This document was written to serve as a guideline for developing the proper Printed Circuit Board (PCB) design and surface mount processes for Littelfuse Circuit Protection's 0201-size Silicon ESD (SESD) devices. Actual studies may be needed to optimize the process and meet product-specific requirements.

**Package Considerations**

Littelfuse 0201-sized SESD devices are approximately 70% smaller than prior generation devices. They are designed to help provide protection for and improve reliability of portable electronics, such as mobile phones, MP3 players, PDAs and digital cameras. The SESD device’s ultra-small footprint — measuring a mere 0.6mm x 0.3mm x 0.3mm — offers designers flexibility in space-constrained applications.

Due to the small size of the SESD device’s package, it is important that the mounting process be consistent with the suggested guidelines outlined in this document. This includes printed circuit board mounting pads, the solder mask and stencil pattern, and assembly process parameters.

To optimize utilization of the package for active silicon, the SESD device uses solderable metal contacts on the underside of the package. Compared to products in plastic molded packages, the SESD device offers a significant performance-per-board-area advantage. The SESD package and a dimensional view of the package bottom are shown in Figures 1 and 2.

**Board Mounting Considerations**

Optimizing the package board mounting process is dependent on defining and controlling the following parameters:

- Solderable metallization and design of the PCB mounting pads
- Solder mask design
- PCB circuit trace width
- Solder paste selection
- Package placement
- Solder paste reflow
- Solder stencil screening
- Final inspection of the solder joints
Recommendations for each of these processes are included in this application note.

Figure 3 shows the orientation of the package on the recommended PCB mounting pads, solder mask and solder stencil. A cross-section of the package mounted on a PCB is shown in Figure 4.

Figure 3. Recommended mounting pattern

Figure 4. Side view of mounted package

The NSMD contact pads have the solder mask pulled away from the solderable metallization, while the SMD pads have the solder mask over the edge of the metallization.

With SMD pads, the solder mask restricts the flow of solder paste on the top of metallization to help prevent the solder from flowing along the side of the metal pad.

In the NSMD process, the solder flows around both the top and sides of the metallization.

NSMD pads are generally preferred over SMD pads, since the copper etching process is capable of a tighter tolerance than the solder-masking process. Additionally, NSMD pads with a solder mask opening that is larger than the metal pad size help improve the reliability of the solder joint.

NSMD pads also facilitate visual inspection of the solder fillet. Many PCB designs include a solder mask web between mounting pads to prevent solder bridging. Testing has demonstrated that, for this package, the solder mask web can cause package tilting during the board mount process, and is not recommended.
APPLICATION NOTE

Surface-mount 0201-size SESD Devices

Based on results of board mount testing, Littelfuse Circuit Protection recommends the mounting pads and solder mask opening shown in Figure 6.

![Figure 6. Recommended mounting pads](image)

The maximum acceptable PCB mounting pads and solder mask opening recommended by Littelfuse are shown in Figure 7.

![Figure 7. Maximum acceptable mounting pads](image)

### PCB Solderable Metalization

The key factor in selecting an acceptable surface finish is to ensure that the land pads have a uniform coating. Irregular surface plating, uneven solder paste thickness, and crowning of the solder plating can reduce the overall surface mount yields.

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

The first coating is an Organic Solderability Protectant (OSP), which is applied over the bare copper features. OSP coating helps reduce and preserve the copper metallization for soldering. It also permits multiple passes through reflow ovens. The coating thickness recommended by OSP manufacturers is between 0.25 and 0.35 microns.

The second coating is plated electroless nickel/immersion gold, or ENiAu. The thickness of this layer is determined by the allowable internal material stresses and the temperature excursions the board will be subjected to throughout its lifetime. Although the gold metallization is typically a self-limiting process, the thickness should be at least 0.05 μm thick, but not exceed 5% of the overall solder volume. Excessive gold in the solder joint can create gold embrittlement and degrade the integrity of the joint.

The third coating is a tin-lead Hot Air Solder Level (HASL). This type of PCB pad finish is not recommended for the SESD package. The principal drawback with this coating is the difficulty in consistently controlling the amount of solder coating applied to each pad, which may result in dome-shaped pads of various heights.

To ensure a consistent mounting process the coating must be conformal, uniform, and free of impurities. Due to the extremely small size of the SESD package, Littelfuse recommends only using electroless nickel/immersion gold metallization to coat the device's copper pads.

### PCB Circuit Trace Width

The width of PCB circuit traces plays an important role in the reduction of component tilting when the solder is reflowed. A solderable circuit trace allows the solder to wick or run down the trace, reducing the overall thickness of the solder on the PCB and under the component.

Due to the small size of the solder pad and component, the solder on the PCB may tend to form a bump and cause the component to slide down the side of the bump, resulting in a tilted component. The best method for preventing tilting is to use a PCB circuit trace equal to the width of the mounting pad. The length of the solder wicking or run out is controlled by the solder mask opening.
**APPLICATION NOTE**

**Surface-mount 0201-size SESD Devices**

**Solder Selection**

Type 4 or smaller sphere-size solder pastes are recommended for mounting the SESD device. Water-soluble flux for cleaning is also recommended.

**Solder Stencil Screening**

A common industry practice is to stencil/screen the solder paste onto the PCB. The recommended stencil thickness for the SESD package is 0.1 mm (0.004 in). The sidewalls of the stencil openings should be tapered to approximately five degrees and include an electro-polish finish to help release the paste when the stencil is removed. The recommended stencil opening size and pitch are shown in Figure 8.

![Figure 8. Recommended stencil pattern](image)

An alternate stencil option is shown in Figure 9. This design increases the amount of solder paste applied to the PCB by increasing the stencil opening size and pitch. The PCB mounting pads and solder mask opening on the board are the same as those recommended in Figure 6.

![Figure 9. Maximum stencil pattern](image)

It is important to note that tilt may occur on some of the packages if the maximum stencil opening is used. In board-mounting studies, the stencil with the largest openings may also improve solder release and slightly increase the package’s shear strength.

**Package Placement**

Due to the small size of the package and the fact that the pads are on the underside of the package, an automated pick-and-place procedure with magnification is recommended.

A dual-image optical system that permits alignment of the underside of the package 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 since the package self-aligns during the reflow process due to the 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**

SESD devices are designed to withstand traditional lead-free reflow profiles. As with all SMT components, it is important that profiles be checked on all new board designs.

In addition, if there are multiple packages on the board, the profile must be checked at different locations on the PCB. Component temperatures may vary because of surrounding components, location of the device on the board, and package densities.

After placing the package on the PCB, a standard surface mount reflow process can be used to mount the part. Figure 10 shows a typical reflow profile for lead-free solder.
**Surface-mount 0201-size SESD Devices**

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 device should not be raised more than 2°C/sec during the initial stages of reflow. The soak zone occurs when the component is raised to approximately 150°C, which should last for 60 to 180 seconds for lead-free profiles. Typically, extending the length of time in the soak zone reduces the risk of voiding within the solder.

Next, when the temperature is raised it will be higher than the liquidus of the solder for 60 to 150 seconds for lead-free profiles depending on the mass of the board. The peak temperature of the profile should be between 245°C and 260°C for lead-free solder.

Note: To remove residual solder flux, use the procedures recommended by the flux manufacturer.

**Final Solder Inspection**

Solder joint integrity is determined by using an X-ray inspection system to detect shorts between pads, open contacts, voids within the solder, and extraneous solder. In addition to searching for these types of defects, the mounted device should also 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 as not to form an hour glass-shaped connection, as shown in Figure 11.

**Rework Procedure**

Due to the fact that the SESD device is a leadless device, the entire package must be removed from the PCB if there is any issue with the solder joints. It is important to minimize the risk of overheating neighboring devices during the removal of the package since the devices are typically in close proximity.

Standard SMT rework systems are recommended for this procedure, since the airflow and temperature gradients must 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 heating the parts to remove excess moisture from the packages.

To condition 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. Heating nozzles can then be used to increase the temperature locally and avoid overheating neighboring devices.

After the device’s solder joints are heated above their liquidus temperature, the package should be removed quickly and the pads on the PCB cleaned. Cleaning is generally done with a blade-style conductive tool with a de-soldering braid. In this instance, a no-clean flux can be used to simplify the procedure.

Next, solder paste is 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 recommended. The same stencil design parameters can be
applied to this new stencil for redressing the pads.

A manual pick-and-place procedure with the same capabilities as described in the “Package Placement” section should be used for the rework process.

The component can be remounted on the PCB by passing it through the original reflow profile, or by selectively heating the specific area of the PCB using the same process that was used to remove the defective package.

Subjecting the entire PCB to a second reflow means that the new part will be mounted consistently using a previously defined profile. The disadvantage of this method 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 unacceptable for a specific application, then the localized reflow option is recommended.

**Notice:**

Information furnished is believed to be accurate and reliable. However, users should independently evaluate the suitability of and test each product selected for their own applications. Littelfuse products are not designed for, and shall not be used for, any purpose (including, without limitation, military, aerospace, medical, life-saving, life-sustaining or nuclear facility applications, devices intended for surgical implant into the body, or any other application in which the failure or lack of desired operation of the product may result in personal injury, death, or property damage) other than those expressly set forth in applicable Littelfuse product documentation. Warranties granted by Littelfuse shall be deemed void for products used for any purpose not expressly set forth in applicable Littelfuse documentation. Littelfuse shall not be liable for any claims or damages arising out of products used in applications not expressly intended by Littelfuse as set forth in applicable Littelfuse documentation. The sale and use of Littelfuse products is subject to Littelfuse Terms and Conditions of Sale, unless otherwise agreed by Littelfuse.