

**Introduction**

The Shell-Op 3D package is a cavity package that utilizes a glass lid and is attached to the board via a solder ball array. The glass lid allows the detector on the die surface to sense blue light, while protecting the chip surface from the environment. The lead-free solder ball array and other package materials allow this package to be compatible with industry standard surface mount assembly techniques (SMT).

The purpose of this document is to outline the basic guidelines of use with Intersil Shell-Op 3D packaging technology to ensure consistent PCB assembly necessary to achieve high yield and reliability. Figure 1 shows the top, side and bottom views of the package.



FIGURE 1. PACKAGE TOP, BOTTOM AND SIDE VIEWS

**Handling Requirements and Considerations**

The Shell-Op 3D package allows the die to receive optical input on the top surface of the device, process the signal and carry the information to the ball contacts on the bottom of the package. This is accomplished with the use of a glass lid and metal traces wrapping around the edge of the package.

With these features, there are some precautions that need to be taken. The glass can chip if picked up from the side of the package by tweezers, or if they come in forceful contact with hard surfaces. The metal traces on the edge of the package can be damaged if tweezers are used and may be shorted together if they come in simultaneous contact with metallic surfaces during operation. Contamination and foreign particles on the surface of the glass should also be avoided as they may shadow the detector and cause it not to function properly. Care should be given to insure that the surface is clean after the units are mounted in assemblies.

The Shell-Op 3D package has been qualified to meet JEDEC MSL3 requirements. The units must be mounted on a board within 168 hours after opening the sealed bag. If the units are exposed to general atmospheric conditions (+30°C or warmer, 60% or more relative humidity) for more than 168 hours, they must be re-baked to remove excess moisture that may have been absorbed in the package.

**Application Board Design**

The design of the PCB mounting pads and the pad finish are critical aspects of forming an optimum solder joint. The pad should be controlled in terms of shape and size, since it contributes to the final dimensions of the solder joint between the pad and board. In the case of conventional packages, non-solder-mask-defined (NSMD) pads show higher reliability than solder-mask-defined (SMD) board designs. However, SMD is recommended for the Shell OP package due to bump size and joint shape concerns.

The size of the opening and soldermask thickness need to be tightly controlled. The soldermask tolerance should be controlled to  $\pm 30\mu\text{m}$  and the thickness should be  $< 25\mu\text{m}$ .

Joint reliability drops rapidly when the soldermask opening is less than the ball diameter. It drops less significantly when the pad is slightly oversized. Therefore Intersil recommends the target soldermask opening to be the ball diameter plus the soldermask tolerance. For example, the 9 ball CSP package has a ball diameter of  $300\mu\text{m}$ . Therefore the soldermask opening should be  $330\mu\text{m} \pm 30\mu\text{m}$ . The copper pad size under the soldermask is not critical as long as the metal edge is always covered with soldermask.

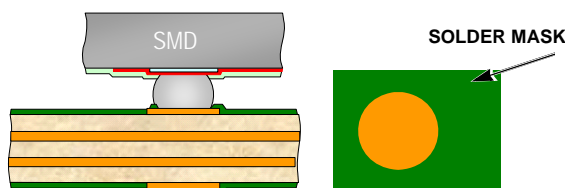


FIGURE 2. TYPICAL SMD PAD

**PCB Mounting Pad Finish**

Two types of PCB pad finishes are compatible with lead-free solder, namely, Organic Solderability Preservative (OSP) on Copper, or, E-less Nickel-Gold (Electro less Ni, Immersion Gold) on Copper. Gold thickness (when used) should be less than 0.5 micron to avoid solder joint embrittlement.

**Placement Location Target**

When needed, global and local placement targets (also called fiducials) for pick-and-place equipment with pattern recognition systems should be included in the PCB layout.

The PCB assembly contractor should be consulted for exact fiducial requirements.

### Solder Paste Stencil Design

A stencil thickness of 0.100mm to 0.125mm. is recommended. Stainless steel stencils manufactured by laser cutting followed by electro-polishing should be used. Additive build-up stencils are also acceptable. Chemical etch fabrication methods for stencil manufacturing is not recommended.

Stencil opening/aperture should have 5° taper wall (trapezoidal profile), and square opening. The side dimension of the square stencil opening should be 25 to 50 microns larger than mounting pad diameter. (e.g. for a mounting pad of 300 micron diameter, the stencil opening should be between 325x325 to 350x350 microns). If problems are seen related to solder bridging, optimizing stencil opening size would be needed. Square openings are desirable as they allow best solder paste deposition for a given aperture area.

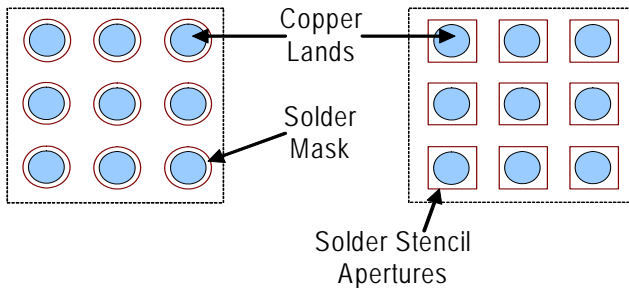


FIGURE 3. MOUNTING PADS WITH SOLDER MASK (LEFT) AND, SOLDER PASTE STENCIL OPENINGS (SQUARE OPENINGS PREFERRED) (RIGHT)

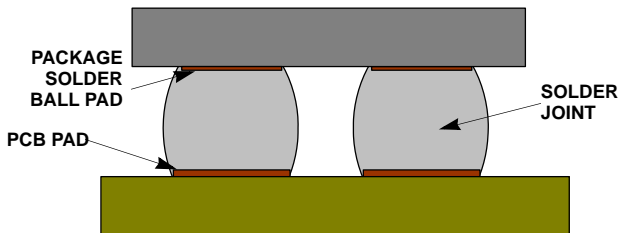


FIGURE 4. TYPICAL SOLDER JOINT CROSS-SECTION

During data conversion from design software to PCB fabrication artwork, the critical dimensions are often changed due to rounding considerations and other conversion effects. Critical dimensions need to be verified on the artwork before PCB manufacturing is begun.

Some of the critical dimensions are:

1. Mounting pad to mounting pad pitch
2. Mounting pad diameter
3. Solder mask opening
4. Alignment of solder mask opening with mounting pad

### Reflow Temperatures

The Shell-Op package is composed of unique materials that are not typically seen in semiconductor products. As a result, it has a maximum limit on reflow peak temperature of +250°C. Normal JEDEC J-STD-020D soldering profiles allow higher temperatures up to +260°C, so care must be taken when setting the reflow profiles for this package.

### Rework

It is strongly recommended not to reuse a part that has been previously mounted on PCB. Shear and tensile forces during removal of the package from the board can damage the solder balls and compromise the solder volume left on the package after removal. For this reason, solder balls on the Shell-Op 3D package should not be considered for reuse after the first mounting. A fresh unit should be used after removing the problem unit from the PCB. Fresh units can be mounted on a rework station with a reflow profile that closely matches the production reflow profile used.

### Typical Assembly Procedure

The assembly procedure for Shell-Op packages is compatible with industry standard surface mount procedures (e.g. IPC-CM+770C and IPC-A610A class 2 standards). ESD protection should be used in all process steps.

NOTE: The size and scale of this package requires the use of automated “pick-and-place” equipment and handling. Use of chip-shooters or manual mounting is not recommended.

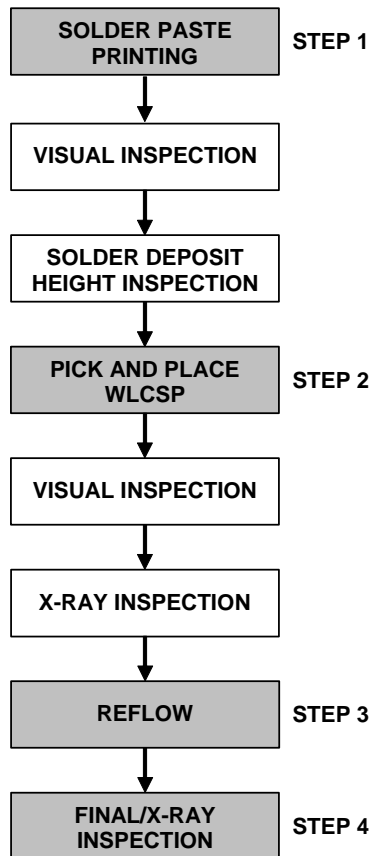


FIGURE 5. PCB MOUNTING PROCESS FLOW

**Step 1:** Screen printing - This is performed using a stencil that uses optical alignment capability and a squeegee. The squeegee speed and pressure should be adjusted per solder paste manufacturer's recommendation. The PCB should be supported during the printing process to prevent bending or flexing.

**Step 2:** Pick-and-Place - PCB should be transferred to Pick-and-Place machine within time limit specified by solder paste manufacturer. Place the device on the PCB using optical alignment. Verify device polarity (pin 1 alignment) using package orientation indicator (pin 1 mark/shape). Ensure minimal force is used during placement of the Shell-Op package to avoid physical damage (Max load 0.5kg for 0.5s). It is recommended that solder balls be dipped into solder paste deposit on the PCB to greater than 20% of the solder paste height.

**Step 3:** Solder Reflow and Cleaning - Intersil Shell-Op packages are compatible with industry standard cleaning processes. Thermal shock, such as spraying cold parts with high-temperature cleaning agents (or vice-versa) should be avoided.

**Step 4:** Final/X-Ray Inspection - Verify assembly quality using inspection techniques such as X-ray inspection. There should be no shorts, no solder bridging between solder bumps, and no loose solder balls under or around the device.

Precaution regarding solder reflow profile - DO NOT use the Tin-Lead reflow profile with this device. The solder balls are lead-free in composition and not compatible with the reflow process used for Tin-Lead solder paste.

Intersil Corporation reserves the right to make changes in circuit design, software and/or specifications at any time without notice. Accordingly, the reader is cautioned to verify that the Application Note or Technical Brief is current before proceeding.