

Introduction

Optical Dual-in-line Flat No-lead (ODFN) packages are developed for light sensor applications. The mechanical structure of an ODFN is similar to that of a conventional DFN, except the molding compound is a transparent material rather than the conventional black compound. Transparent molding compound allows transmission of ambient light through the molding compound to reach the die sensor area. The ODFN is a Pb-free package, with 100% matte tin finish on the terminals. Figure 1 shows top and bottom views of a 6 Ld ODFN for illustration. As seen from the left image, the package finish is translucent to allow light transmission and at the same time, protects the device itself.

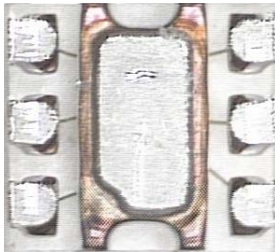
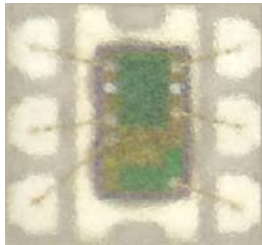


FIGURE 1. 6 LD ODFN TOP VIEW (TOP) AND BOTTOM VIEW (BOTTOM)

ODFN follows conventional DFN/QFN guidelines for PCB land pattern design and surface mount processing. Some additional recommendations are made to accommodate the special clear mold compound, which has a lower glass transition temperature (T_g) and a higher coefficient of thermal expansion (CTE) compared to conventional molding compounds. Currently, ODFN packages are available in 5, 6 and 8 pin counts. This document provides guidelines for assembly and handling of ODFN during board mount process.

Part I - ODFN Board Mounting Guidelines

The ODFN package board mounting process is similar to that of conventional DFN/QFN packages. However, there are some key differences in the materials used, therefore it is recommended that users account for these differences in their application method. Specific guidelines in this document are meant to accommodate for the differences in the materials.

Product Packing

ODFN products have been qualified under JEDEC MSL-4 test criteria, and are shipped in either a tube or tape-and-reel format. The packing quantity varies depending on the application and purchasing options. The moisture protection seal should not be broken until the board mounting process is ready. If the seal is broken, please follow standard instructions for proper baking per Jedec standard criteria for MSL-4 products.

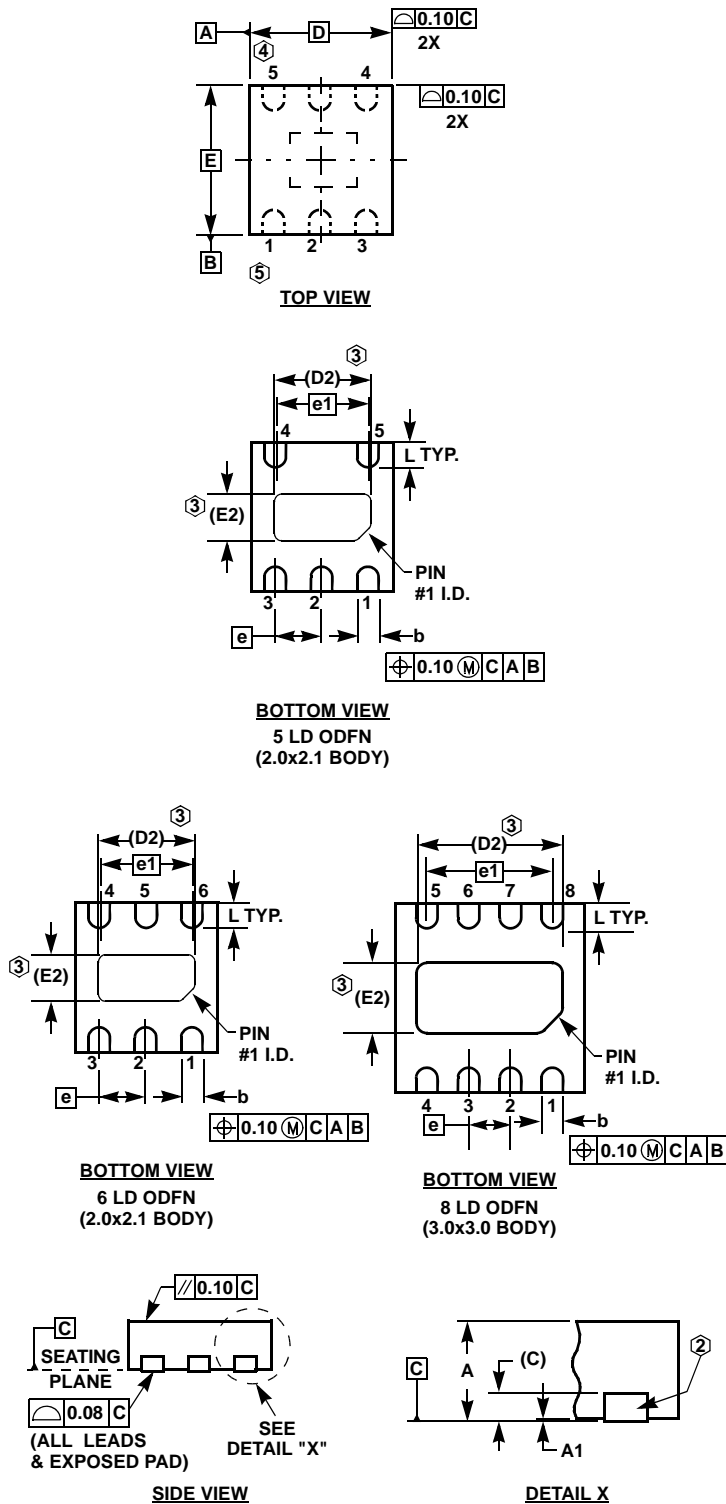
Package Construction

ODFN packages have a 0.65mm lead pitch and a nominal package thickness of 0.7mm. Figure 2 shows a detailed package outline drawing for the ODFN family with dimensions and tolerances. It is important to design PCB land patterns that correspond to the lead dimension as explained in subsequent section.

Similar to DFN and QFN packages, the ODFN has exposed pad under the package. As shown in Figure 2, the chamfered corner of the exposed thermal pad indicates Pin 1 location for the product. Similar to DFN or QFN's the exposed pad provides robustness during the board mount process.

ODFN packages are assembled on plated Copper lead-frames, and individual units are singulated by sawing process. The occasional presence of a slight oxide layer at the sawn surface of the Copper leads is not a concern for solder joint quality. Poor wetting to this exposed side edge does not impact the solder joint quality or reliability.

Optical Dual Flat No-Lead Family (ODFN)



MDP0052

OPTICAL DUAL FLAT NO-LEAD FAMILY

SYMBOL	ODFN5	ODFN6	ODFN8	TOLERANCE	NOTE
A	0.70	0.70	0.70	±0.05	
A1	0.02	0.02	0.02	+0.03/-0.02	
b	0.30	0.30	0.30	±0.05	
c	0.20	0.20	0.20	Reference	2
D	2.00	2.00	3.00	Basic	
D2	1.35	1.35	2.29	Reference	3
E	2.10	2.10	3.00	Basic	
E2	0.65	0.65	1.40	Reference	3
e	0.65	0.65	0.65	Basic	
e1	1.30	1.30	1.95	Basic	
L	0.35	0.35	0.40	±0.05	

Rev. 4 5/06

NOTES:

1. Dimensioning and tolerancing per ASME Y14.5M-1994.
2. Exposed lead at side of package is a non-functional feature.
3. Dimension D2 and E2 define the size of the exposed pad.
4. ODFN 5 Ld version has no center lead (shown as dashed line).

FIGURE 2. 5 LD ODFN, 6 LD ODFN AND 8 LD ODFN PACKAGE DIMENSIONS AND TOLERANCES

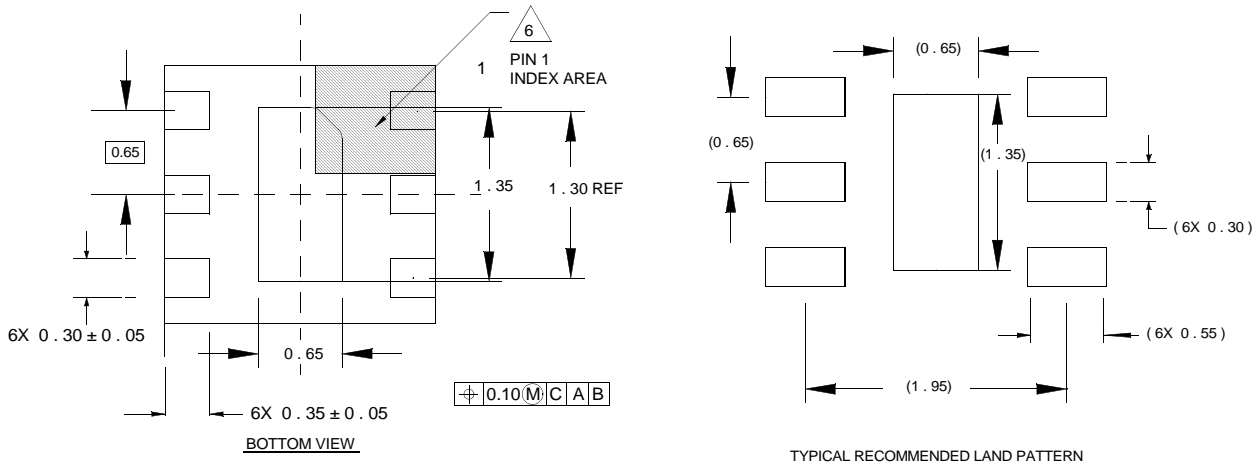


FIGURE 3. 6 LD ODFN FOOTPRINT (LEFT) AND CORRESPONDING LAND PATTERN FOR PCB DESIGN (RIGHT)

PCB Land Pattern Design

Intersil Technical Brief 389

<http://www.intersil.com/design/packages/#TechBriefs> provides detailed information for PCB design for DFN and QFN package, and this is applicable to ODFN packages. Additionally, package specific land pattern information is available on package outline drawings which are located on the following website:

<http://www.intersil.com/design/packages/>

The package outline drawings are also found in product datasheets. An example of land pattern recommendation is shown in Figure 3 (6 Ld ODFN). The main features can be summarized as follows:

- 1:1 match with exposed pad area (6 Ld ODFN case, 0.65mmx1.35mm)
- 1:1 match with pin width (6 Ld ODFN case, 0.3mm)
- Land length for pin = pin length + 0.2mm (extended from the package edge)

Figure 4 shows a reference solder joint shape after ODFN package mounting on PCB. Solder joint under the exposed pad is intended to provide the package stand-off height and robust assembly.

The pad definition on the board is recommended to be non-solder-mask-defined (NSMD), though solder-mask-defined (SMD) pads of the same effective wettable dimension are acceptable as well. A Nickel/Gold surface finish with 0.2 micron maximum gold thicknesses is recommended for good solder wettability and shelf-life for the SMT process. OSP surface finish is also acceptable, but requires appropriate controls on shelf life and exposure of PCB to environment. HASL or solder plated finishes must not be used for these products.

Solder Stencil

Either 0.100mm or 0.125mm thick stainless steel stencil is recommended.

The stencil should be laser-cut followed by electro-polish (chemical finishing is not recommended), or alternatively an additive build-up stencil may be used.

Higher reliability on board can be obtained by increased solder paste volumes. However, this needs to be controlled, as solder bridging may occur when the solder paste is excessive.

It is highly recommended to solder the exposed package pad to the corresponding landing pad on the PCB. At the same time, excessive solder under the exposed area may lead to open solder joints under the functional pins due to excess

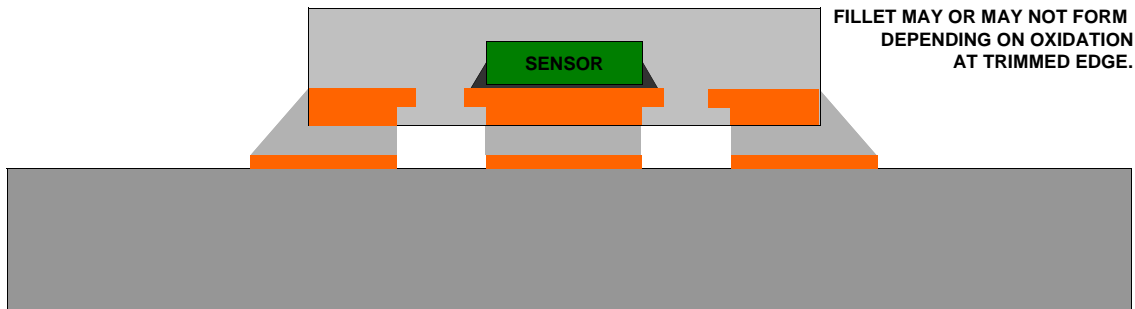


FIGURE 4. ODFN PACKAGE ON PCB WITH A REFERENCE SOLDER JOINT SHAPE

stand-off created by the central pad. Therefore, it is important to balance openings for the pins and opening for exposed pad area. The optimal value for center pad is considered to be 70% to 80% solder paste coverage under the exposed pad area.

Solder Paste

The package itself is lead-free, and is compatible with both eutectic tin/lead or lead-free tin/silver/copper solders. These packages have been qualified at a +240°C maximum temperature reflow profile for eutectic solder, and at a +260°C maximum temperature reflow profile for lead-free solder. Solder paste with "no-clean" flux and "type 3" or "type 4" solder particle size distribution is recommended.

Reflow Profile

Direct infrared (IR) heating of these packages should not be done as it can damage the part. Pure convection reflow of these parts is recommended. Typical reflow profiles per Jedec 20D criteria are recommended for the eutectic Sn/Pb and lead-free Sn/Ag/Cu solders. Peak temperature for the eutectic Sn/Pb profile is not to exceed +235°C and for the lead-free profile is not to exceed +260°C respectively. ODFN packages have been qualified up to a maximum reflow temperature of +260°C

ODFN packages have been qualified up to a maximum reflow temperature of +260°C.

Visual Inspection

Visual inspection of solder joints should be done to verify that there is no solder bridging between pads, that the solder joint is "bright-and-shiny" (lead-free appears 'dull' compared to SnPb), and that the package is not tilted or off-center with respect to the PCB land pattern. A solder fillet at the edge of package leads is not a requirement, and in fact may not form at all. Hand solder touch-up is not recommended as excessive heat from air nozzle or soldering irons can damage the transparent mold compound.

Part II - ODFN Specific Application Guidelines

The primary function of an ambient light sensor package is to allow transmission of light from the exterior to the light sensing IC, while at the same time serving as a mechanical protection to the IC. The transparency requirement does not allow conventional filler loadings as a means for controlling mechanical properties of the mold compound (such as coefficient of thermal expansion (CTE), modulus, glass transition temperature (Tg), moisture absorption, and moisture sensitivity). As such, the CTE of a clear epoxy is higher than a conventional black epoxy with fillers, has a lower modulus, and a lower Tg. The following sections outline important features of the ODFN product, that need to be understood for proper application of the product.

Sensor Location and Optics Design

In general, the package body center does not coincide with the center of the light sensor. As shown in Figure 1 (top), the sensor location (green area in top part of IC) is offset from the geometric center of the IC. The location of the sensor itself is specific to the product in question, and therefore should be obtained from the product datasheet. Figure 5 shows an example of the sensor location description. In this example, the sensor area is 0.43mmx0.61mm, and it is offset from the center of the IC by 0.25mm. The application set-up should be designed to lead the light to the center of the sensor area and not to the center of the package. On the other hand, the sensor surface height is always located at 0.28 ±0.10mm below the top surface of the package. Solder joint and package height (0.7mm) should also be considered in calculating the sensor surface height from the PCB top surface.

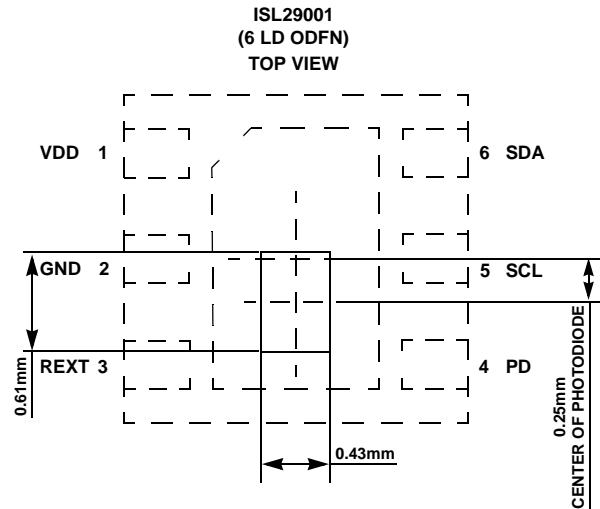


FIGURE 5. SENSOR LOCATION DESCRIPTION EXAMPLE - ISL29001 WITH 6 LD ODFN

Moisture Sensitivity and Bake Conditions

ODFN 5, 6, and 8 Lead packages have been qualified at JEDEC moisture sensitivity level 4 (MSL4) for both +240°C and +260°C solder reflow profiles. The properties of clear mold compound are such that moisture saturation occurs in less than 48 hours. Therefore, it is important to control the exposure time once the moisture protection seal is broken. See Figure 6.

It is recommended that ODFN components be baked if the exposure time exceeds 24 hours prior to board mounting. The recommended baking condition is +110°C for 4 hours. Due to a lower than typical Tg, the usual +125°C dry bake condition should not be used. A bake temperature higher than +110°C can result in discoloration of clear molding compound. See Figure 7.

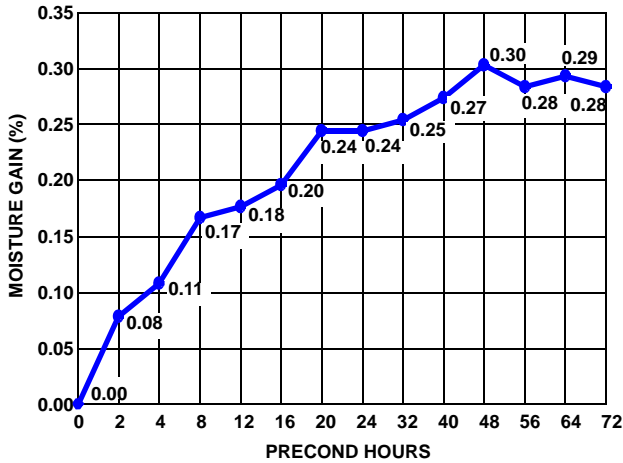


FIGURE 6. MOISTURE ABSORPTION CURVE UNDER +30°C/60% RELATIVE HUMIDITY CONDITIONS

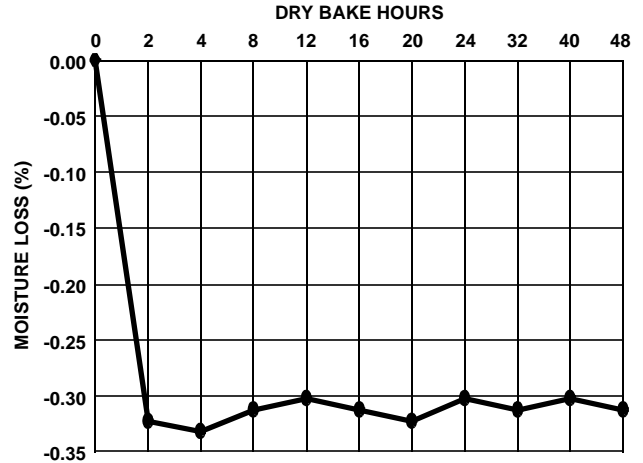


FIGURE 7. MOISTURE DESORPTION CURVE AT +110°C

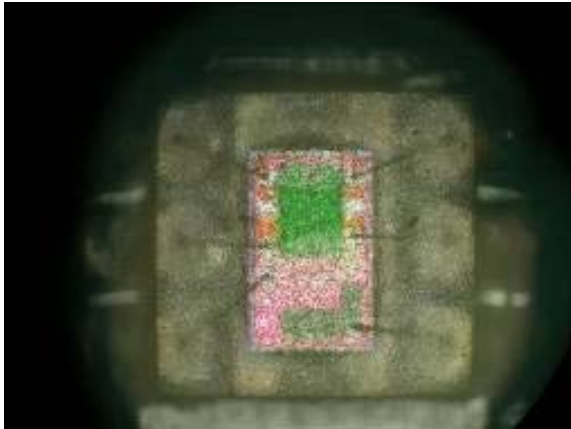


FIGURE 8. 6 LD ODFN MOUNTED WITH PREFERRED REFLOW CONDITION (LEFT), AND A PART THAT HAD EXCESSIVE TEMPERATURE EXPOSURE (RIGHT)

Pick-and-Place With Clear Package

These optically clear packages are suitable for vision-based placement machines. For machines without "auto vision recognition", it may be necessary to manually adjust the machine sensitivity to avoid recognition errors. These packages are not recommended to be placed with mechanical centering placement machines.

Rework and Associated Risks

Applications with ODFN products can be reworked using a reflow profile that closely matches the production reflow profile described in an earlier section. ODFN packages should not be exposed to >+260°C during rework operation. Do not reuse the same ODFN product upon removal from the PCB. Excessive heating of clear mold compound can result in change in color of the mold compound (as shown in Figure 8) and can also compromise wire bond integrity due to high coefficient of thermal expansion of the mold compound material.

Marking and Traceability

ODFN products do not have conventional product marking on the top of the package, since marking characters on top of the package can affect and impede light transmittance to the light sensor. However, Intersil is able to keep product traceability by marking a dot matrix on the exposed pad area under the Tin plating or using markings made on the bottom of the package (one or both methods may be used and can be tracked by Intersil). In case of a returned product, Intersil can process the part to trace the assembly information.

In case of a failure, the failed parts should be shipped to Intersil preferably without removing the parts from PCB. Parts that are still assembled on PCB during failure analysis are best for achieving good results.

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.

For information regarding Intersil Corporation and its products, see www.intersil.com