is illustrated in Fig 3 left column. In case of a SMD pad, the copper will normally extend 75 µm down to 50 µm underneath the solder mask on all sides. In other words, the copper dimension is 0.1 mm to 0.15 mm larger than the solder mask dimension. These values may vary depending on the class of PCBs used. This allows for tolerances in copper etching and solder mask placement, during PCB production.

If the solder mask layer starts outside of the solder lands, and does not cover the copper, this is referred to as NSMD. The “effective” solder pad is equal to the copper area. In case of a NSMD, the solder mask should be at least 50 µm away from the solder land on all sides. In other words, the solder mask dimension is 100 µm larger than the copper dimension. These values may vary depending on the class of PCBs used. The main requirement is that the solder mask is far enough away from the copper, so that – with the given tolerances in solder mask application – it does not extend onto the copper. A NSMD footprint is shown in Fig 3 right column.



a. Solder Mask Defined solder pads (SMD)

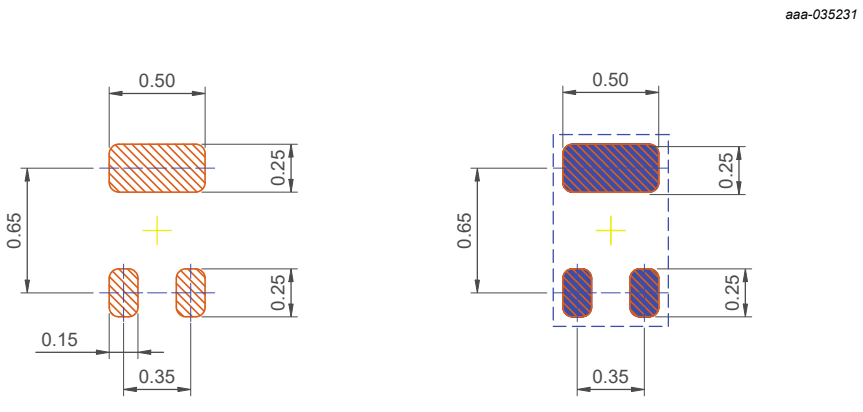
b. Non Solder Mask Defined solder pads (NSMD)

Fig. 3. SMD versus NSMD solder pads

3.3. Solder Pad Design for DSN1006-3 (SOT8026)

3.3.1. Recommended reflow solder footprint

Based on the small dimensions of the DSN1006-3 and the given tolerances for PCB manufacturing, it is recommended to use NSMD solder pads. The solder footprints with dimensions are shown in Fig 4.



reflow solder foot print

reflow solder foot print and package outline

Fig. 4. Recommended Solder Footprint for SOT8026

4. Solder stencil

4.1. Stencil recommendations

Due to small apertures and pad dimensions a high quality stencil should be applied. Nexperia uses a stainless steel stencil, manufactured by laser-cut and with plasma coating.

The recommended stencil thickness is 80 µm for the DSN1006-3 package.

For the recommended Nexperia footprint (see chapter 3.3.1, Fig 4) the optimum stencil aperture is 0.2 x 0.25 mm^2 for the small pad. Based on experience of stencil manufacturers the rounded corners with a radius of 0.03 mm are sufficient for a good solder paste release at printing. For stencil apertures refer to Fig. 5.

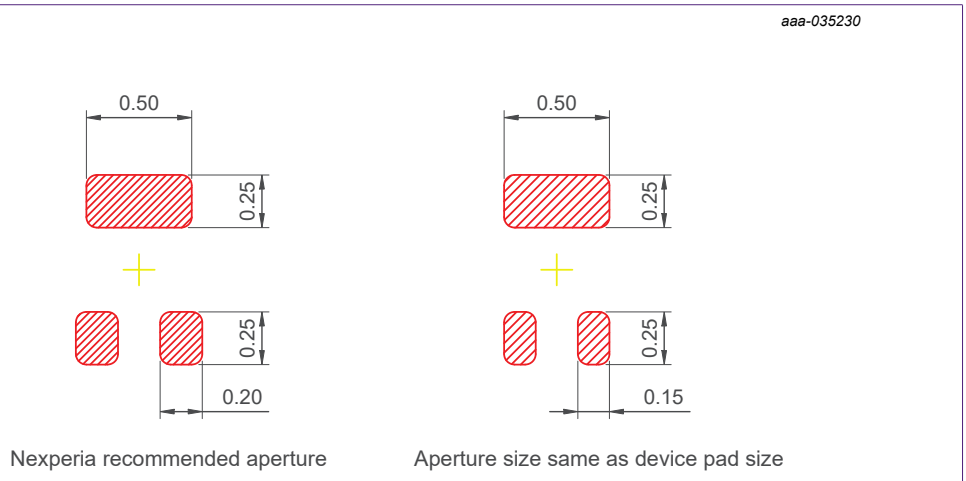
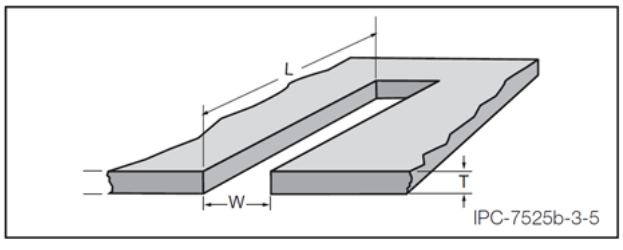


Fig. 5. Stencil Apertures for DSN1006-3

4.2. Stencil aperture design

Key design guidelines for stencil apertures are the area- and aspect ratio. The area ratio is for standard approach >0.66. Smaller values are possible with adequate process control. Of course, it depends on the manufacturing environment and other requirements of the manufacturer. The aspect ratio should be >1.5 which is less critical to fulfill. For explanation of area and aspect ratio refer to Fig 6.



Cross-Sectional View of A Stencil

$$Area\ Ratio = \frac{\text{Area of Aperture Opening}}{\text{Area of Aperture Walls}} = \frac{\text{L x W}}{\text{2 x (L + W) x T}}$$

$$Aspect\ Ratio = \frac{\text{Width of the Aperture}}{\text{Thickness of the Stencil Foil}} = \frac{\text{W}}{\text{T}} > 1{,}5$$

Fig. 6. Explanation of area ratio and aspect ratio

Table 1. Area and aspect ratio for stencil apertures as investigated

*Stencil thickness **T = 80 µm***

	Aperture size	Area ratio Target >0.62	Aspect ratio Target >1.5
Nexperia recommended footprint	200 x 250 µm^2	0.69	2.5
Footprint equals device pad size	150 x 250 µm^2	0.59	1.9

Table 1 shows the values for aspect- and area ratio of the considered stencil apertures for a stencil thickness of 80 µm. It results in an acceptable area ratio of 0.69 for Nexperia footprint recommendation.

5. Solder paste

Beside stencil aperture and thickness the used solder paste has a significant impact on the printing performance. The paste used is SAC lead free solder. Solder pastes are available in different solder powder grain sizes. Refer to Table 2.

Table 2. Survey of solder paste types (grain sizes)

Type	Less than 0.5%, larger than	10% max, between	80% min, between	10% max, less than
1	160	150 - 160	75 - 150	75
2	80	75 - 80	45 - 75	45
3	60	45 - 60	25 - 45	25
4	50	38 - 50	20 - 38	20
5	40	25 - 40	15 - 25	15
6	25	15 - 25	5 - 15	5
7	15	11 - 15	2 - 11	2

Use a solder paste type 4.5 and higher (smaller grain size) in combination with a stencil aperture thickness of 80 µm for the DSN1006-3 (SOT8026) packages.

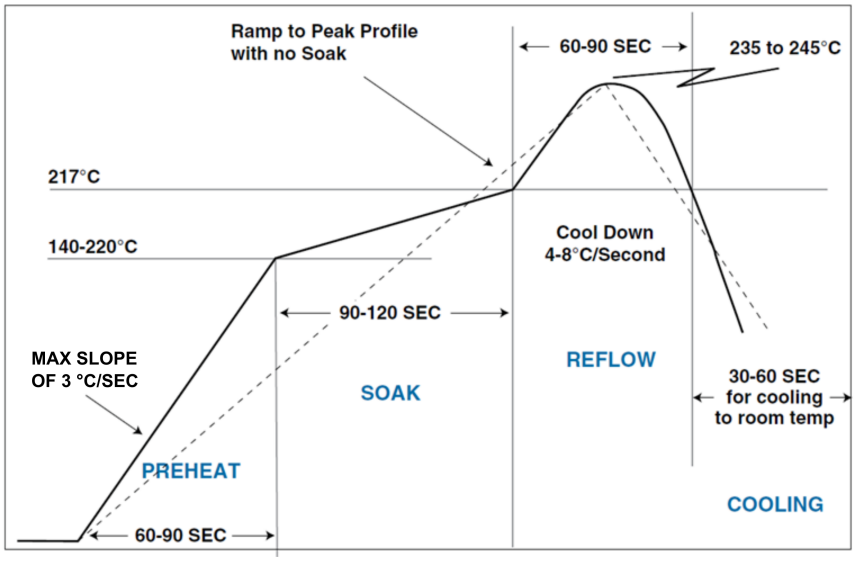
As solder paste is sensitive to age, temperature, and humidity, follow the handling recommendations of the paste manufacturer.

6. Soldering process

For soldering of DSN1006-3 packages, following solder processes were taken into account:

- **Convection reflow under nitrogen atmosphere is clearly preferred**
- Convection reflow under air atmosphere also works, but:
  - Using an unfavorable layout, products lean towards undefined tilting and rotation and solder joints show a tendency of increased voiding
  - Solder joint surfaces are rough, flux residues often become darker and the soldering behavior may deteriorate
- Vapor phase soldering is also possible

For investigation of reflow soldering a profile as recommended for SAC Alloys by IPC-7095 was applied. Refer to Fig 7.



Source IPC-7095C

Fig. 7. Reflow solder profile as applied for investigation

7. Handling recommendations

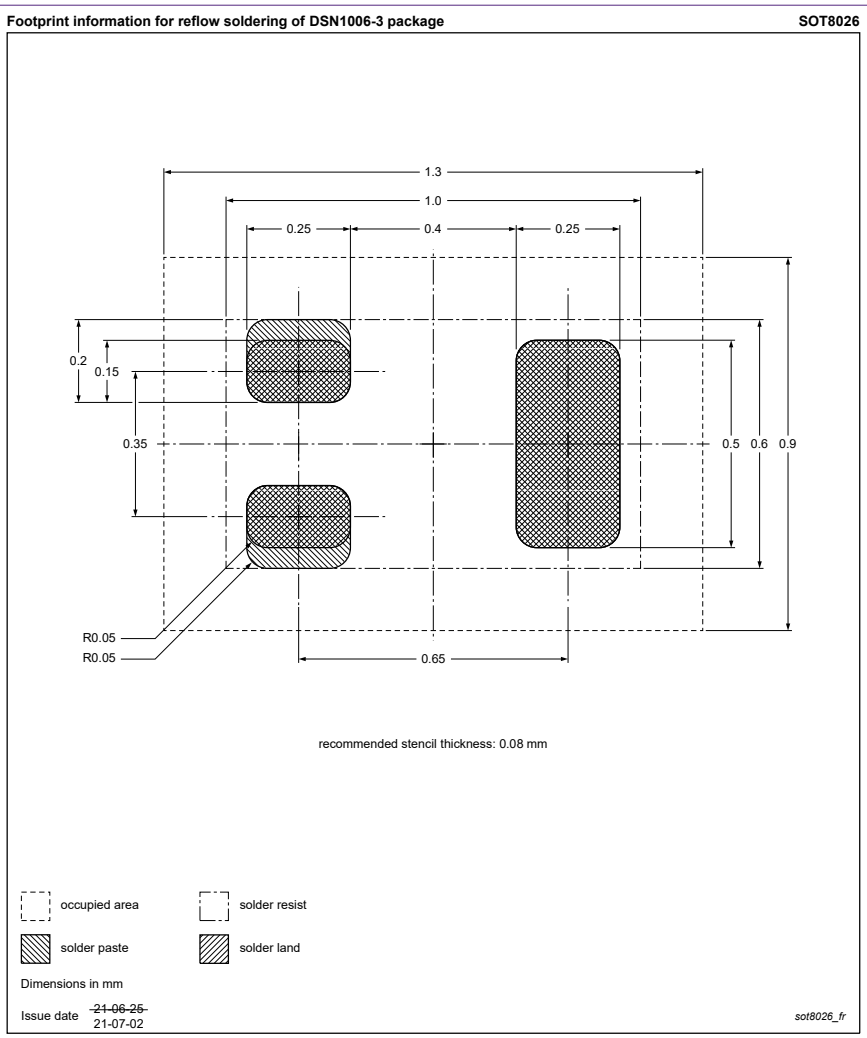
Besides the PCB and stencil design requirements, the small size of the DSN1006-3 and as consequence the low weight of the component requires that some attention be paid to the pick and place (P&P) process. One effect which may cause problems during the pick and place (tape out) process is electrostatic charge. Nexperia has implemented preventive measures in production.

During extensive P&P trials, it was observed that a relative humidity below 30% in the production area leads to increased P&P (tape out) errors caused by electrostatic charging. Therefore, the environment should be controlled to >30% RH. In any case the feeders should be carefully connected to ground to avoid electrostatic charging. Another observation is that feeders of some P&P suppliers require inserts or springs below the carrier tape. For embossed carrier tapes of such small components the inserts require a gap for the carrier tape pockets to achieve a smooth indexing without vibration. In this case, P&P machine suppliers should be contacted for recommendations. It was found that it is beneficial for an optimum tape out yield if the cover tape peel-off position is as close as possible to the pick-up position of the devices. That prevents any rotation of products due to mechanical movement and vibrations. A risk for rotation was still observed even the products covered by a metal plate after cover tape peel-off. Manual handling by tweezers (e.g. for PCB repair) is strictly not recommended.

8. Summary

8.1. Recommended solder footprint and stencil aperture

The recommended solder footprint including stencil aperture can be seen in Fig 8.



Recommended stencil thickness: 80 µm

Fig. 8. Solder footprint and stencil aperture

8.2. Further recommendations

8.2.1. Stencil layout

- Stencil printing with solder paste type 4.5 sphere size and stencil thickness of 80 µm is recommended
- Stencil aperture dimension as shown in Fig 5 and Fig 8 is recommended
- To get best printing (and soldering) results the cleaning cycle of the stencil should be well controlled

8.2.2. Solder pad design

- NSMD pads with a gap between Cu pad and solder resist of 50 µm recommended
- Conductor (Cu trace) between solder pads on PCB is not recommended
- Connection of solder pads by µVia is not recommended
- Connection by Cu traces (lines) is preferred
- Connection by route through might also possible, but due to large solder mask clearance, devices tend to tilt and rotate

8.2.3. Solder paste

Solder paste composition SAC

- Type 4.5 solder paste is recommended

8.2.4. Soldering process

- Convection reflow under nitrogen atmosphere is preferred
- Convection reflow under air atmosphere also works, but:
  - Using an unfavorable layout, products lean towards undefined tilting and rotation and solder joints show a tendency of increased voiding
  - Solder joint surfaces are rough, flux residues often become darker and the soldering behavior may deteriorate
- Vapor phase soldering is also possible

8.2.5. Handling recommendations

- Manual handling with tweezers (e.g. for repair) is very unfavorable
- Feeder of P&P machines: In case inserts required underneath the carrier tape a gap in this insert for the carrier tape pocket should be implemented. Asked P&P machine supplier for further recommendations
- Keep control of thawing time of solder paste bundle to avoid too much humidity in paste
- To prevent drying of flux in solder paste, maintain the relative humidity of shop floor at solder paste print until reflow to 40 % to 60 %
- Relative humidity for P&P should be: RH >30 %

9. Revision history

Table 3. Revision history

Revision number	Date	Description
1.0	2022-08-09	Initial version

10. Legal information

Definitions

Draft — The document is a draft version only. The content is still under internal review and subject to formal approval, which may result in modifications or additions. Nexperia does not give any representations or warranties as to the accuracy or completeness of information included herein and shall have no liability for the consequences of use of such information.

Disclaimers

Limited warranty and liability — Information in this document is believed to be accurate and reliable. However, Nexperia does not give any representations or warranties, expressed or implied, as to the accuracy or completeness of such information and shall have no liability for the consequences of use of such information. Nexperia takes no responsibility for the content in this document if provided by an information source outside of Nexperia.

In no event shall Nexperia be liable for any indirect, incidental, punitive, special or consequential damages (including - without limitation - lost profits, lost savings, business interruption, costs related to the removal or replacement of any products or rework charges) whether or not such damages are based on tort (including negligence), warranty, breach of contract or any other legal theory.

Notwithstanding any damages that customer might incur for any reason whatsoever, Nexperia's aggregate and cumulative liability towards customer for the products described herein shall be limited in accordance with the Terms and conditions of commercial sale of Nexperia.

Right to make changes — Nexperia reserves the right to make changes to information published in this document, including without limitation specifications and product descriptions, at any time and without notice. This document supersedes and replaces all information supplied prior to the publication hereof.

Suitability for use — Nexperia products are not designed, authorized or warranted to be suitable for use in life support, life-critical or safety-critical systems or equipment, nor in applications where failure or malfunction of an Nexperia product can reasonably be expected to result in personal injury, death or severe property or environmental damage. Nexperia and its suppliers accept no liability for inclusion and/or use of Nexperia products in such equipment or applications and therefore such inclusion and/or use is at the customer's own risk.

Applications — Applications that are described herein for any of these products are for illustrative purposes only. Nexperia makes no representation or warranty that such applications will be suitable for the specified use without further testing or modification.

Customers are responsible for the design and operation of their applications and products using Nexperia products, and Nexperia accepts no liability for any assistance with applications or customer product design. It is customer's sole responsibility to determine whether the Nexperia product is suitable and fit for the customer's applications and products planned, as well as for the planned application and use of customer's third party customer(s). Customers should provide appropriate design and operating safeguards to minimize the risks associated with their applications and products.

Nexperia does not accept any liability related to any default, damage, costs or problem which is based on any weakness or default in the customer's applications or products, or the application or use by customer's third party customer(s). Customer is responsible for doing all necessary testing for the customer's applications and products using Nexperia products in order to avoid a default of the applications and the products or of the application or use by customer's third party customer(s). Nexperia does not accept any liability in this respect.

Export control — This document as well as the item(s) described herein may be subject to export control regulations. Export might require a prior authorization from competent authorities.

Translations — A non-English (translated) version of a document is for reference only. The English version shall prevail in case of any discrepancy between the translated and English versions.

Trademarks

Notice: All referenced brands, product names, service names and trademarks are the property of their respective owners.

List of Tables

List of Figures

Contents

For more information, please visit: http://www.nexperia.com
For sales office addresses, please send an email to: salesaddresses@nexperia.com
Date of release: 9 August 2022