Board Level Application Note for 0402, 0502 and 0603 DSN2 Packages

Prepared by: Denise Thienpont, Steve St. Germain
ON Semiconductor

Introduction
ON Semiconductor has introduced an expanded family of 2-lead Schottky DSN packages (Dual Silicon No-lead) of various sizes. These include the 0402, 0502 and 0603 (1.0x0.6mm, 1.4x0.6mm and 1.6x0.8mm, respectively). It is important to follow the suggested board mounting guidelines outlined in this document. These guidelines include printed circuit board mounting pads, soldermask and stencil pattern and assembly process parameters.

Package Overview
The DSN2 package is a chip level package with solderable metal contacts on the underside of the package, very similar to DFN style packages. The DSN style package enables 100% utilization of the package area of the active silicon, thus offering a significant performance per board area advantage as compared with products in plastic molded packages. The finished package is shown in Figure 1.

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.
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 2 illustrates the color scheme used throughout this document.

Figure 2. Color Legend

Figure 3 is an illustration of a cross section of the package mounted on a PCB.
**Printed Circuit Board Solder Pad Design**

Suggested guidelines for board assembly of the 0402, 0502 and 0603 DSNs are given in Table 1. The preferred PCB mounting pattern for each device was determined by comparing shear data and visual inspection of several pad geometry / solder mask opening / stencil opening configurations.

**Table 1**

<table>
<thead>
<tr>
<th></th>
<th>0402 DSN</th>
<th>0502 DSN</th>
<th>0603 DSN</th>
</tr>
</thead>
<tbody>
<tr>
<td>dimensioned</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>bottom view of DSN</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>recommended</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PCB pad and</td>
<td><img src="image4" alt="Image" /></td>
<td><img src="image5" alt="Image" /></td>
<td><img src="image6" alt="Image" /></td>
</tr>
<tr>
<td>soldermask</td>
<td>pattern</td>
<td>pattern</td>
<td>pattern</td>
</tr>
<tr>
<td>recommended</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>stencil dimensions</td>
<td><img src="image7" alt="Image" /></td>
<td><img src="image8" alt="Image" /></td>
<td><img src="image9" alt="Image" /></td>
</tr>
</tbody>
</table>
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 4. 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.

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.

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.

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 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 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.

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 paste 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.

Stencil screening of the solder paste onto the PCB is commonly used in the industry. The recommended stencil thickness for this part is 0.127 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 Recommended stencil opening size and pitch shown on the recommended PCB mounting pads and solder mask opening is given in Table 1.
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 5 and 6 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 solder).

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

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

Rework Procedure
Since the DSN two pin 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.