ON Semiconductor

Is Now

To learn more about onsemi™, please visit our website at
www.onsemi.com

AND8455/D

Board Level Application Note for SOD-923 Two Pin Diode Package

ON Semiconductor®

www.onsemi.com

APPLICATION NOTE

Introduction

ON Semiconductor's SOD-923 represents the latest in small surface mount two pin Diode package technology. Due to the small size of the package, it is important that the mounting process follow the suggested guidelines outlined in this document. This includes printed circuit board mounting pads, the soldermask and stencil pattern, and the assembly process parameters.

Package Overview

Where space is limited, the SOD-923 package size reduces board space requirements. A finished package is shown in Figure 1. Figure 2 is a dimensioned view of the bottom of the package.

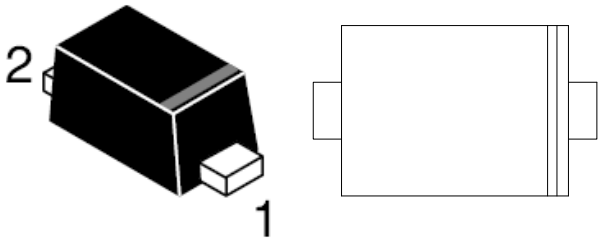

Figure 1. SOD−923 Two Pin Package

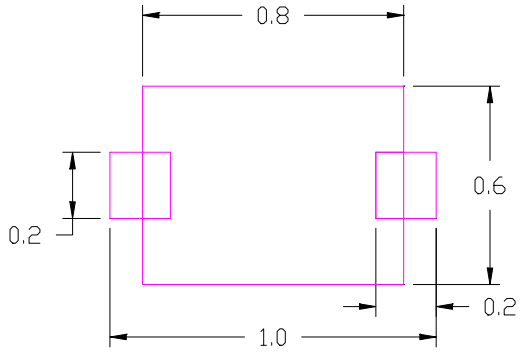

Figure 2. Dimensioned Bottom View of Package

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.

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

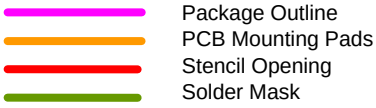

Figure 3. Color Legend

Figure 4 shows the size and orientation of the package on the recommended PCB mounting pads, solder mask and solder stencil. Figure 5 is an illustration of a cross section of the package mounted on a PCB.

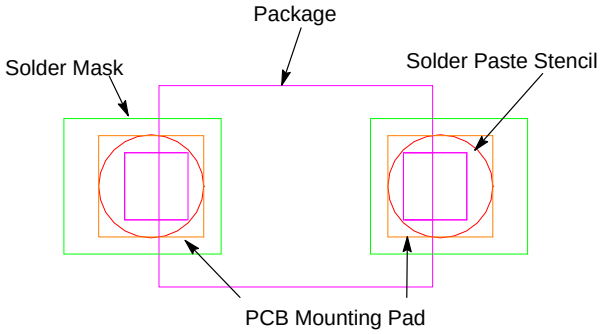

Figure 4. Package shown on Recommended Mounting Pattern including Stencil

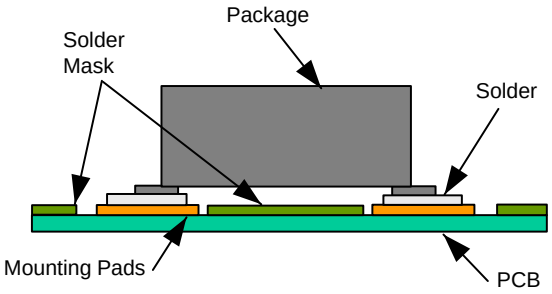

Figure 5. Cross-Section of Mounted Package

Printed Circuit Board Solder Pad Design

Based on results of board mount testing, ON Semiconductor's recommended mounting pads and solder mask opening are shown in Figure 6. During these studies, solder mounting patterns for both SC101AC and SOD-882 were also evaluated and found acceptable. Figure 7 shows the maximum acceptable PCB mounting pads and solder mask openings.

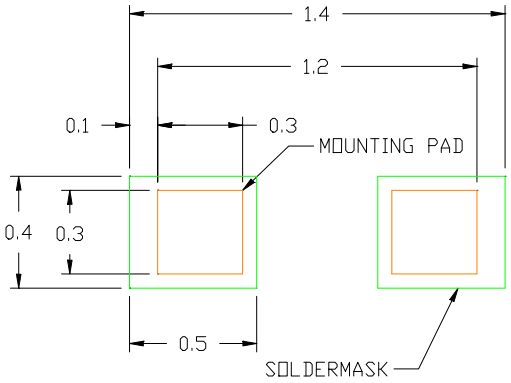

Figure 6. Recommended Mounting Pattern

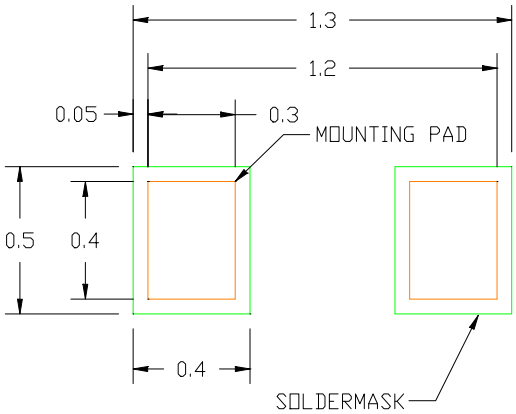

Figure 7. Maximum Recommended Mounting

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 8. 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.

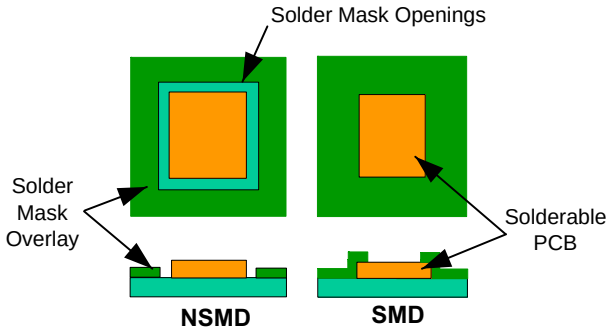

Figure 8. Comparison of NSMD vs. SMD Pads

Typically, NSMD pads are preferred over SMD pads. It is easier to define and control the location and size of copper pad verses 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.

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 0.25 and 0.35 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 μm 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 is a tin-lead coating, commonly called Hot Air Solder Level (HASL). This type of PCB pad finish is not recommended for this type packages. The major issue is the inability to consistently control the amount of solder coating applied to each pad. This results in dome-shaped pads of

various heights. As the industry moves to finer and finer pitch, solder bridging between mounting pads becomes a common problem when using this coating.

It is imperative that the coating is conformal, uniform, and free of impurities to insure a consistent mounting process. Due to the package's extremely small size, we only recommend the use of the electroless nickel/ immersion gold metallization over the copper pads.

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 0.1 mm (0.005 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 9 for the recommended stencil opening size and pitch shown on the recommended PCB mounting pads and solder mask opening from Figure 6.

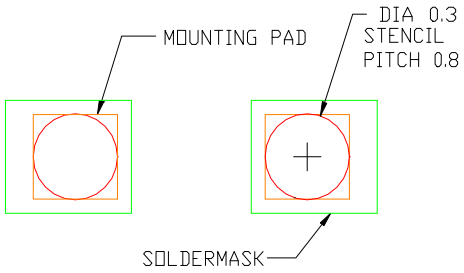

Figure 9. Recommended Stencil Pattern.

A second stencil option is shown in Figure 10. This option increases the amount of solder paste applied to the PCB through the stencil. This second option increases the stencil opening size and pitch. The PCB mounting pads and solder mask opening on the board do not change from the recommendations in Figure 6.

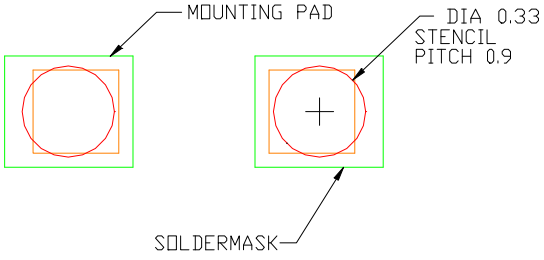

Figure 10. Maximum Stencil Pattern

Note: If the maximum stencil opening option from Figure 10 is used, tilt may occur on some of the packages. This was evident in the board mounting study we conducted. Using this stencil with the largest openings may improve solder release from the stencil along with slightly increasing the package shear strength but as stated, package tilt or undesirable movement in the mounted package may occur.

Package Placement

Due to the small package size and because the pads are on the underside of the package, an automated pick and place procedure with magnification is recommended. A dual image optical system where the underside of the package can be aligned to the PCB should be used. Pick and place equipment with a standard tolerance of +/- 0.05 mm (0.002 in) or better is recommended. The package self aligns during the reflow process due to the surface tension of the solder.

Solder Reflow

Once the package is placed on the PCB, a standard surface mount reflow process can be used to mount the part. Figures 11 and 12 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. These variations may require small adjustments to the profile for an optimized process.

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.

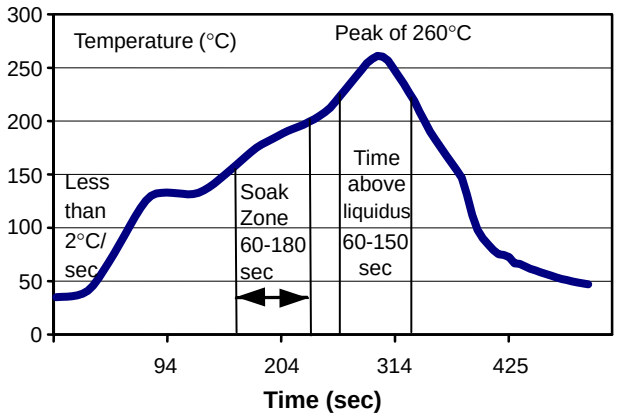

Figure 11. Typical Reflow Profile Lead Free Solder

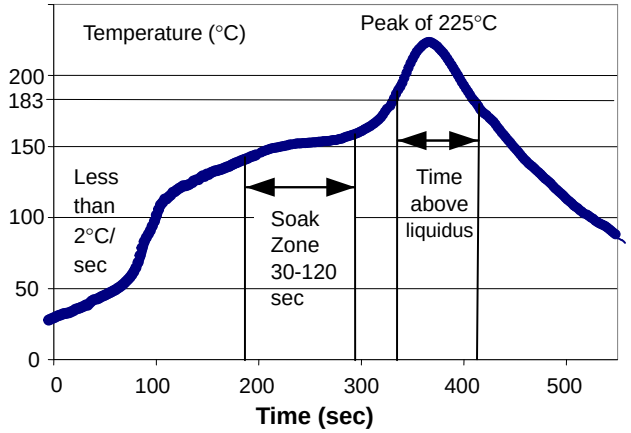

Figure 12. Typical Reflow Profile Eutectic Tin / Lead Solder

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 13. “Hour Glass” solder joints are a reliability concern and should be avoided.

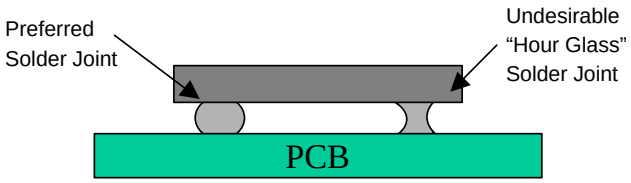

Figure 13. Illustration of Preferred and Undesirable Solder Joints

Rework Procedure

Since the SOD−923 two pin package is a Flat Leaded 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.

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: 81−3−5817−1050

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative