

## Board Level Application Note for XLLGA 3-Lead 0.62x0.62 Package



ON Semiconductor®

<http://onsemi.com>

### APPLICATION NOTE

#### Introduction

ON Semiconductor has introduced a new XLLGA 3 lead 0.62x0.62 package. This application note will provide suggested board mounting guidelines. These guidelines include printed circuit board mounting pads, solder mask and stencil pattern, and assembly process parameters.

#### Package Overview

The XLLGA is an extremely thin leadframe land grid array package with solderable metal contacts on the underside of the package, similar to DFN style packages. The finished package is shown in Figure 1. Figure 2 is a dimensioned view of the bottom of the package.

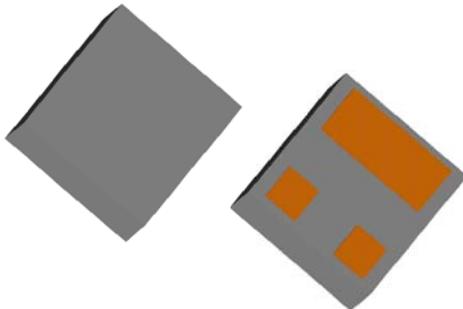


Figure 1. XLLGA 3-Lead 0.62x0.62 Package

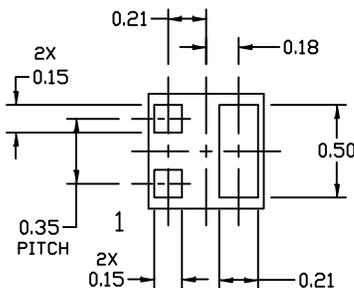


Figure 2. Dimensioned Bottom View of Package

#### Board Mounting Process

The package board mounting process can be optimized by first defining and controlling the following:

1. Solderable metallization and design of the PCB mounting pads.
2. Solder mask design guidelines.
3. Stencil for applying solder paste on to the PCB mounting pads.
4. Choice of proper solder paste.
5. Package placement.
6. Reflow of the solder paste.
7. Final inspection of the solder joints.
8. PCB circuit trace width.

Recommendations for each of these items are included in this application note. Figure 3 illustrates the color scheme used throughout this document.



Figure 3. Color Legend

Figure 4 is an illustration of a cross section of the package mounted on a PCB.

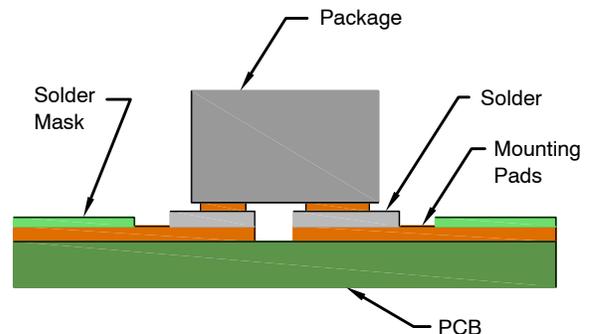


Figure 4. Cross-Section of Mounted Package

#### Printed Circuit Board Solder Pad Design

Based on results of board mount testing, ON Semiconductor's recommended mounting pad and solder mask opening geometry is shown in Figure 5. Minimum acceptable mounting pad and solder mask opening geometry is shown in Figure 6. These patterns were determined by comparing shear data and visual inspection of several mounting pad and solder mask opening geometries.

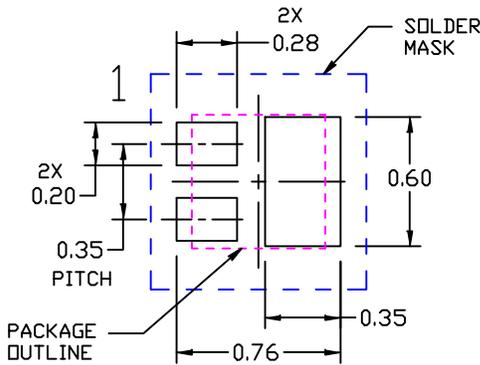


Figure 5. Recommended Mounting Pattern

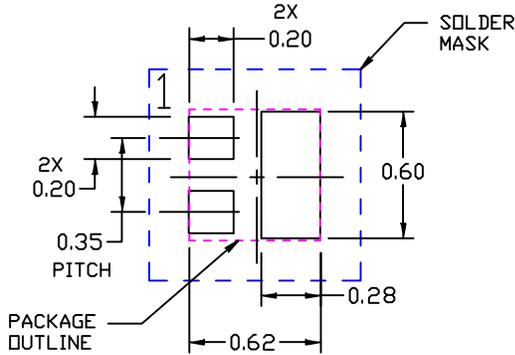


Figure 6. Minimum Acceptable Mounting

**Solder Mask**

Two types of PCB solder mask openings commonly used for surface mount leadless style packages are:

1. Non Solder Masked Defined (NSMD)
2. Solder Masked Defined (SMD)

The solder mask is pulled away from the solderable metallization for NSMD pads, while the solder mask overlaps the edge of the metallization for SMD pads as shown in Figure 7. For SMD pads, the solder mask restricts the flow of solder paste on the top of the metallization and prevents the solder from flowing down the side of the metal pad. This is different from the NSMD configuration where the solder flows both across the top and down the sides of the PCB metallization.

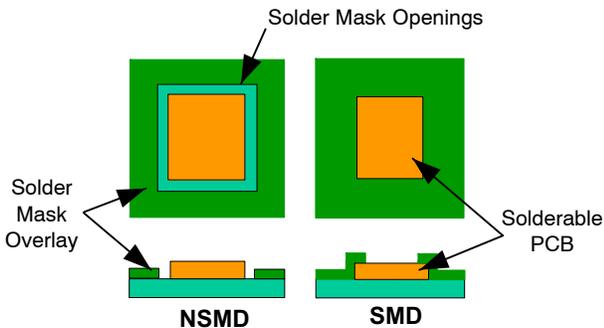


Figure 7. Comparison of NSMD vs. SMD Pads

Typically, NSMD pads are preferred over SMD pads. It is easier to define and control the location and size of copper pad versus the solder mask opening. This is because the copper etch process capability has a tighter tolerance than

that of the solder mask process. NSMD pads also allow for easier visual inspection of the solder fillet.

Many PCB designs include a solder mask web between mounting pads to prevent solder bridging. For this package, testing has shown that the solder mask web can cause package tilting during the board mount process. Thus, a solder mask web is not recommended.

**PCB Solderable Metallization**

There are currently three common solderable coatings which are used for PCB surface mount devices- OSP, ENiAu, and HASL.

The first coating consists of an Organic Solderability Protectant (OSP) applied over the bare copper features. OSP coating assists in reducing oxidation in order to preserve the copper metallization for soldering. It allows for multiple passes through reflow ovens without degradation of solderability. The OSP coating is dissolved by the flux when solder paste is applied to the metal features. Coating thickness recommended by OSP manufacturers is between 0.2 and 0.6 microns.

The second coating is plated electroless nickel/immersion gold over the copper pad. The thickness of the electroless nickel layer is determined by the allowable internal material stresses and the temperature excursions the board will be subjected to throughout its lifetime. Even though the gold metallization is typically a self-limiting process, the thickness should be at least 0.05  $\mu\text{m}$  thick, but not consist of more than 5% of the overall solder volume. Excessive gold in the solder joint can create gold embrittlement. This may affect the reliability of the joint.

The third PCB pad protective coating option is Hot Air Solder Level (HASL) SnPb. Since the HASL process is not capable of producing solder joints with consistent height, this pad finish is not recommended. Inconsistent solder deposition results in dome-shaped pads of varying height. As the industry moves to finer and finer pitch, solder bridging between mounting pads becomes a common problem with this coating.

It is imperative that the coating is conformal, uniform, and free of impurities to insure a consistent mounting process. Due to the small size of the DSN, only the electroless nickel / immersion gold metallization over the copper pads is recommended.

**PCB Circuit Trace Width**

The width of the PCB circuit trace can play an important role in the reduction of component tilting during solder reflow. Due to the small size of the solder pad and component, solder deposited on the PCB pads may form dome shaped bumps. When these solder bumps are uneven in height, the reflowed DSNs may not lie flat on the PCB. The packages may be tilted. Component tilt can be minimized by allowing a controlled amount of solder to wick away from the pad along the metal trace. This reduces the overall thickness of the solder remaining on the PCB pad under the component and in turn prevents the solder from

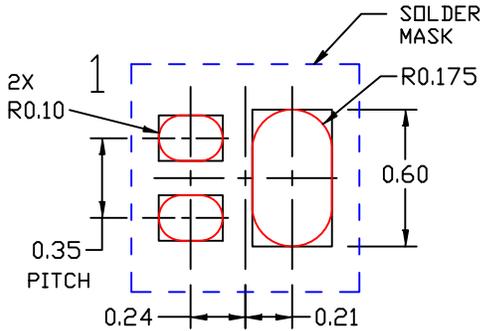
forming a ball on the PCB pad. Degree of solder run out is controlled by the solder mask opening.

**Solder Type**

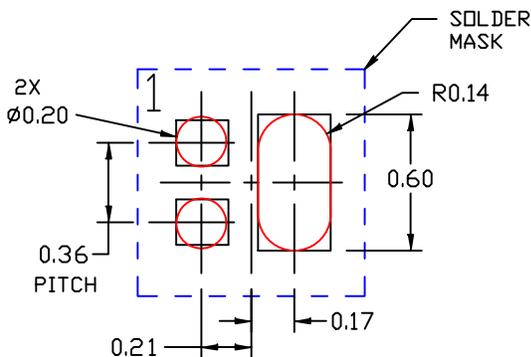
Solder pastes such as Cookson Electronics' WS3060 with a Type 4 or smaller sphere size are recommended. WS3060 has a water-soluble flux for cleaning. Cookson Electronics' PNC0106A can be used if a no-clean flux is preferred.

**Solder Stencil Screening**

Stencil screening of the solder paste onto the PCB is commonly used in the industry. The recommended stencil thickness for this part is 0.076 mm (0.003 in). The sidewalls of the stencil openings should be tapered approximately five degrees along with an electro-polish finish to aid in the release of the paste when the stencil is removed from the PCB. See Figure 8 for recommended stencil opening size and pitch shown on the recommended PCB mounting pads and solder mask opening from Figure 5. See Figure 9 for recommended stencil opening size and pitch shown on the minimal acceptable PCB mounting pads and solder mask opening from Figure 6.



**Figure 8. Recommended Stencil Pattern**



**Figure 9. Minimum Acceptable Mounting Stencil Pattern**

**Package Placement**

An automated pick and place procedure with magnification is recommended for component placement since the pads are on the underside of these very small packages. A dual image optical system enables alignment of the underside of the package to the PCB and should be used. Pick and place equipment with a standard tolerance of +/- 0.05 mm (0.002 in) or better is recommended. Once placed onto the board, the package self-aligns during the reflow process due to surface tension of the solder.

Note: A Pressure Sensitive Adhesive (PSA) tape and reel cover tape is recommended for best pick and place results.

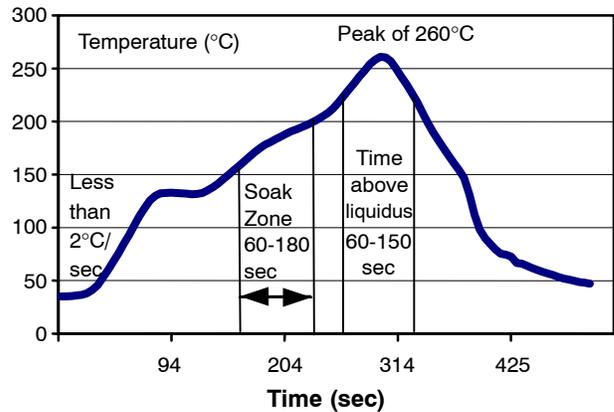
**Solder Reflow**

Once the component is placed on the PCB, a standard surface mount reflow process can be used to mount the part. Figures 10 and 11 are examples of typical reflow profiles for lead free and standard eutectic tin lead solder alloys, respectively.

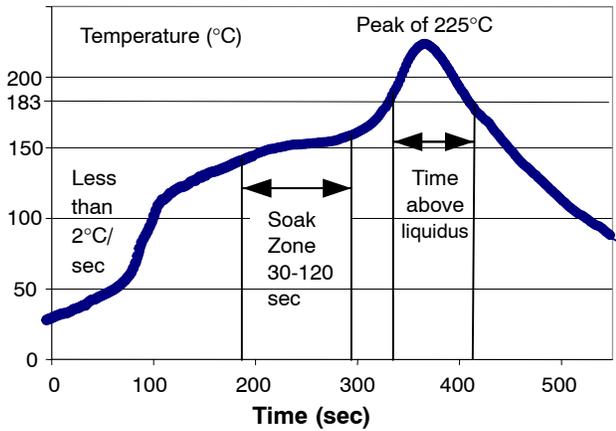
The preferred profile is provided by the solder paste manufacturer and is dictated by variations in chemistry and viscosity of the flux matrix in the solder paste. These variations may require small adjustments to the profile for process optimization.

In general, the temperature of the part should increase by less than 2°C/sec during the initial stages of reflow. The soak zone occurs at approximately 150°C and should last for 60 to 180 seconds for lead free profiles (30-120 sec for eutectic tin lead profiles). Typically, extending the length of time in the soak zone reduces the risk of voiding within the solder. The temperature is then increased. Time above the liquidus of the solder is limited to 60 to 150 seconds for lead free profiles (30-100 sec for eutectic tin lead profiles) depending on the mass of the board. The peak temperature of the profile should be between 245 and 260°C for lead free solder alloys (205-225°C for eutectic tin lead solders).

If required, removal of the residual solder flux can be done using the recommended procedures set forth by the flux manufacturer.



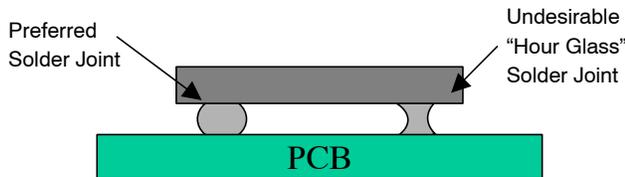
**Figure 10. Typical Reflow Profile Lead Free Solder**



**Figure 11. Typical Reflow Profile Eutectic Tin / Lead Solder**

**Final Solder Inspection**

Solder joint integrity is determined by using an X-ray inspection system. With this tool, defects such as shorts between pads, open contacts, and voids within the solder and extraneous solder can be identified. In addition, the mounted device should be rotated on its side to inspect the sides of the solder joints for acceptable solder joint shape and stand-off height. The solder joints should have enough solder volume and stand-off height so that an “Hour Glass” shaped connection is not formed as shown in Figure 12. “Hour Glass” solder joints are a reliability concern and should be avoided.



**Figure 12. Illustration of Preferred and Undesirable Solder Joints**

**Rework Procedure**

Since this package is a leadless device, the package must be removed from the PCB if there is an issue with the solder joints.

Standard SMT rework systems are recommended for this procedure since airflow and temperature gradients can be carefully controlled. It is also recommended that the PCB be placed in an oven at 125°C for 4 to 8 hours prior to package removal to remove excess moisture from the packages. To control the region to be exposed to reflow temperatures, the PCB should be heated to 100°C by conduction through the backside of the board in the location of the device. Typically, heating nozzles are then used to increase the temperature locally and minimize any chance of overheating neighboring devices in close proximity.

Once the device’s solder joints are heated above their liquidus temperature, the package is quickly removed and the pads on the PCB are cleaned. Cleaning of the pads is typically performed with a blade style conductive tool with a de-soldering braid. A no clean flux is used during this process to simplify the procedure.

Solder paste is then deposited or screened onto the site in preparation for mounting a new device. Due to the close proximity of the neighboring packages in most PCB configurations, a miniature stencil for the individual component is typically required. The same stencil design parameters can be applied to this new stencil for redressing the pads.

Again, a manual pick and place procedure with the aid of magnification is recommended. A system with the same capabilities as described in the Package Placement section should be used.

Remounting the component onto the PCB can be accomplished by either passing it through the original reflow profile, or by selectively heating the specific region on the PCB using the same process used to remove the defective package. The benefit of subjecting the entire PCB to a second reflow is that the new part will be mounted consistently using a previously defined profile. The disadvantage is that all of the other soldered devices will be reflowed a second time. If subjecting all of the parts to a second reflow is either a concern or unacceptable for a specific application, then the localized reflow option is the recommended procedure.

Optimal board mounting results can be achieved by following these suggested guidelines.

**ON Semiconductor** and **ON** are registered trademarks of Semiconductor Components Industries, LLC (SCILLC). SCILLC owns the rights to a number of patents, trademarks, copyrights, trade secrets, and other intellectual property. A listing of SCILLC’s product/patent coverage may be accessed at [www.onsemi.com/site/pdf/Patent-Marking.pdf](http://www.onsemi.com/site/pdf/Patent-Marking.pdf). 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](mailto: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: 81-3-5817-1050

**ON Semiconductor Website:** [www.onsemi.com](http://www.onsemi.com)  
**Order Literature:** <http://www.onsemi.com/orderlit>

For additional information, please contact your local Sales Representative