Assembly of FlipFET™ Devices

by Hazel Schofield and Martin Standing, International Rectifier

FlipFET™ is a new generation of HEXFET® Power MOSFET package from International Rectifier. FlipFET™ combines the latest die design and wafer level packaging technology to give the smallest package footprint possible.

The FlipFET™ is a modified HEXFET die that has gate, source and drain connections on the front face. The die face is passivated to give protection from the environment.

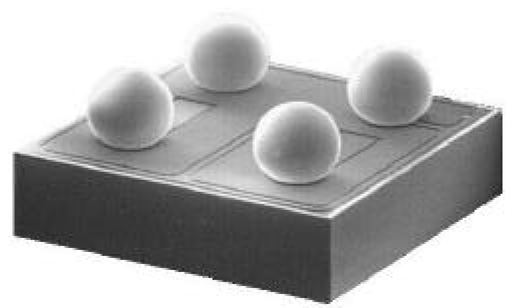

Figure 1 - SEM Photo of FlipFET™ Device

This application note introduces the FlipFET™ devices and discusses the assembly issues and PCB technology involved in their use.

Introduction

Although strictly speaking FlipFET™ is a flip chip, its properties are very similar to a BGA or CSP package:

- 0.8 mm pitch
- Eutectic solder interconnects
- Assembled using printed solder
- Reliable on organic substrates
- Requires no underfill
- Fully tested
- Presented in tape and reel or wafer format

The above attributes allow FlipFET™ to fit into existing SMT assembly processes using standard reflow profiles.

Important factors in the assembly process are:

- Board design
- Board materials and finish
- Stencil design
- Solder paste choice
- Pick and place accuracy
- Reflow profile

The implications of these and the requirements for assembly of FlipFET™ are discussed over the following pages.

Board Design

A well-designed and manufactured printed circuit board is required for all fine-pitch components. FlipFET™ is no exception to this rule. The following recommendations are based on an extensive assembly investigation, as well as industry information and research.

Pad Design

There are two options for the design of the pads as shown in Fig. 2.

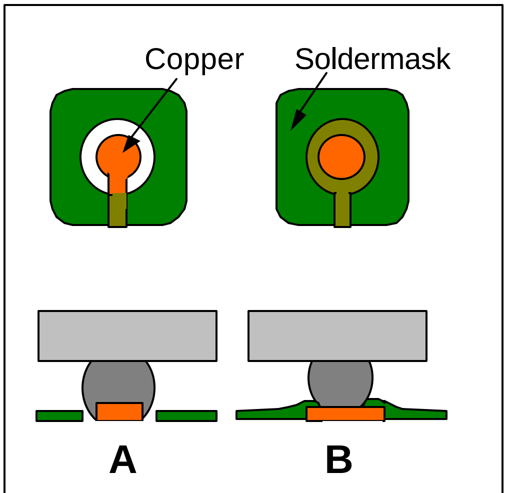

Figure 2 - Pad Styles

In both cases, the pad diameter available for soldering is 250µm. Option A shows non-soldermask-defined pads. This will allow wetting of the solder around the sides of the pads, giving a strong solder joint. When designing this type of pad, it should be noted that the area of track exposed by the soldermask will also be available for soldering. This should be kept equal on each of the four pads to ensure planarity of the device, and should be kept to a maximum of 150µm wide to prevent excessive movement of the solder away from the ball position.

Option B shows solder mask defined pads. Some industry reports have indicated that the angle formed between copper, mask and solder may form an area more susceptible to thermal fatigue than that found in option A. Work is currently ongoing to determine at what die size this becomes significant.

When designing with non-soldermask defined pads the quality of the etch process is important. It is essential that the artwork is adjusted to add the required etching tolerances to the pad size. This will ensure a finished pad of 250µm; pads less than 200µm diameter are not acceptable. Whilst the thickness of the copper will not directly affect the reliability of the assembly, etching of pads of this size into copper of greater than 1oz/ft^2 is extremely difficult.

The clearance of the soldermask from the edges of the pad should be a minimum of 50µm, to allow space for the solder to flow around the edges. The positional and dimensional tolerances in the soldermask process should be taken in account when making this specification.

When designing with soldermask defined pads, the thickness of the copper will not affect the assembly. It must be ensured in this case that the washout during developing of the mask is good. Thin layers of mask around the edges of the aperture will prevent full wetting of the solder and lead to a less reliable joint.

The heat path from the active face of a FlipFET™ to the PCB is extremely short giving it an excellent thermal performance. The solder balls are the primary direction of heat flow, making the printed circuit board the primary heat sink. The thermal requirements of the assembly should therefore be considered when designing the printed circuit board.

Surface Finishing

Over the past few years, the finishing industry has been introducing new products at a very high rate. Whilst this means that a finish can be chosen to fit the application, it makes the choosing process very lengthy. FlipFET™ has been designed to be suitable for use with a wide range of finishes; the only constraint is that it must be flat - Hot Air Levelled solder (HAL) is not recommended as it does not give consistent solder volumes on each pad. This would lead to tilted die and reduced reliability.

Finishes such as Electroless Nickel/Immersion Gold, Organic Solderability Preservatives and immersion Silver and Tin are all suitable.

If electroplated Nickel/Gold is considered for use, the presence of brittle Gold/Tin intermetallics should be carefully investigated. These brittle layers will lead to a significant reduction in the fatigue life of the solder joints. This problem can be reduced by keeping the gold thickness to a minimum. The standard gold thickness for immersion gold at less than 0.2µm does not introduce enough gold into the solder joints for this to be a concern with this finish.

Board Materials

FlipFET™ has been designed to be reliable on all board types, including organic such as FR-4 or polyimide as well as ceramic.

Assembly Process

The two basic process flows available for assembly of FlipFET™ are shown in Figure 3.

Option A will be discussed first as it allow easy integration into a standard SMT process.

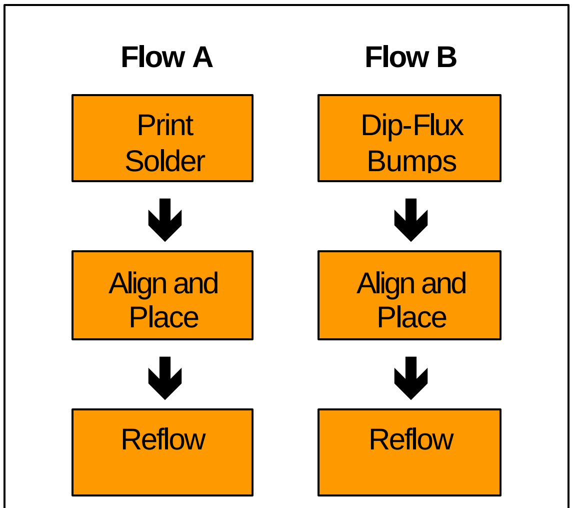

Figure 3 - FlipFET™ Assembly Process Options

Stencil Design

One of the most important factors in the soldering process is consistency in the volume of solder deposited on the pads.

During the assembly trials, various combinations of stencil thickness and aperture size were considered. Clearly larger apertures in thinner stencils will be the easiest printing combination. With a 75µm stencil, apertures in the range 250-300µm diameter were found to give good results. When using larger apertures, it is probable that some of the paste will be deposited on top of the soldermask; this can lead to the formation of solderballs.

When using thicker stencils (150µm and above), release of the paste becomes more difficult. Poor release in itself may not be a problem, but variability in the release will lead to inconsistent solder volumes. Other studies have found that square apertures, particularly those offset to form a diamond may give better paste release in these situations.

There are a number of methods of forming solder stencils, including chemical etching, laser cutting and electroforming. Electroforming in particular gives a very smooth sidewall that can aid paste release.

It is also possible to specify stencils that have apertures that taper from bottom (board side) to top (print side). A difference of 25-50µm can also aid paste release.

Solder Paste

A low residue no clean solder paste with Sn63/Pb37 alloy is recommended for use with FlipFET™ devices. Powder type 3 has been found to be suitable. The particular choice of paste will depend on the surface finish. Finishes that are oxidised and not easily solderable such as OSPs will require a more active flux than those that are easily soldered such as gold. As with all fine pitch components great care should be taken to avoid formation of extraneous solder balls, these could become lodged beneath the die or between the contacts causing a short circuit.

It may be possible to use a solder paste requiring cleaning, however the cleaning process must be carefully developed and monitored to ensure that all corrosive materials are removed from under the die.

Printing Process

The percentage of the pad required to be covered by solder paste will depend on both the flux and the surface finish. With electroless nickel/immersion gold and a moderately active flux, even 50% coverage of the pads was found to be enough to completely wet the pads during reflow. With some combinations however, full coverage may be required. A picture of printed paste is shown in Figure 4.

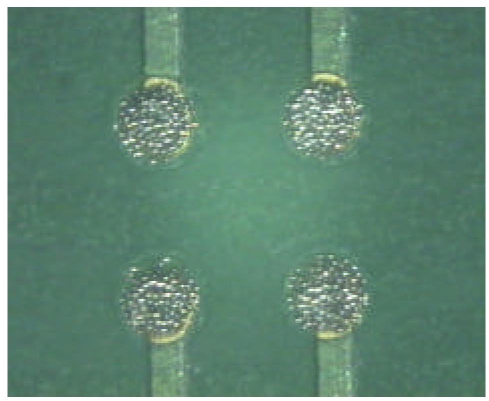

Figure 4 - Printed Solder Paste

Dip Fluxing

Flip chip fluxing stations are available for many pick and place machines. These generally consist of a flat surface onto which the flux is applied, and a doctor blade that is adjusted to give the required film height. It is very important that the film is consistent in thickness to ensure that an equal amount is applied to each ball. Again, a low residue, no-clean flux is recommended, but the comments on cleaning in the section on solder paste also apply here.

The required thickness of the film will depend on the activity of the flux and the surface finish. The maximum thickness used should be 125µm or half the height of the ball. This will prevent transfer of the flux onto the die surface.

The flux should be tacky enough that it evenly coats the balls, and holds the device in place during the transfer to the reflow oven, however it should not be so tacky that strings form during removal of the device from the flux film.

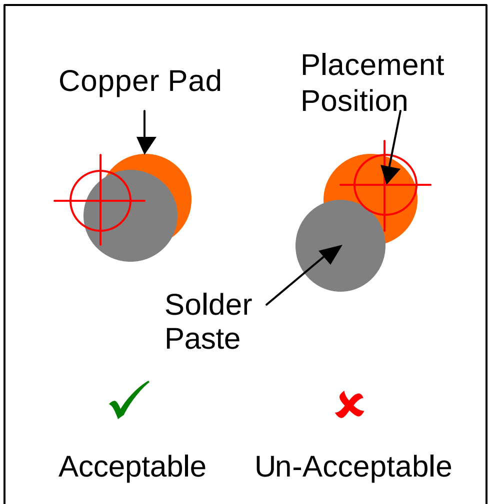

Figure 5 - Print and Placement Configurations

Pick and Place

The placement accuracy required is +/- the radius of the pads on the board; i.e. +/- 125µm if the design recommendations above are being followed. This will ensure that the centre of the ball is always placed on the pad. It should be noted however that if the accuracy of the solder print is poor the solder ball may not be in contact with the printed paste and will have no flux available to aid reflow. This is likely to lead to a non-wetted joint. Acceptable and unacceptable configurations are shown in Figure 5.

Due to the surface tension of the solder, devices that have all balls placed in solder on the pads will self align to give symmetrical joints.

FlipFET™ die are presented bumps down in tape and reel format. They have been 100% inspected for solder defects as well as fully tested at 100% of rated values. They can therefore be picked up and placed directly with no need for intermediate operations.

Automated vision systems can either be programmed to determine the position of the edges of the die, or the positions of the individual bumps. In both cases, it is very important that the lighting is correct. For edge detection standard lighting should prove sufficient, however side lighting may need to be considered for the detection of bumps.

Reflow Process

Solder paste manufacturers recommendations make a good basis for development of a reflow profile. It is very important to verify the profile on the board being used. Differences in the thermal mass of boards and components can make a large difference to the actual temperatures reached.

It must be ensured that all FlipFET™ joints reflow evenly and that any printed solder fully coalesces with the solder on the die to form a uniform joint. A picture of a good solder joint is shown in Figure 6.

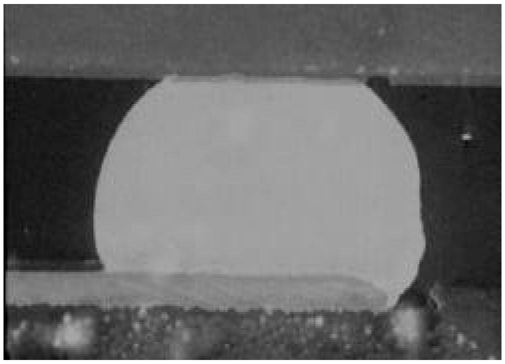

Figure 6 - Microsection of a good solder joint

Underfill

Underfill is not required for this device. Reliability tests have shown it to pass both temperature cycling and high humidity/bias tests without the need for any further encapsulation.

In certain harsh environment applications, it may be advisable to underfill this device. This will give further improved temperature cycling performance as well as protection against the environment.

Underfill materials should be carefully chosen to give good adhesion to the die passivation and soldermask materials and any remaining flux residue.

Materials that act as both flux and underfill are available from several suppliers. These may also be suitable for use with FlipFET™ devices.