AND8405

Board Mounting Considerations for ICEPaK Packages

Prepared by: Phillip Celaya
ON Semiconductor

ON Semiconductor®

http://onsemi.com

APPLICATION NOTE

Introduction

Power management devices have become very critical to system applications. With the need to use FET devices to manage power in the system, dissipating heat out of the system is becoming a key challenge for system engineers. The industry is focusing on new package types that can thermally dissipate the heat away from the board for optimum solutions. The ICEPak package is ideal for this, since heat dissipation can be done in both directions: into the board and out the top using a heatsink approach.

Package Overview

ICEPak packages are very similar to CSP type packages. The package is light weight and small and has a bumped die. Figure 1 shows both, a top and bottom view of the ICEPak package. Figure 2 shows a cross-section of an ICEPak Package. The package is thinner than most conventional power packages, but with better electrical and thermal performance.

Figure 1. Top and Bottom View of ICEPak Package

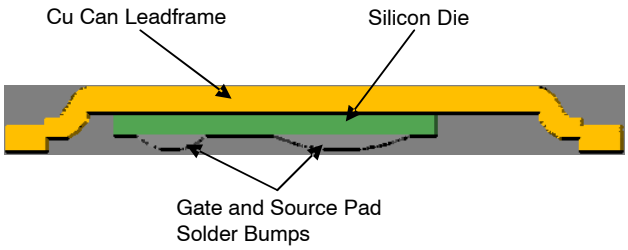

Figure 2. Cross Section of ICEPak

Printed Circuit Board (PCB) Design

SMD and NSMD Pad Configurations

The Solder Masked Defined (SMD) and Non-Solder masked Defined (NSMD) pad configurations are commonly used for surface mount packages.

With SMD configured pads, the solder mask covers the outside perimeter of the rectangular contact pads. With this configuration, the solder flows over the top surface of the contact pad, and is prevented from flowing along the sides of the pads by the solder mask.

With NSMD configured pads, there is a gap between the solder mask and the rectangular contact pad. With this configuration, the solder flows over the top surface and the sides of the contact pad. Either configuration can be used.

Recommended PCB Pad Diameters:

In order to assist in designing the PC boards for the ICEPak Packages, Figures 3, 4 and 5 show the recommended board pad dimensions for the SMD or NSMD pad configuration.

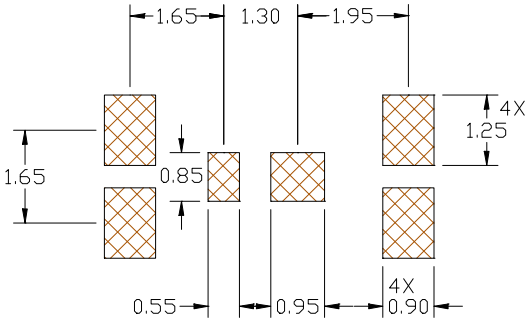

Figure 3. Recommended PCB Pad Layout for ICEPak 4.8x3.8mm-B

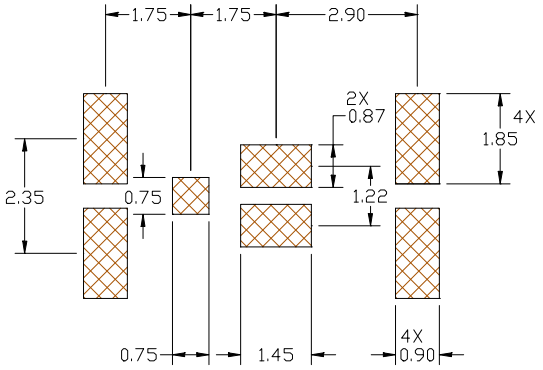

Figure 4. Recommended PCB Pad Layout for ICEPak 6.3x4.9mm-E

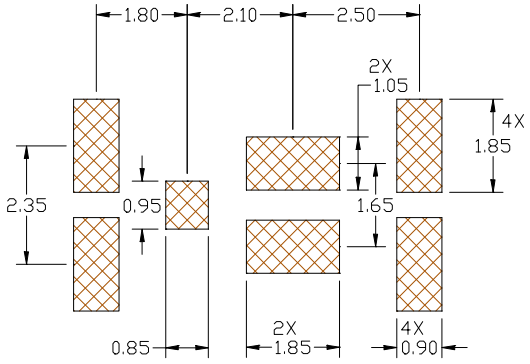

Figure 5. Recommended PCB Pad Layout for ICEPak 6.3x4.9mm-C

SMT Process Recommendations

Plating of the PCB Pads

There are several plated solderable metallizations which are used for PCB surfaces mount devices. In either case, it is imperative that the plating is uniform, conforming, and free of impurities to ensure a consistent solderable system.

The common plating selection consists of plating electroless nickel over the copper pad, followed by immersion gold. The allowable stresses and the temperature excursions the board will be subjected to throughout its lifetime will determine the thickness of the electroless nickel layer. Gold thickness is recommended to be 0.15 μm +/- 0.05 μm. Having excessive gold in the solder joint can create gold embrittlement, which may affect the reliability of the joint.

Solder Stencil Design

The solder is typically patterned onto the PCB by using a 127 μm to 152 μm (0.005 in to 0.006 in) thick stencil screen. The stencil type should be laser-cut stainless steel and electropolished.

The stencil aperture opening sizes should be square shaped openings. This screen is designed and manufactured to only allow a specific amount of solder to be placed on the bond pads. It is recommended that the side walls of the screen openings be tapered approximately 5 degrees to facilitate the release of the paste when the screen is removed from the PCB. Figures 6 - 8 show recommended stencil designs for all three part types.

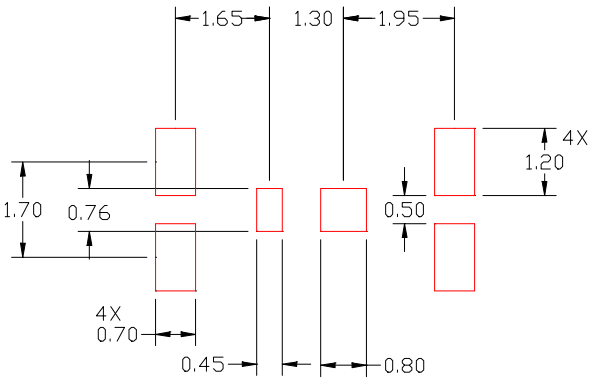

Figure 6. Stencil Design for ICEPak 4.8x3.8-B

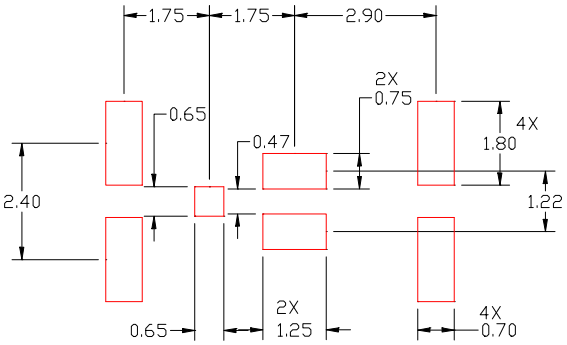

Figure 7. Stencil Design for ICEPak 6.3x4.9-E

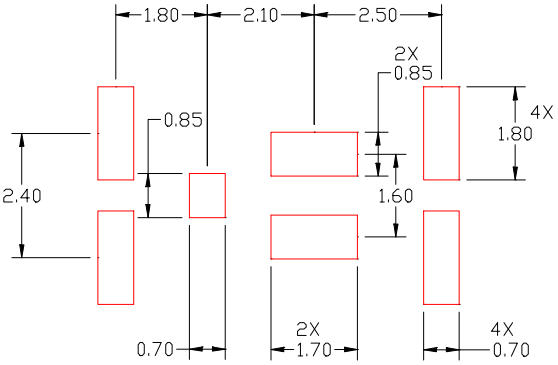

Figure 8. Stencil Design for ICEPak 6.3x4.9-C

Solder Type

Either type 3 or 4 solder paste is recommended. Solder past must be a no-clean type paste. ICEPaks once mounted to the PCB board, will have little to no stand-off height between the package and the PCB board. Therefore, cleaning under the package is not an option.

Package Placement

On Semiconductor recommends that an adequate force be used to depress the package into the solder during pick and place operation. A depression of 0.050-0.075 mm is recommended. This is due to the fact that the silicon die is recessed into the package from the Leadframe seating plane. Forces of 150-200 grams were used for our board evaluation with good results.

Solder Reflow

Once the package is placed on the PC board along with the solder paste, a standard surface mount reflow process can be used to mount the part. Forced Convection reflow in nitrogen is recommended.

An example of a standard Pb-free profile is shown in Figure 9. The exact recommended reflow profile will be determined, and is available, by the manufacture of the paste since the chemistry and viscosity of the flux matrix will vary. These variations will require small changes in the profile in order to achieve an optimized process.

In general, the peak temperature of the profile should be between 210-225°C for Eutectic Solder profiles and 250-260°C for Pb-Free solder profiles.

Solder Joint Inspection

The inspection of the solder joints is commonly performed with the use of and X-ray inspection system. With this tool, one can locate defects such as shorts between pads, open contacts, voids within the solder as well as any extraneous solder.

Rework Process

It is important to minimize the chance of overheating neighboring devices during the removal of the package since these devices are typically in close proximity to other packages, Standard SMT rework systems designed for BGA packages are recommended for this procedure since the airflow and temperature gradients can be carefully controlled. This procedure entails that the PCB board be heated normally at 100°C prior to the heating apparatus being applied over the device. Nitrogen Atmosphere is typically used to control oxidizing. It is also recommended that the PCB board be placed in an oven at 125°C for 12 hours prior to heating the parts to remove excess moisture from the packages.

Note: ON Semiconductor does not recommend that devices that have been removed be reused. We recommend that a new device be used in place of the old one and the old device is disposed of.

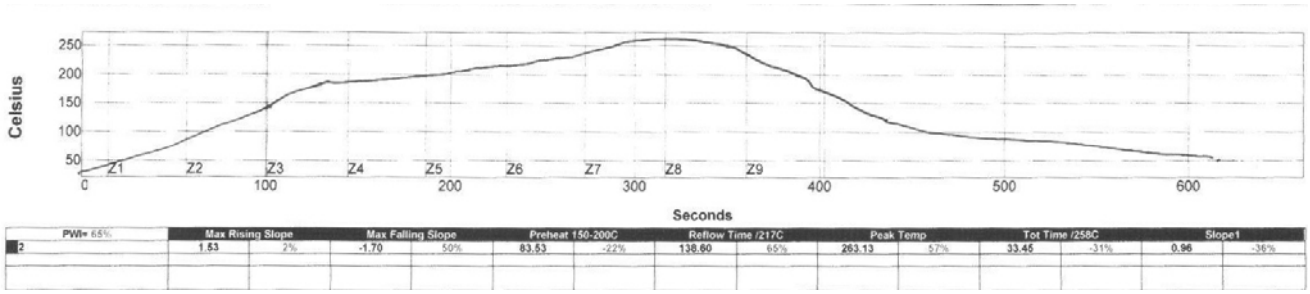

Figure 9. Reflow Profile for Pb-Free Solder

ICePAK is a trademark of Semiconductor Components Industries, LLC (SCILLC).

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
P.O. Box 5163, Denver, Colorado 80217 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81−3−5773−3850

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative

AND8405/D