

Application Note AN-1079

Board Mounting 0.5A FlipKY™

By
 Hazel Schofield and Philip Adamson, International Rectifier

For part numbers
 IR0520CSPTR/PBF, IR0530CSPTR/PBF, IR05H40CSPTR/PBF

Table of Contents

	Page
Introduction	2
Device Construction	2
Device Outline.....	2
Packaging	3
Design Considerations	3
Substrates.....	3
Substrate Design.....	4
Assembly Consideration	4
Solder Pastes.....	4
Screen Design.....	4
Pick and Place Nozzle Considerations.....	5
Device Placement.....	5
Blow Off Settings.....	6
Reflow and Cleaning	6
Underfill	6
Rework Guidelines	7
To Replace a WLP Device:	7
Handling After Board Mount	7

The 0.5A FlipKY™ package is a Schottky diode device delivered in the form of a wafer level package (WLP). This package has both the anode and the cathode on the same side of the die. The leadless nature of WLPs can offer significant advantages with regard to PCB area and thermal performance. The 1A range of FlipKY product is covered in AN-1011. A Pb-free version of the 0.5A FlipKY is also available in view of the legislation regarding the elimination of lead from electronic products. This document is designed to help the customer prepare for, and assemble 0.5A FlipKY in an effective manner.

Introduction

The 0.5A FlipKY™ package is a Schottky diode device delivered in the form of a wafer level package (WLP). This package has both the anode and the cathode on the same side of the die. The leadless nature of WLPs can offer significant advantages with regard to PCB area and thermal performance. The 1A range of FlipKY product is covered in AN-1011. A Pb-free version of the 0.5A FlipKY is also available in view of the legislation regarding the elimination of lead from electronic products. This document is designed to help the customer prepare for, and assemble 0.5A FlipKY in an effective manner.

Device Construction

The modified 0.5A FlipKY die has anode and cathode solder bumps on the front face of the die. The manufacturing process used to create the bumps is well known and widely documented throughout the industry. Figure 1 shows a typical 3-bump WLP device.

Figure 1 shows how 0.5A FlipKY WLP devices are constructed. The wafer is coated with silicon nitride passivation to protect it from the external environment. Only the areas that define the solder bump positions are left unpassivated. The under bump metallurgy (UBM) is deposited here to prevent the wafer metallization from interacting with the solder and to seal the passivation, which prevents the ingress of moisture around the contacts.

The UBM is approximately 5um electroless plated nickel, finished with a thin layer (0.1um) of immersion gold.

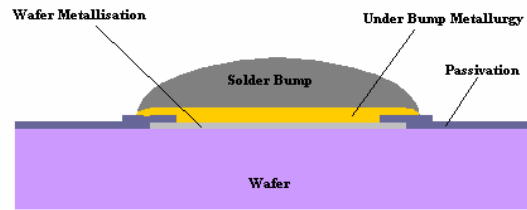


Figure 1
Construction of a 0.5A FlipKY (not to scale)

Device Outline

The nominal dimensions of the 0.5A FlipKY die are 1.244mm x 0.914mm. The 0.5A FlipKY has 3 solder bumps: 2 cathode pads (0.200x0.300mm) and 1 anode pad (0.300x0.400mm). Figure 2 shows the outline drawing of the 0.5A FlipKY. Figure 3 shows a side view of the 0.5A FlipKY. It can be seen that the device is between 0.460-0.520mm in height. This is made up of a wafer thickness of 0.355-0.395mm and a bump height of 0.105-0.125mm.

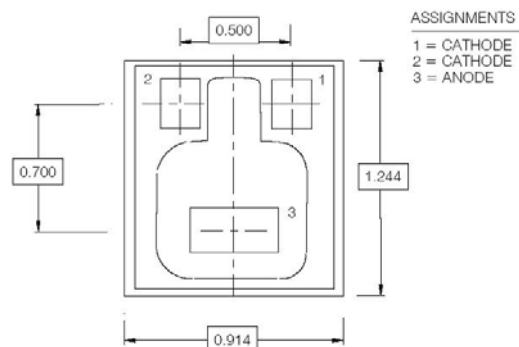


Figure 2 *Outline drawing of 0.5A FlipKY*

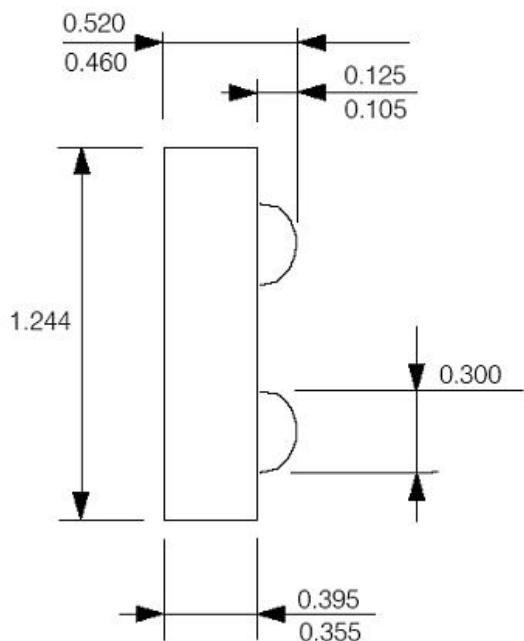


Figure 3 Side view of 0.5A FlipKY

0.5A FlipKY are laser marked on the top of the device for convenience and traceability. The laser mark consists of a “Ball 1” mark that represents the end of the device with the cathode solder bumps. This allows the user to easily check if the orientation of the assembled device is correct. Users will also find a part number and a lot number mark, which allows traceability.

Packaging

0.5A FlipKY are supplied in tape and reel format. The position of Ball 1 is standard for all 0.5A FlipKY devices (shown in Figure 4). The Ball 1 mark corresponds to one of the two cathode bumps.

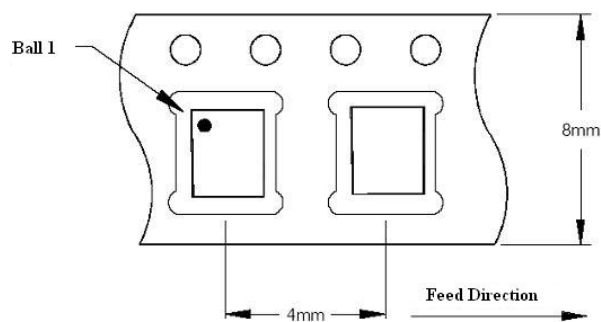


Figure 4

Diagram showing 0.5A FlipKY in tape and reel.

WLP devices can be used from the tape without intermediate operations. They are 100% inspected for solder defects and are tested at all rated values.

As no moisture-absorbing materials are used in WLP construction, no special storage conditions are required to prevent popcorning during reflow. This can be considered equivalent to JEDEC Moisture Sensitivity Level 1 (MSL1), although this specification and test method does not apply to this type of package.

Design Considerations

The recommendations in this section are based on International Rectifier’s investigations, combined with research into standard practices within the industry. However, they may need to be adjusted to suit specific production environments.

A key factor in the reliability of bumped components is the consistency of the solder joints. For maximum reliability, all joints must be of the same shape and size. This is primarily determined by the substrate design but is also influenced by the manufacturing and assembly processes used.

Substrates

International Rectifier developed WLP technology for use with organic substrates such as epoxy glass-woven and flexible laminates, but this does not preclude its use with ceramic substrates and other materials. Reliability testing and analysis is

carried out on substrates finished with electroless nickel immersion gold because these are easy to store and solder, but many other surface finishes are available and most are suitable for use with WLP devices. The exception is hot-air-levelled solder, which is not recommended because it cannot coat pads of the required diameter with a consistent volume of solder.

Substrate Design

There are many factors that should be taken into account when designing a substrate for WLP devices. Key factors in most circuits are reliability and thermal performance. WLP devices have inherently short heat paths to the substrate so their thermal performance can be greatly enhanced by effective substrate design. Both reliability and thermal performance can be affected by the choice between solder-mask-defined and pad-defined (non-solder-mask-defined) layouts. Figure 5 shows the two styles.

Some industry reports recommend pad-defined layouts for solder bumped packages because such designs can enhance thermal cycling capability. However, there is a trade-off in thermal performance compared with the larger areas of copper possible with solder-mask-defined layouts, which conduct heat away from the devices more efficiently. International Rectifier conducts all its reliability qualifications using solder mask defined layouts to ensure that its components perform reliably with them, and is therefore happy to recommend this style to its customers.

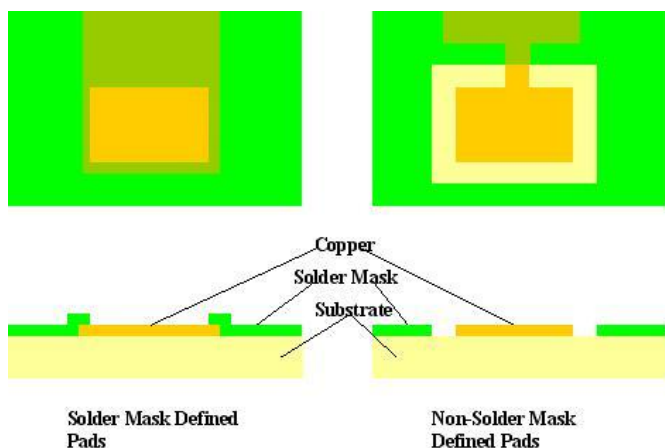


Figure 5 Diagram showing solder mask defined (SMD) and non-solder mask defined (NSMD) pads.

0.5A FlipKY, as for all WLPs, are susceptible to mechanical damage if the assembled component is subjected to mechanical contact. This often results in the functionality and reliability of the component being adversely affected. It is therefore recommended that PCB designs do not situate the devices in areas where they are likely to be subjected to mechanical contact during later stages of assembly.

It is recommended that PCB pads are slightly larger than the solder bumps on the 0.5A FlipKY die. Figure 6 shows the recommended dimensions for solder mask defined PCB pads.

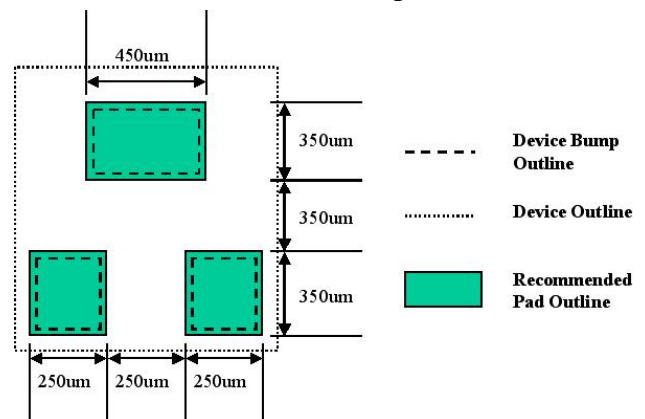


Figure 6 Diagram showing recommended solder mask defined PCB pad dimensions (drawing not to scale).

Assembly Considerations

International Rectifier’s WLP devices are designed to be assembled using existing processes and standard reflow profiles. However, procedures and conditions can have a profound influence on assembly quality. This section outlines practices that have given good results during evaluations.

Solder Pastes

International Rectifier recommends low residue, no clean solder pastes but the best choice of paste

depends on surface finish. For example, finishes such as organic solderability preservatives require a more active flux than gold. The solder alloy should be the same as the solder bumps (Sn63Pb37 for eutectic part numbers and SnAg3-4Cu0.5-0.7 for Pb-free part numbers) and of type 3 or finer.

As with all fine pitch components, great care must be taken to avoid the formation of extraneous solder balls. These may become dislodged under the die or between contacts, causing short circuits.

Screen Design

For internal evaluations, screens of 0.100mm (100µm) and 0.150mm (150µm) thickness have been used. Screens of these thicknesses have shown consistent solder deposition. Figure 7 shows images of suitable solder deposition and variable, unsuitable solder deposition. Assemblers should ensure that any stencils used show consistent amounts of solder being deposited onto the PCB pads.



Figure 7 Images showing a good solder print and a bad solder print

Aperture sizes of the same area to the board pads and 20% smaller than the board pads have both been evaluated. Both are suitable for assembly of 0.5A FlipKY.

These different screen thicknesses and aperture sizes seem to produce satisfactory results for solder deposition. It was observed during internal trials that using a 0.150mm stencil with reduced apertures can result in a short deposition of solder paste. It was found that using 2 sweeps of the squeegee, and reducing the speed with which board and stencil are separated after print could resolve this.

Screens can be etched, laser-cut or electroformed. Electroforming gives a very smooth sidewall that can improve solder paste release; electropolishing laser-cut screens can have a similar effect.

Tapering the apertures from the substrate to print side by 2-5° can also improve solder paste release (see Figure 8).

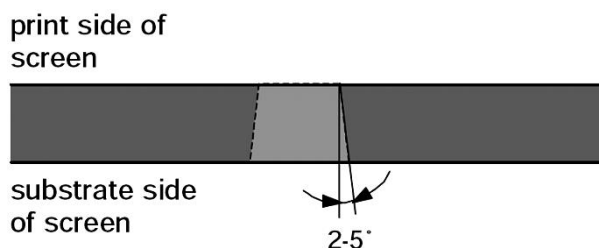


Figure 8

Diagram showing tapered screen apertures.

In summary, IR recommends that customers use a 0.100mm screen when assembling 0.5A FlipKY, however, a 0.150mm screen can be used with satisfactory results. Screen apertures are recommended to be slightly oversized compared to the device bumps. The oversized apertures should be the same dimensions as the PCB pad recommendations. However, a 20% reduction in aperture area can be used to yield satisfactory results. Customers should check that the chosen screen design delivers consistent amounts of solder deposition during the printing process.

Pick and Place Nozzle Considerations

Because of the small dimensions of the 0.5A FlipKY, choice of nozzle for the pick and place operation is critical. Internal evaluations have shown that using the wrong size/shape nozzle can cause large numbers of assembly defects.

From the trials conducted on the 0.5A FlipKY package, the recommended nozzle for the placement of 0.5A FlipKY is a 0.5mm nozzle, or equivalent (this can vary depending on equipment manufacturer).

Device Placement

0.5A FlipKY must be placed in the printed solder during pick and place. For this reason, 0.5A FlipKY components should be assembled using pick and place equipment with a placement tolerance no greater than 100µm. Slight misplacement of FlipKY components should self-correct during reflow. Overtravel/pressure settings

should also ensure that the device bumps come into contact with the printed solder paste.

Blow Off Settings

During internal trials at IR, it was found that components could occasionally “stick” to the pick and place nozzle. This could result in FlipKY devices not being placed on the PCB. This was found to be a consequence of the pick and place machine settings and nozzle condition.

If customers encounter this type of issue, they should check the nozzle for any signs of damage or blockage, and also check the vacuum filters of the pick and place equipment.

Reflow and Cleaning

The recommended reflow profile is dependant on the type of FlipKY bump solder. 0.5 FlipKY is available with Sn63Pb37 solder and SnAgCu solder. SnPb solder melts at ca. 183°C. For this application, reflow profile guidelines are shown in Figure 9. This example consists of four zones:

Zone 1: Preheat zone, typically heating at 1-2°C sec⁻¹

Zone 2: Soak zone, at 120-160°C for 60-120sec

Zone 3: Reflow zone peaking at 210-230°C and remaining above 183°C for 30-60sec

Zone 4: Cooling zone, typically bringing the board back to room temperature at 2-4°Csec-1

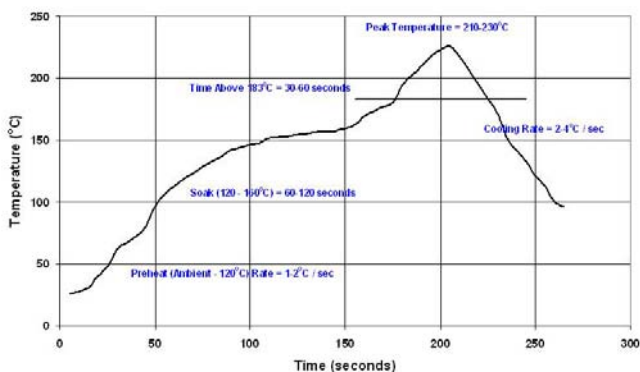


Figure 9 Example of reflow profile for assembling

Eutectic 0.5A FlipKY

Recommended reflow profile guidelines for SnAgCu solder (which melts at ca. 217-221°C) can be seen in Figure 10 and also consist of four zones:

Zone 1: Preheat zone, typically heating at 1-2°C sec⁻¹

Zone 2: Soak zone, at 150-200°C for 60-120sec

Zone 3: Reflow zone peaking at 250-265°C and remaining above 217°C for 40-80sec

Zone 4: Cooling zone, typically bringing the board back to room temperature at 2-4°Csec-1

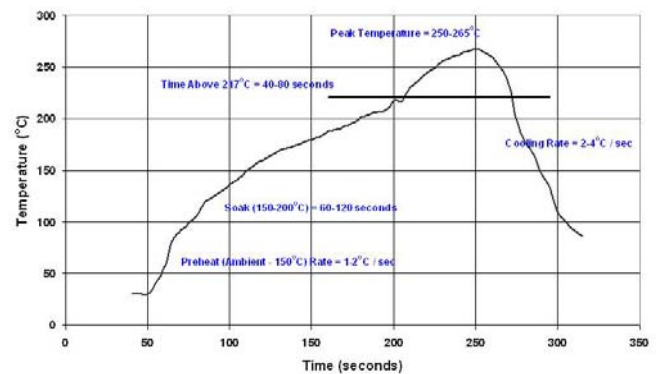


Figure 10 Example of reflow profile for assembling Pb-free 0.5A FlipKY

It is important to consider solder paste manufacturers recommendations when setting up a reflow profile. It is also important to consider the thermal mass of surrounding components and PCB material.

It is recommended that customers use a no-clean solder paste. This means that no cleaning is necessary after reflow. If customers choose to use a cleaning process, they must ensure that cleaning results in complete removal of all contaminants under the device.

Underfill

0.5A FlipKY products do not need underfill to achieve consumer level reliability. No underfill

materials were used in the qualification of 0.5A FlipKY products. However, specific applications where the component is exposed to extreme levels of shock or vibration may require underfill material to be used.

Underfill materials should be carefully selected to give good adhesion to the silicon nitride passivation, solder mask and any flux residue. Materials acting as both flux and underfill are available from several suppliers and may be suitable for use.

Rework Guidelines

Modern rework stations for ball grid array and leadless packages often use two heating stages. The first heats the substrate, either with a conventional hot plate or a hot-air system. The second stage uses a hot-air system for localized heating, often with the option of unheated air for faster cooling of the solder interconnections on the replaced device; this improves the solder grain structure.

The device placement mechanism or arm usually has a hot-air de-soldering gun as part of the pick head equipped with a vacuum cup and thermocouple. Once the solder reflow temperature has been reached, the vacuum is automatically engaged to allow the device to be removed from the substrate. This reduces the risk of causing damage by premature removal.

To replace a WLP device:

Note: If you usually bake to remove residual moisture before rework, insert your normal procedure here.

1: heat the substrate to approximately 100°C using substrate heating stage. This reduces the amount of heating required from the hot-air de-soldering tool, which in turn reduces the risk of damaging either the substrate or surrounding components.

2: Lower the placement arm to bring the de-soldering tool into contact with the device. When

the device and solder interconnects reach reflow temperature, lift the placement arm to remove the device from the substrate.

3: Clear the residual solder and flux from the site with a blade-type de-soldering tool and de-soldering braid. Take care in cleaning the site: damage to the solder-resist may produce undesirable results. When the site is ready, apply new solder paste with a micro screen and squeegee.

4: Heat the site to approximately 100°C using the substrate heating stage. Position the new device with the placement arm and then use the de-soldering tool to heat both device and solder interconnects to reflow temperature. Retract the arm, leaving the device in Place. Cool as quickly as possible to achieve good grain structure in the new joint.

Handling After Board Mount

There should be no mechanical contact with the 0.5A FlipKY die after assembly onto the PCB. Mechanical contact from handling and tooling/equipment can cause damage to the 0.5A FlipKY components.