Board Level Application Notes for DFN and QFN Packages
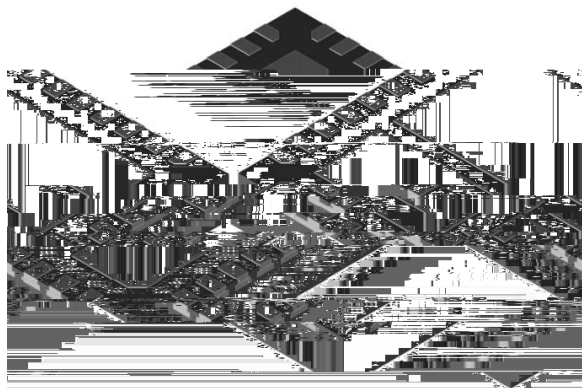
AND8211/D

INTRODUCTION

Various onsemi components are packaged in an advanced Dual or Quad Flat−Pack No−Lead package (DFN/QFN). The DFN/QFN platform represents the latest in surface mount packaging technology. It is important to follow the suggested board mounting guidelines outlined in this document. These guidelines include printed circuit board mounting pads, solder mask and stencil pattern and assembly process parameters.

DFN/QFN Package Overview

The DFN/QFN platform offers a versatility which allows either a single or multiple semiconductor devices to be connected together within a leadless package. This packaging flexibility is illustrated in Figure 1 where three devices are packaged together with a custom pad configuration in a QFN.

Figure 1. Underside of a Three−Chip 40 Pin QFN Package

Figure 2

DFN/QFN Board Mounting Process

The DFN/QFN board mounting process can be optimized by first defining and controlling the following:

1. PCB solder pad design.
2. PCB solder mask design.
3. Solderable metallization on PCB pads.
4. Solder screening onto PCB pads.
5. Choice of solder paste.
6. Package placement.
7. Reflow of the solder paste.
8. Final inspection of the solder joints.

Recommendations for each of these items are included in this application note.

Printed Circuit Board Solder Pad Design Guidelines

Refer to the case outline (specification sheet) drawing for the specific DFN/QFN package to be mounted. Based on the case outline's "nominal" package footprint dimensions, the PCB mounting pads need to be larger than the nominal package footprint (See Figure 4).

Note: On the occasion that there is not enough board space to grow the PCB mounting pads per these guidelines, the recommendation would be to come as close to these guidelines as possible.

Color Legend
For PCB Mounting Pads used in Figure 4
— Package Footprint
— PCB Mounting Pads

Figure 4. 10 Pin DFN Package Footprint Shown with PCB Mounting Pads

Printed Circuit Board Solder Mask Design Guidelines

SMD and NSMD Pad Configurations

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)

Figure 5. Comparison of NSMD vs. SMD Pads

The solder mask web (between openings) is the controlling factor in the pattern, and needs to be held to a minimum of 0.1016 mm (0.004 in). This minimum is the current PCB suppliers standard minimum web for manufacturability. Because of this web restriction, solder mask openings around PCB pads may need to be less than the recommended shown. Whenever possible, keeping to the range given will provide for the best results.

Due to ever shrinking packages with finer pitches between mounting pads, a solder mask web may not be possible. It may be necessary to have a single solder mask window opening around the package without solder mask web between mounting pads. When this occurs, care must be taken to control the solder during reflow. Where the web aided in controlling the solder, in its absence, solder may bridge between mounting pads causing shorts.

Figure 6. Typical DFN Package – PCB Mounting Pads Shown with Solder Mask Openings (NSMD)

SAME SIZE AS
PCB MOUNTING PAD

PCB CENTER MOUNTING PAD
PACKAGE OUTLINE

70-80% OF PCB
CENTER MOUNTING PAD.
BROKEN UP INTO
SMALLER OPENINGS

Color Legend
For Stencil Openings used in Figure 7 and 8.

Package Outline
PCB Center Mounting Pads
Stencil Opening

Figure 7. Typical for 0.5 mm Pitch or Greater
DFN/QFN Package with Stencil Openings Shown
Over PCB Mounting Pads