



## 1.0 - Introduction

Altera ENPIRION power converter packages are designed with a plastic leadframe package technology that utilizes copper leadframes and mold caps with System in Package (SiP) construction to form a Quad Flat No-lead (QFN) package. QFN package technology is ideal for power conversion devices due to the solderable exposed bottom copper leadframe for thermal dissipation and the resulting low package thermal resistance, the smaller package footprint and thickness, large lead size and pitch, and excellent lead co-planarity. The Package Outline Drawings of each product with the overall package dimensions and details of the solderable pads can be found in their respective datasheets.

In order to ensure high performance and reliable solder connection of ENPIRION QFN packages to the PCB, this document provides general recommendations and guidelines for PCB design and solder assembly.

## 2.0 - PCB Design Recommendations

The ENPIRION power converter package requires a high quality solder connection to a PCB for optimum electrical and thermal package performance. Recommendations for the design and layout of each product can be found in their respective datasheets. Recommendations in this document assume that design and layout guidelines in the device's datasheet are followed.

### 2.1 - Pad Layout

Figure 1 and Figure 2 shows a typical ENPIRION QFN package and the recommended PCB footprint. The dashed line in Figure 2 represents the outline of the ENPIRION package body after solder attach to the PCB. See the ENPIRION Datasheet for names and interconnection of the package leads.

**Recommendation:** The PCB layout should NOT have any traces or other top layer metal routed under the ENPIRION package body outline. All traces should extend outside the ENPIRION package body.

**Recommendation:** The PCB layout should include a top layer solder mask that fully covers the non-solderable (keep-out) areas under the ENPIRION package body outline.

The ENPIRION QFN package design uses a non-pullback lead design where the leads extend completely to the edges of the package and are exposed on the sides of the package (see the typical Package Outline Drawing in Figure 1). This non-pullback lead design allows for filleting of the solder on each lead resulting in a robust and reliable solder joint.

**Recommendation:** The PCB layout should provide pad lengths that extend beyond the package body by at least 0.20mm (8 mil) to provide solder filleting on the leads for improved solder joint reliability, inspection, and lead probing and to allow for better package placement tolerances.

## 2.2 - Pad Design

### 2.2.1 Solder Mask Pad Definition

In PCB design, the surface mount solder pads can be defined as either Solder Mask Defined (SMD) or Non-Solder Mask Defined (NSMD). The difference between these two solder mask pad definitions is in the proximity of the solder mask to the metal pad. In SMD pads the solder mask opening is smaller than the metal pad and overlaps the metal on all sides. The solder mask opening defines the solderable area of the pad. In NSMD pads the solder mask opening is larger than the metal pad and does not overlap the metal. The metal edge defines the solderable area of the pad. Figure 3 shows the details of SMD and NSMD pads.

Since the metal etching process in PCB manufacture has significantly tighter alignment and etch tolerances than the alignment registration of the solder masking process, which is typically  $\pm 0.075$  mm (3 mil), a more accurate solder pad land pattern can be obtained with NSMD pads. Likewise, with SMD pads, the solder mask that overlaps the metal pad introduces additional height above the metal surface that may affect solder joint adhesion and reliability.

Recommendation: The PCB layout should use NSMD pad definitions for all leads and pads for ENPIRION QFN packages. The solder mask opening must be defined at least 0.075 mm (3 mil) larger than the metal pad on all sides.

### 2.2.2 Solder Pad Land Pattern

The ENPIRION QFN package typically has symmetric solder pads around the perimeter of the package for signals and larger internal solder pad(s) for GND. (See the typical ENPIRION Package Outline Drawing in Figure 1 or in the device datasheet).

Recommendation: The PCB layout should conform exactly to the power converter QFN package pads exemplified in Figure 2. The 0.20 mm (8 mil) extended pad length defined in Section 2.1 should be applied. This will provide the highest solder joint reliability between the ENPIRION QFN package and the PCB.

Design variation in the PCB pad layout is allowed for any optional probe pads adjacent to signal pads that extend outside the ENPIRION QFN package footprint.

Recommendation: The PCB layout may include probe pads that should be designed with a small trace from the signal pad connecting the probe pads as shown in Figure 4. This trace must be at least 0.20 mm (8 mil) long and covered with solder mask to prevent solder from flowing onto the probe pad.

### 2.2.3 GND Pad Thermal Vias

Some ENPIRION QFN packages have a large GND pad that is used to allow heat from the ENPIRION package to dissipate into the PCB. An array of plated through-hole thermal vias should be located in the GND pad of the PCB to provide a thermal connection from the PCB GND pad to additional metal layers of the PCB. The recommended thermal via quantity, layout, and diameter are found in the Layout Recommendation section of each datasheet.

Recommendation: The PCB layout should include plated through-hole thermal vias for efficient heat dissipation from the ENPIRION QFN package into the PCB. One of the following thermal via types should be used:

- Open plated through-hole vias that will provide lower PCB fabrication costs but may fill with solder during ENPIRION package reflow,
- Plugged and capped plated through-hole vias that will provide higher PCB fabrication costs but will not fill with solder during ENPIRION package reflow.

See Section 3.1.1 for recommendations for soldering with either open or plugged and capped plated through-hole vias in the PCB GND pad.

### 2.3 - Pad Plating

A uniform metal plating thickness on the PCB will ensure reliable, high ENPIRION QFN package solder assembly yield.

Recommendation: The following plating guidelines for various PCB metal plating processes should be used:

- For an Electroless, Nickel-Immersion, Gold (Au) finish (ENIG), to prevent solder joint embrittlement, the thickness of the Au must be between 0.05  $\mu\text{m}$  to 0.20  $\mu\text{m}$ .
- A PCB with an Organic Solderability Preservative (OSP) coating is recommended as an alternative to an ENIG plating
- For a PCB with a Hot Air Solder Leveling (HASL) finish, the surface finish should be maintained within a 28 $\mu\text{m}$  range.

### 3.0 – Soldering and Reflow

This section discusses the soldering and reflow process in detail.

#### 3.1 – Solder Paste Deposition

A stencil-printing process will be required for deposition of solder paste to the PCB for reflow of the ENPIRION QFN package to the PCB. The stencil-printing process requires the use of an aperture based metal stencil where solder is transferred through the apertures onto the solder pads of the PCB. To minimize solder voids and ensure maximum electrical and thermal connectivity of the ENPIRION package to the PCB, large pads, solder volume, and solder leaching must be considered in the stencil design. The design and fabrication of the stencil determines the quality of the solder paste deposition onto the PCB and the resulting solder joint after reflow. The primary stencil parameters are aperture size, thickness, and fabrication method.

##### 3.1.1 Solder Stencil

Figure 5 shows the solder stencil aperture layout for the ENPIRION QFN package. Solder stencil recommendation can also be found in the device datasheet.

Recommendation: The stencil aperture openings for all leads should have a 1:1 size ratio with the leads of the ENPIRION QFN package.

Recommendation: The stencil aperture opening for the large internal GND pad should have a size ratio from 0.3:1 to 0.5:1 depending on the use of open or plugged and capped through-hole plated thermal vias in the PCB GND pad.

- If open through-hole plated vias are used in the PCB GND pad then the stencil aperture opening should have a maximum size ratio of 0.5:1. Note: the quantity and diameter of the vias will determine the amount of solder that will flow into the vias during reflow reducing the solder volume between the ENPIRION GND pad and the PCB GND pad. The solder stencil aperture layout should use one large aperture opening for the GND pad with dimensions as shown in Figure 5.
- If plugged and capped through-hole vias are used in the PCB GND pad then the stencil aperture opening should have a size ratio near 0.3:1. Note: less solder paste will be required since the vias will not fill with solder during reflow. The solder stencil aperture layout should use smaller aperture openings (known as a window pane solder stencil aperture) for the GND pad with dimensions as shown in Figure 5. Solder stencil recommendation can also be found in the device datasheet.

Recommendation: The stencil should be made from stainless steel and have a thickness between 0.102 (4mil) and 0.127 $\mu$ m (5mil).

Recommendation: The stencil should be fabricated by chemical etching with electropolishing or by laser cutting. A tapered wall (up to 5°) on the apertures will facilitate solder paste release when the stencil is lifted from the PCB.

### 3.1.2 Solder Paste

Various types and grades of solder paste can be used for surface mounting the power converter QFN package. For leaded applications, a Sn-Pb solder can be used and for leadfree application a Sn-Ag (SA) or Sn-Ag-Cu (SAC) solder can be used.

Recommendation: Any Type 3 solder paste that is either water-soluble or no clean is acceptable.

## 3.2 – Package Placement

The ENPIRION QFN package is placeable onto the PCB using industry standard component pick-and-place systems that have a placement accuracy of  $\pm 0.05$ mm ( $\pm 2$  mil).

Recommendation: For higher placement accuracy a component pick-and-place system with a top-and-bottom vision system should be used.

Recommendation: The ENPIRION QFN package should be released between 25—50 $\mu$ m (1—2 mil) into the solder paste.

## 3.3 – Solder Reflow

The ENPIRION QFN package may be surface mount soldered using standard IR or IR convection SMT reflow process. The ENPIRION QFN package is qualified for a maximum of three (3) reflow cycles at 260°C peak reflow temperature according to the IPC/JEDEC J-STD-020C standard.

Recommendation: When using a solder paste with a no-clean flux a Ni (nitrogen) purge should be used during reflow.

The thermal reflow profile for component placement onto a PCB and the actual temperature of the component is dependent on various factors including: PCB thickness; PCB Cu (copper) weight;

component density; component location; and size and thermal mass of other surrounding components. When using an IR reflow oven the location of other larger components can cause shadowing onto a specific component.

Recommendation: The thermal reflow profile for the ENPIRION QFN package should be determined for each specific location on the PCB.

### 3.3.1 Solder Reflow Profile

Typical solder reflow profiles for both leaded and leadfree solders are shown in Figures 6 and 7, respectively.

**Table 1 Recommended Solder Reflow Profile Parameters for Leaded Solder**

Leaded Solder	Reflow Parameters		
Reflow Profile Segment	Min	Recommended	Max
Preheat Temperature (°C)	100	140	150
Preheat Time (sec)	60	110	120
Ramp Up Rate (°C/sec)	1.5	2	3
Dwell Time above 183°C (sec)	50	75	85
Peak Temperature (°C)		215	240
Dwell Time at Peak (sec)	1	5	10
Ramp Down (°C/sec)	3	4	6

**Table 2 Recommended Solder Reflow Profile Parameters for Lead-Free Solder**

Lead-Free Solder	Reflow Parameters		
Reflow Profile Segment	Min	Recommended	Max
Preheat Temperature (°C)	150	175	200
Preheat Time (sec)	60	130	180
Ramp Up Rate (°C/sec)	2	3	4
Dwell Time above 217°C (sec)	60	90	120
Peak Temperature (°C)		250	265
Dwell Time at Peak (sec)	10	12	20
Ramp Down (°C/sec)	3	4	4

## 3.4 – Solder Join Inspection

An inspection of the solder joint between the solder pads of the ENPIRION QFN package and the PCB should be performed.

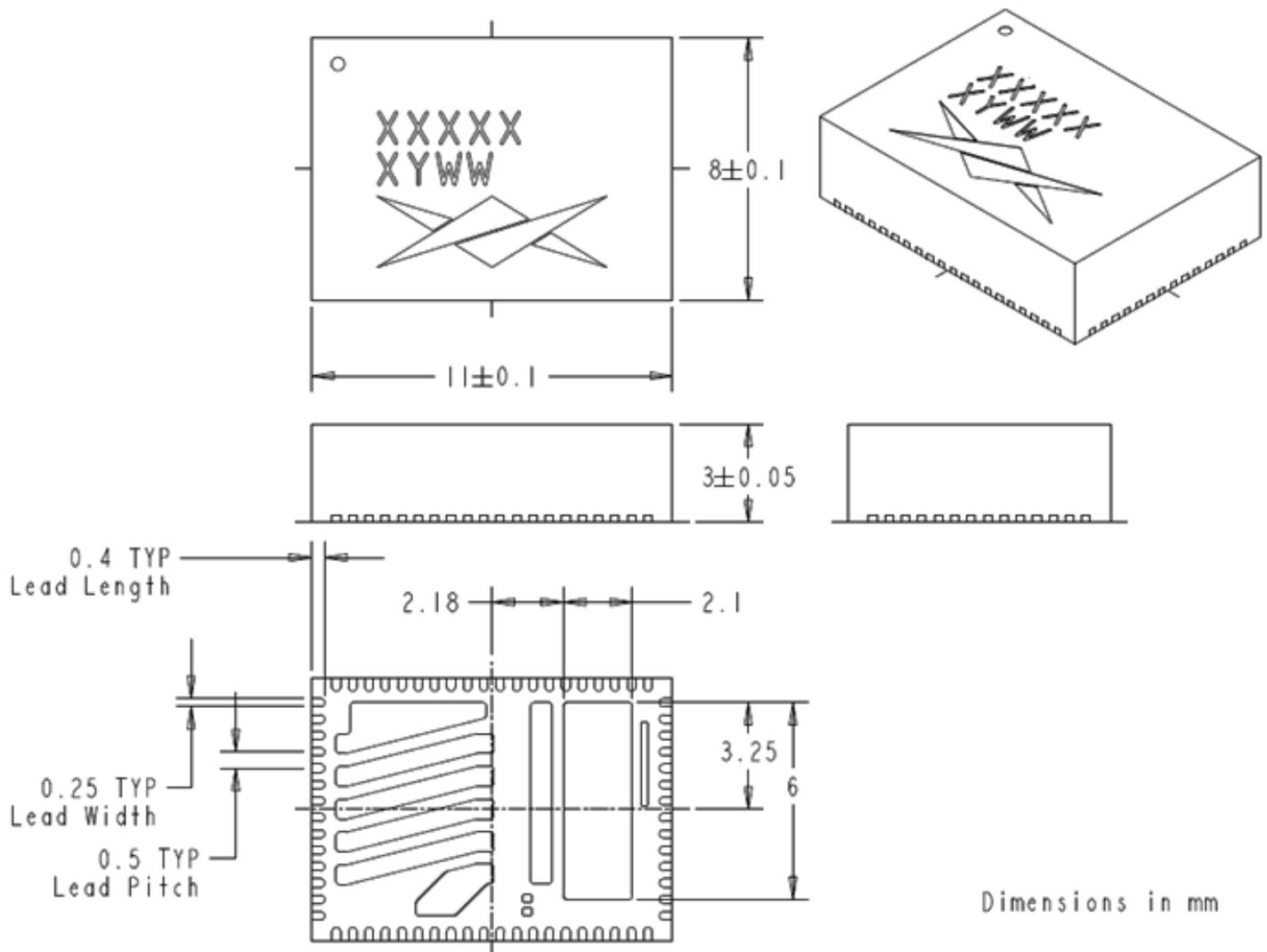
Recommendation: Sample inspection monitoring at regular intervals during solder reflow should be performed of the solder joint between the solder pads of the ENPIRION QFN package and the PCB.

The best visual inspection tool for inspection of the ENPIRION QFN package solder joint on the PCB is a transmission X-ray, which can identify defects such as solder bridging, shorts, opens, and large voids (Note: small voids in large solder joints are not detrimental to the reliability of the solder joint). An additional visual inspection tool consists of side view inspection, such as 90° mirror projection, to determine component flatness and solder joint volume.

**Recommendation:** If possible, both transmission X-ray and a side view inspection process should be performed for visual solder joint inspection.

### 4.0 – Summary

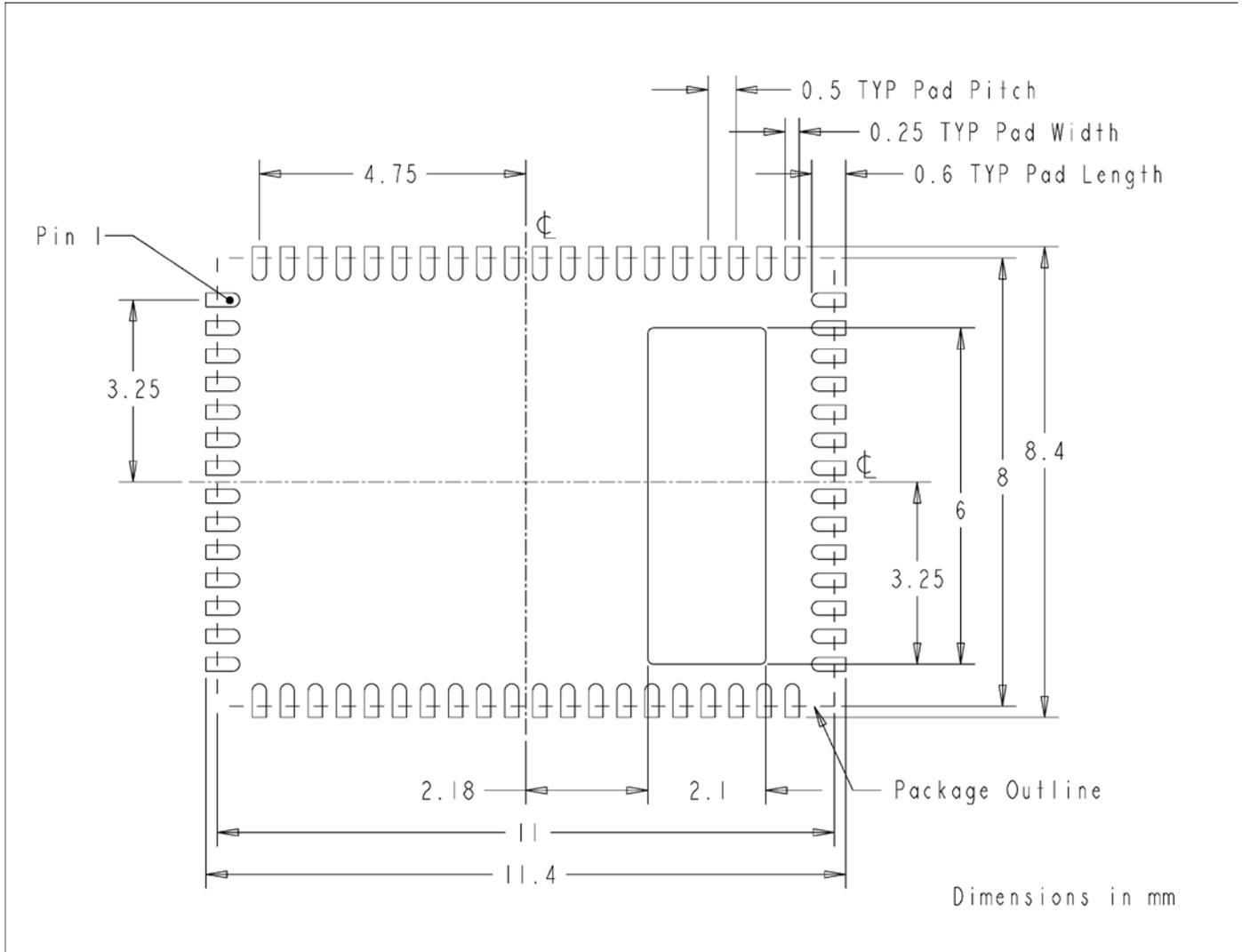
This paper presents recommendations for PCB design and process procedures for use with the Altera Enpirion QFN power converter packages. Proper and recommended design principles for PCB land pattern design and construction are presented. A detailed process description and recommendations for soldering and reflow including solder stencil printing, PCB preheating and localized heating, solder reflow, and QFN package placement are presented.



**Figure 1:** Sample ENPIRION Package Outline Drawing (Bottom View)

**Packing and Marking Information:** [www.altera.com/support/reliability/packing/rel-packing-and-marking.html](http://www.altera.com/support/reliability/packing/rel-packing-and-marking.html)

Package and Mechanical



**Figure 2:** Sample PCB Land Pattern for soldering ENPIRION QFN package (Top View)

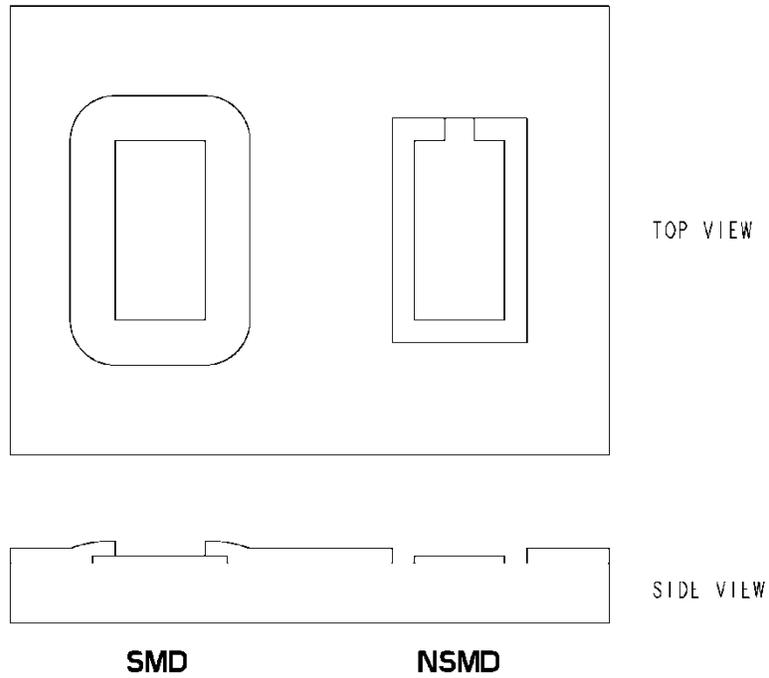


Figure 3: PCB Pad SMD and NSMD definitions

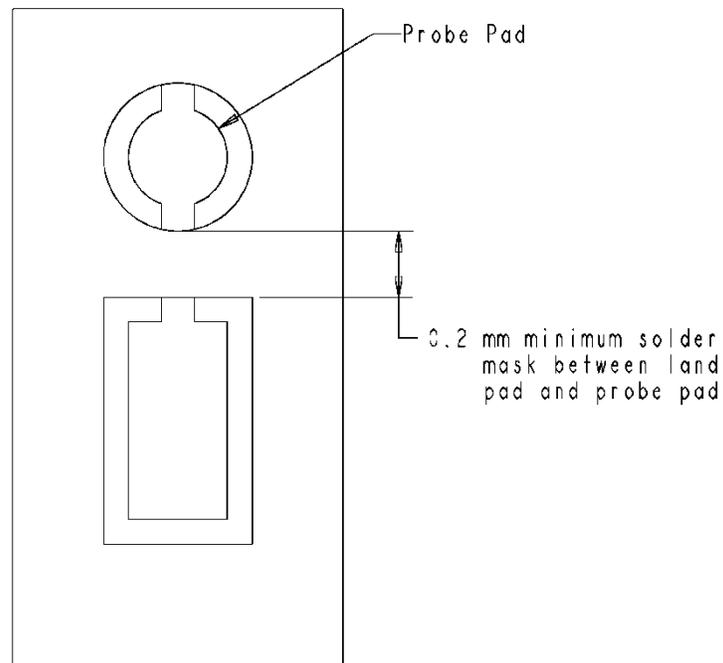


Figure 4: PCB pad design for probe pads

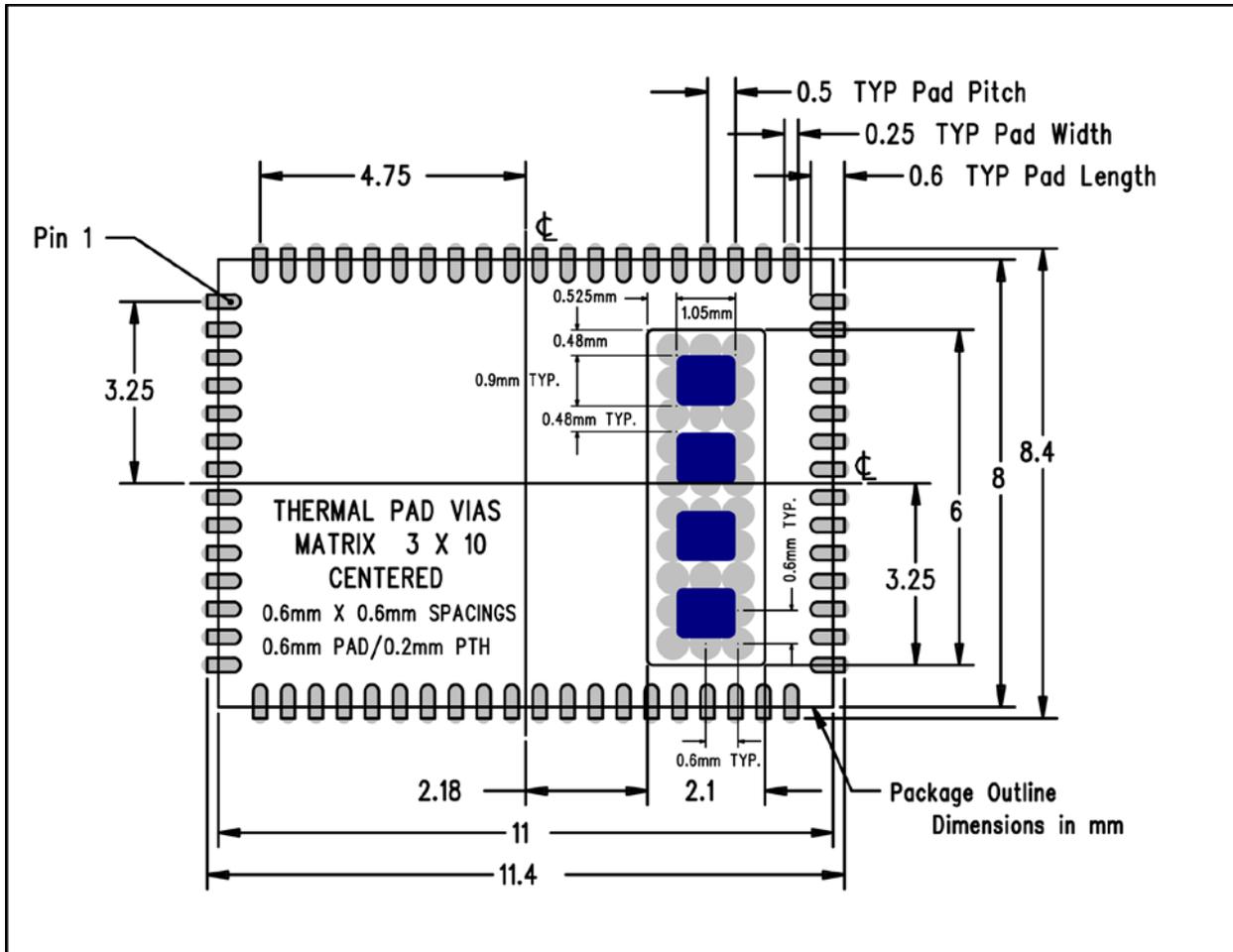


Figure 5: Sample Solder stencil aperture layout for the ENPIRION QFN package (Top View)

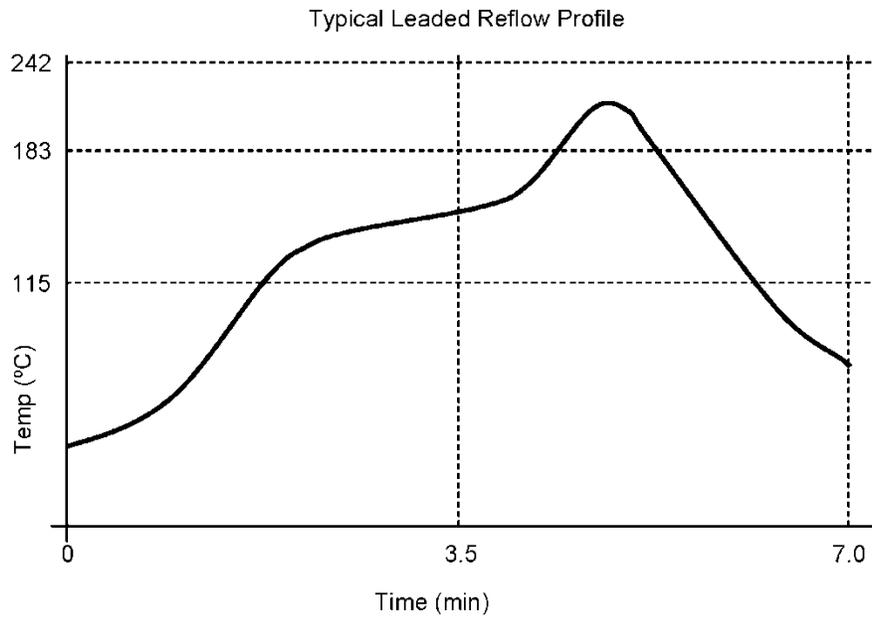


Figure 6: Typical Leaded Reflow Profile

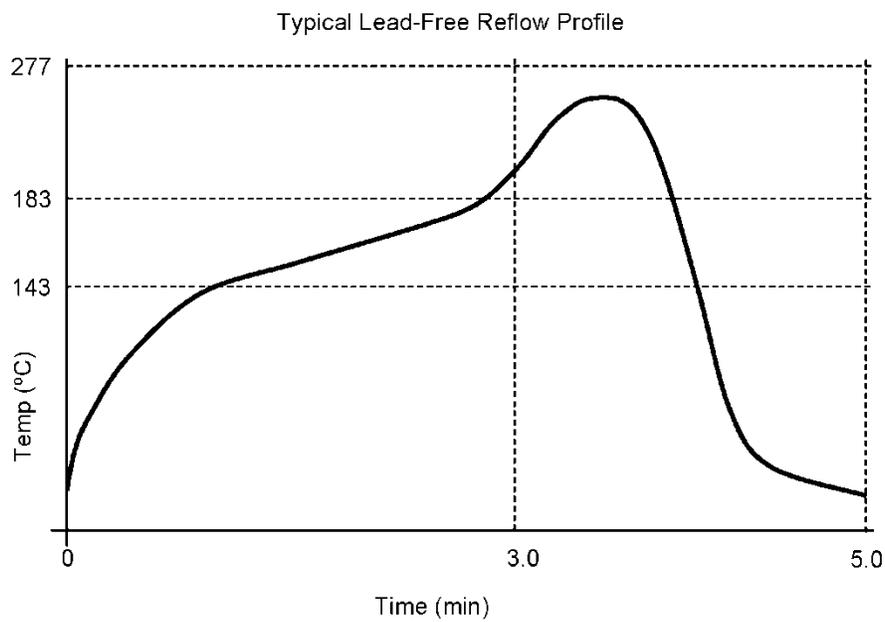


Figure 7: Typical Lead-Free Reflow Profile

### Contact Information

Altera Corporation  
101 Innovation Drive  
San Jose, CA 95134  
Phone: 408-544-7000  
[www.altera.com](http://www.altera.com)

© 2013 Altera Corporation—Confidential. All rights reserved. ALTERA, ARRIA, CYCLONE, ENPIRION, HARDCOPY, MAX, MEGACORE, NIOS, QUARTUS and STRATIX words and logos are trademarks of Altera Corporation and registered in the U.S. Patent and Trademark Office and in other countries. All other words and logos identified as trademarks or service marks are the property of their respective holders as described at [www.altera.com/common/legal.html](http://www.altera.com/common/legal.html). Altera warrants performance of its semiconductor products to current specifications in accordance with Altera's standard warranty, but reserves the right to make changes to any products and services at any time without notice. Altera assumes no responsibility or liability arising out of the application or use of any information, product, or service described herein except as expressly agreed to in writing by Altera. Altera customers are advised to obtain the latest version of device specifications before relying on any published information and before placing orders for products or services.