



Board Mounting Considerations for ULLGA Packages

Prepared by: Phillip Celaya
ON Semiconductor

Introduction

Ultra Leadframe Land Grid Array (ULLGA) packages are often the package of choice for optimizing device electrical performance. They are light weight, thin and minimize the use of board space. To take advantage of ULLGA packaging, special preparations and guidelines have to be followed to ensure proper mounting of the device onto the PCB. This document outlines many of the processes and board design considerations for mounting the 6- and 8-Pin ULLGAs for 3 different pad pitches.

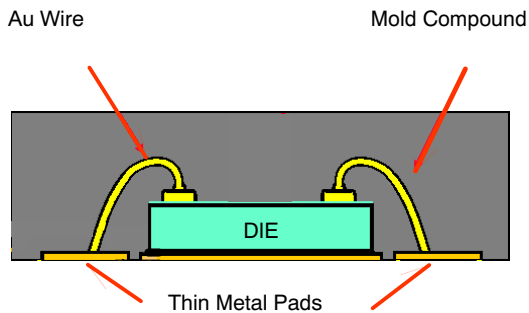


Figure 1. Cross Section of ULLGA Package

Package Overview

ULLGA packages are very similar to Quad Flat No Lead (QFN) and Dual Flat No Lead (DFN) packages. The major difference being that the ULLGA does not have a copper leadframe for the interconnect pads. Figure 1 is an example of a cross section showing a ULLGA package.

The ULLGA in general is a thinner package than the QFN/DFN type package.

Printed Circuit Board (PCB) Design

SMD and NSMD Pad Configurations

The Solder Masked Defined (SMD) and Non-Solder masked Defined (NSMD) pad configurations are commonly used for surface mount ULLGA packages.

With SMD configured pads, the solder mask covers the outside perimeter of the rectangular contact pads. With this configuration, the solder flows over the top surface of the contact pad, and is prevented from flowing along the sides of the pads by the solder mask.

With NSMD configured pads, there is a gap between the solder mask and the rectangular contact pad. With this configuration, the solder flows over the top surface and the sides of the contact pad.

Since all of the 6- and 8-Pin ULLGA packages are considered fine pitch ($\leq 0.5\text{mm}$), it is recommended that NSMD pads be used. The soldermask openings should be a window design larger than the footprint.

Recommended PCB Pad Design

Figure 2 shows the recommended board pad dimensions for 6-Pin and 8-Pin ULLGAs using the NSMD design configurations for the 0.35mm pitch packages.

Figure 3 shows the overlay of the package footprint on top of the recommended PCB footprint for the 0.35 mm pitch packages. Figures 4 and 5 show the recommended PCB footprints and package overlay on top of recommended PCB design for the 6-Pin ULLGA package for both the 0.4 and 0.5mm pitch packages.

Figure 6 and 7 show the recommended PCB footprint and package overlay on top of recommended PCB design for the 8-Pin ULLGA packages for both the 0.40 and 0.50 mm pitch packages.

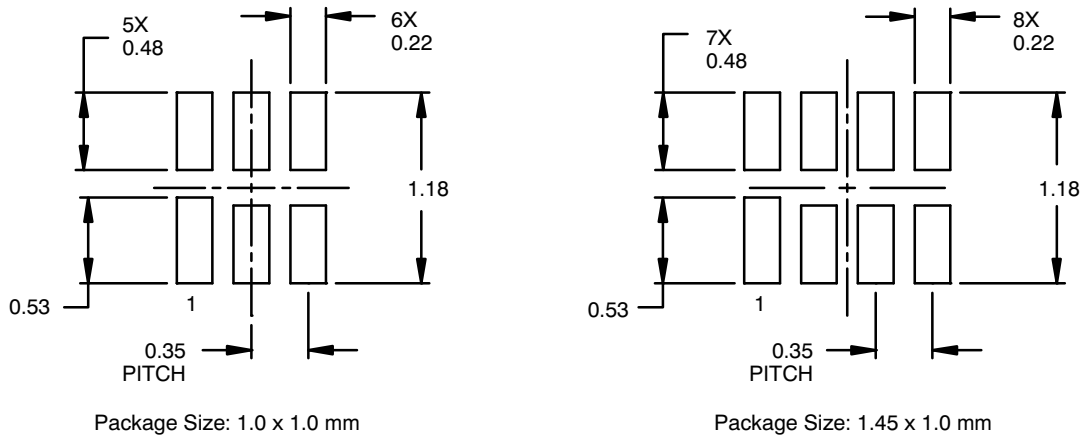


Figure 2. Recommended PCB Pad Design for 6- and 8-Pin ULLGA Packages at 0.35 mm Pitch

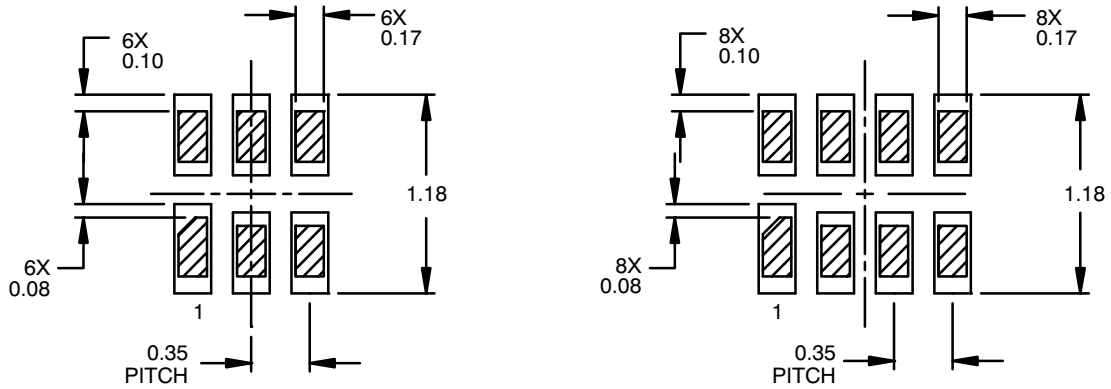


Figure 3. Package Footprint Overlaid on Recommended PCB Pad Design for 0.35 mm pitch 6- & 8-Pin ULLGA

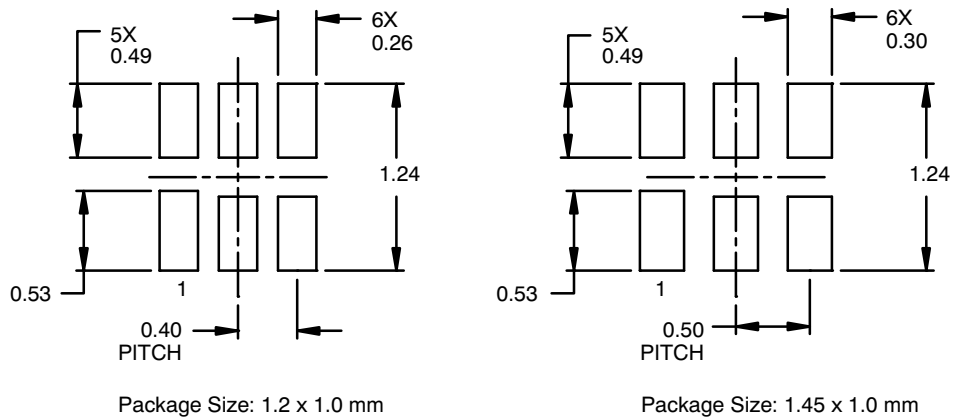


Figure 4. Recommended PCB Pad Design for 6-Pin ULLGA Packages at 0.4 and 0.5 mm Pitch

AND8317

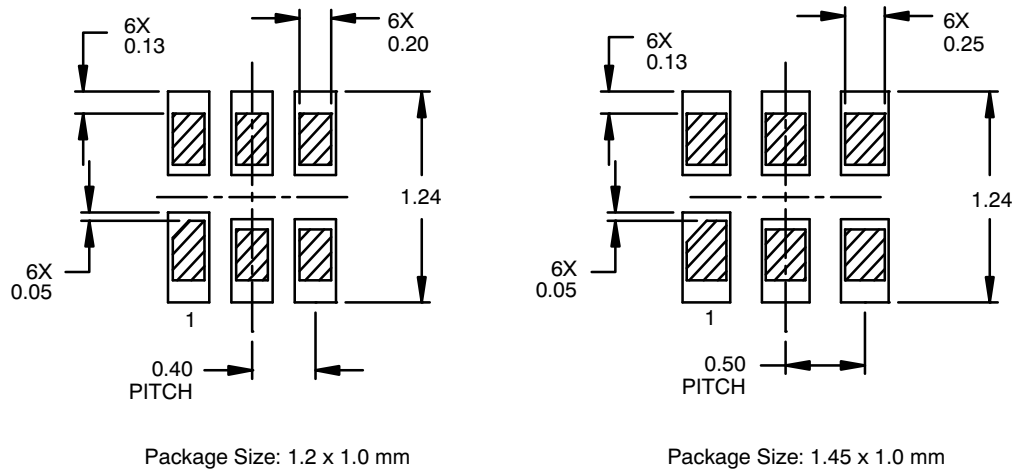


Figure 5. Package Footprint Overlaid on Recommended PCB Pad Design for 6-Pin ULLGA at 0.4 and 0.5 mm Pitch

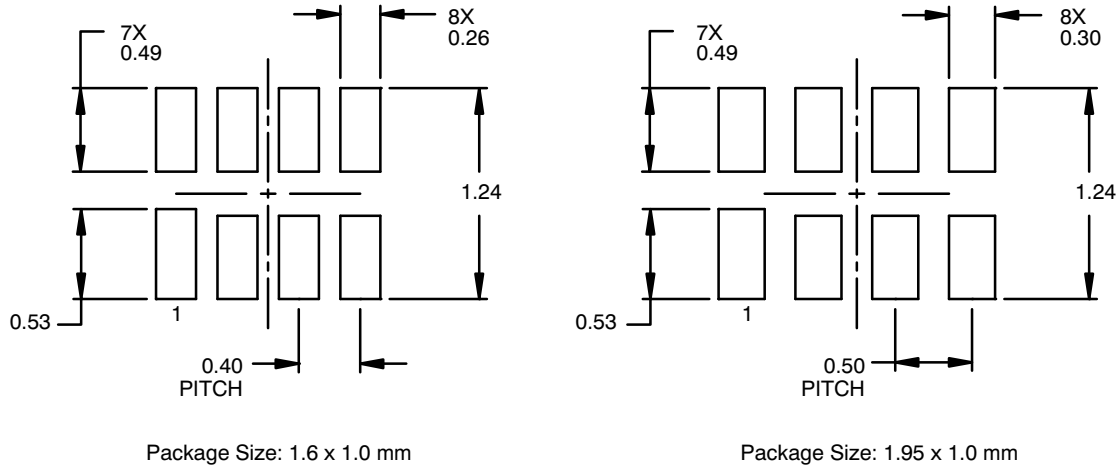


Figure 6. Recommended PCB Pad Design for 8-Pin ULLGA Packages at 0.4 and 0.5 mm Pitch

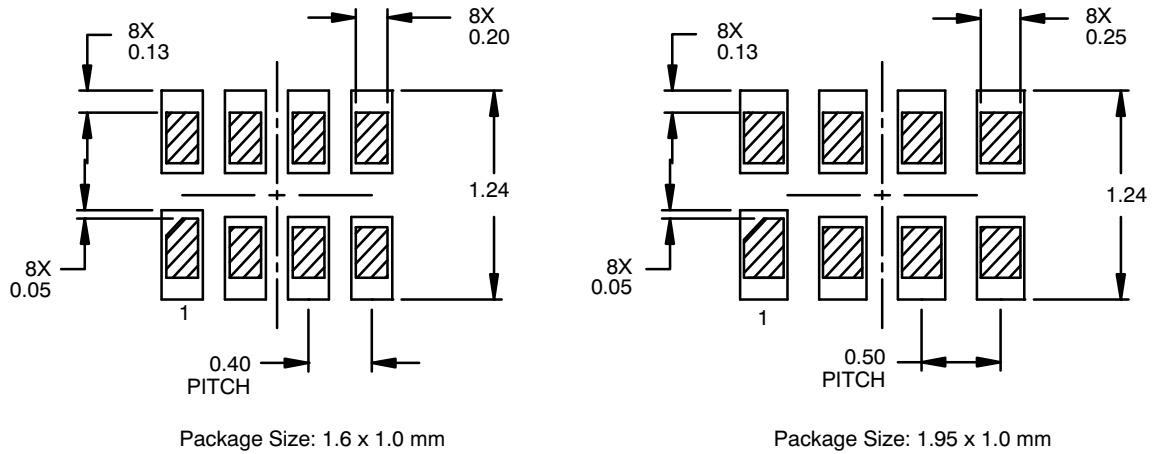


Figure 7. Package Footprint Overlaid on Recommended PCB Pad Design for 8-Pin ULLGA

The recommended PCB pad footprints were also designed so that competitor's packages would also be able to be mounted without changing the PCB pad design. Figure 8 shows the package footprint of the SOT891 package overlaid on top of the recommended PCB pad design for the 1.0x1.0mm, 6-pin ULLGA package.

Figure 9 shows the 6-pin DFN package footprint overlaid on top of the recommended PCB pad design for the 1.45x1.0 mm package size.

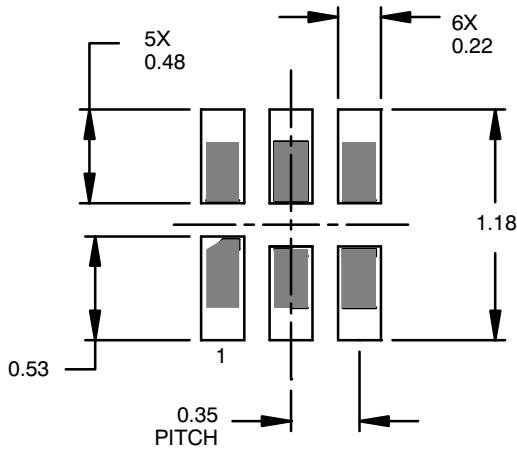


Figure 8. SOT891 Package Overlayed on Top of Recommended ULLGA PCB Pad Design

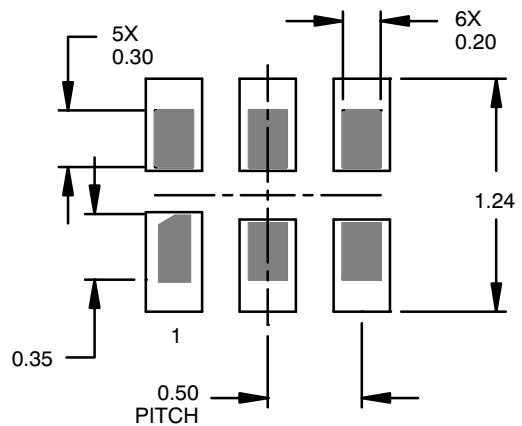


Figure 9. DFN Footprint Overlayed on Top of Recommended ULLGA PCB Pad Design

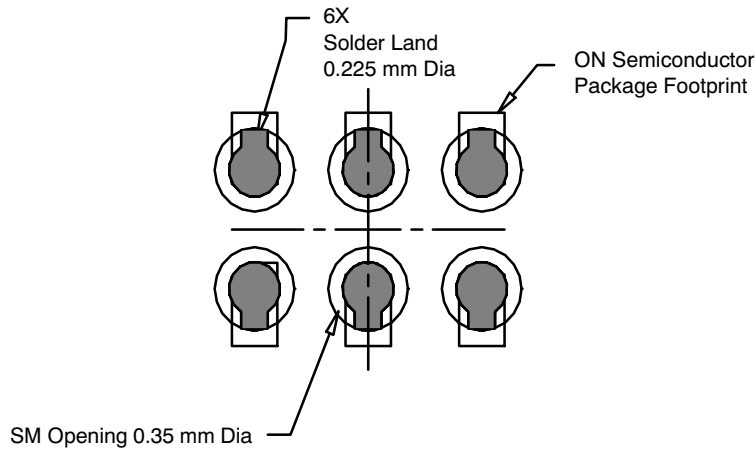


Figure 10. ON Semiconductor Package Footprint Overlayed onto BGA Footprint

SMT Process Recommendations

Plating of the PCB Pads

There are two common plated solderable metallizations which are used for PCB surface mount devices. In either case, it is imperative that the plating is uniform, conforming, and free of impurities to ensure a consistent solderable system.

The first metallization consists of plating electroless nickel over the copper pad, followed by immersion gold. The allowable stresses and the temperature excursions the board will be subjected to throughout its lifetime will determine the thickness of the electroless nickel layer. Gold thickness is recommended to be $0.15 \mu\text{m} \pm 0.05 \mu\text{m}$. Having excessive gold in the solder joint can create gold embrittlement, which may affect the reliability of the joint.

The second recommended solderable metallization is the use of an Organic Solderability Preservative coating (OSP) over the copper plated pad. The organic coating assists in preserving the copper metallization for soldering.

Solder Stencil Design

The solder is typically patterned onto the PCB by using a 127 to 104 μm (0.005 to 0.004 in) thick stencil screen. The stencil type should be laser-cut stainless steel and electropolished.

The stencil aperture opening sizes should be oblong shaped openings. The sizes of these openings are shown below in Figures 11 and 12 for the various pad pitches of the ULLGA packages.

This screen is designed and manufactured to only allow a specific amount of solder to be placed on the bond pads. It is recommended that the side walls of the screen openings be tapered approximately 5 degrees to facilitate the release of the paste when the screen is removed from the PCB.

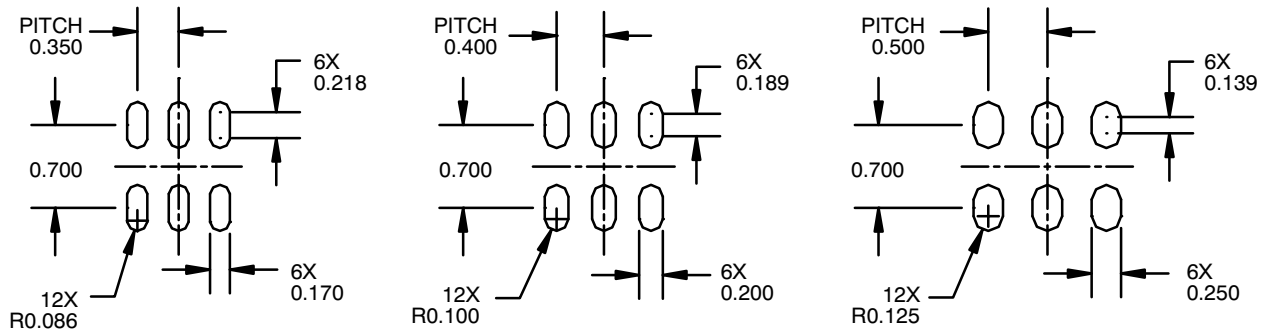


Figure 11. Recommended Stencil Designs for 6-Pin ULLGA Packages with Different Pitches

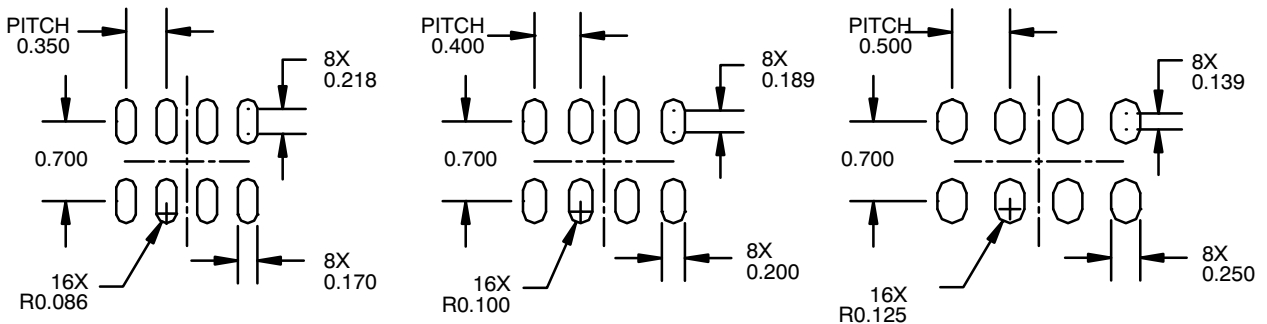


Figure 12. Recommended Stencil Designs for 8-Pin ULLGA Packages with Different Pitches

Solder Type

Either type 4 or 5 solder paste is recommended for surface mount applications. Solder paste must be a no-clean type paste. ULLGAs once mounted to the PCB board, will have no stand-off height between the package and the PCB board. Therefore, cleaning under the package is not an option.

Package Placement

Pick and place equipment with the standard tolerance of $\pm 0.10 \text{ mm}$ or better is recommended. Lower force may be

recommended especially since ULLGA pads pitches are very small and do not allow much spacing between pads. The use of low-air force nozzles are recommended to reduce the risk of shorting during the SMT process. A force of 50–70 grams is recommended when placing the packages onto the PCB board.

Solder Reflow

Once the package is placed on the PC board along with the solder paste, a standard surface mount reflow process can be used to mount the part. Forced convection reflow in nitrogen is recommended.

An example of a standard Pb-free profile is shown in Figure 13. The exact recommended reflow profile will be

determined, and is available, by the manufacture of the paste since the chemistry and viscosity of the flux matrix will vary. These variations will require small changes in the profile in order to achieve an optimized process.

In general, the peak temperature of the profile should be between 210–225°C for Eutectic Solder profiles and 250–260°C for Pb-Free solder profiles.

AC CHARACTERISTICS

Profile Features	Pb-Free Assembly
Average Ramp-up Rate (T_{smax} to T_p)	3°C/Second Max
Preheat Temperature Min (T_{smin}) Temperature Max (T_{smax}) Time (t_{smin} to t_{smax})	150°C 200°C 60–180 Seconds
Time Maintained Above Temperature (T_T) Time (t_T)	217°C 60–150 Seconds
Peak Classification Temperature (T_p)	260°C +5/-09
Time Within 55°C of Actual Peak Temperature (t_p)	20–40 Seconds
Ramp Down Rate	6°C/Second Max
Time 25°C to Peak Temperature	8 Minutes Max

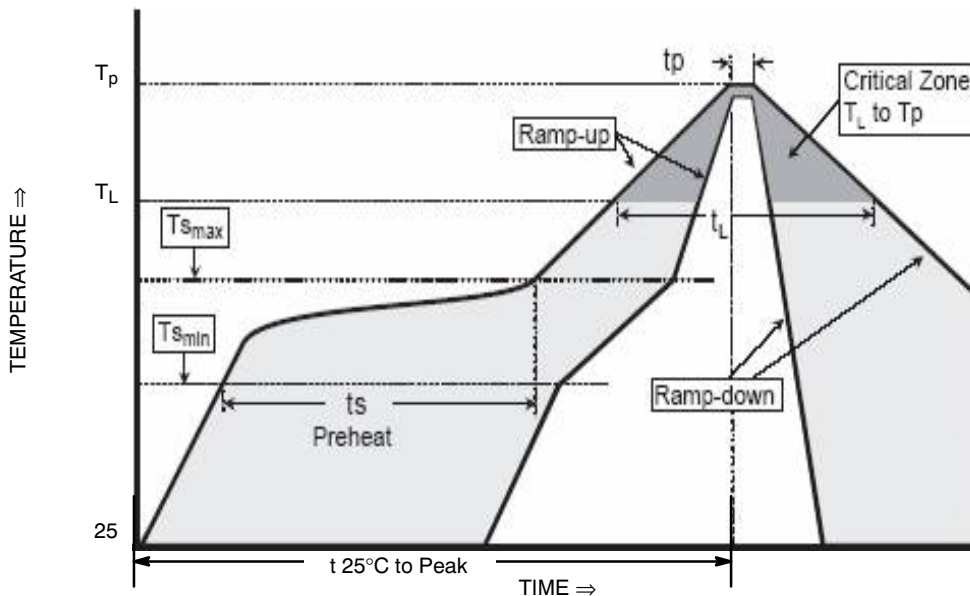


Figure 13. Typical Pb-Free Solder Heating Profile

Solder Joint Inspection

The inspection of the solder joints is commonly performed with the use of an X-ray inspection system. With this tool, one can locate defects such as shorts between pads, open contacts, voids within the solder as well as any extraneous solder.

Rework Process

It is important to minimize the chance of overheating neighboring devices during the removal of the package since these devices are typically in close proximity to other packages, Standard SMT rework systems are recommended

for this procedure since the airflow and temperature gradients can be carefully controlled. Nitrogen atmosphere is typically used to control the pads from oxidizing. It is also recommended that the PC board be placed in an oven at 125°C for 12 hours prior to heating the parts to remove excess moisture from the packages.

Once the solder joints are heated above their liquidus temperature, the package can be removed and the pads on the PC board cleaned. The solder paste is again dispensed and a new package can be reflowed onto the PC board.

ON Semiconductor and **ON** are registered trademarks of Semiconductor Components Industries, LLC (SCILLC). SCILLC reserves the right to make changes without further notice to any products herein. SCILLC makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does SCILLC assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation special, consequential or incidental damages. "Typical" parameters which may be provided in SCILLC data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including "Typicals" must be validated for each customer application by customer's technical experts. SCILLC does not convey any license under its patent rights nor the rights of others. SCILLC products are not designed, intended, or authorized for use as components in systems intended for surgical implant into the body, or other applications intended to support or sustain life, or for any other application in which the failure of the SCILLC product could create a situation where personal injury or death may occur. Should Buyer purchase or use SCILLC products for any such unintended or unauthorized application, Buyer shall indemnify and hold SCILLC and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that SCILLC was negligent regarding the design or manufacture of the part. SCILLC is an Equal Opportunity/Affirmative Action Employer. This literature is subject to all applicable copyright laws and is not for resale in any manner.

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:

Literature Distribution Center for ON Semiconductor
P.O. Box 5163, Denver, Colorado 80217 USA
Phone: 303-675-2175 or 800-344-3860 Toll Free USA/Canada
Fax: 303-675-2176 or 800-344-3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800-282-9855 Toll Free
USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81-3-5773-3850

ON Semiconductor Website: www.onsemi.com
Order Literature: <http://www.onsemi.com/orderlit>
For additional information, please contact your local
Sales Representative