Rework Process for TQFN Packages

Prepared by Qian Ouyang

March 2, 2013

ABSTRACT

MPS proprietary Thin Quad Flat package No leads (TQFN package) is evolved from the industry standard Quad Flat package No leads (QFN package). It provides superior electric and thermal performance over the standard QFN package and is very suitable for high current DC to DC converters. This application note introduces the rework guidelines of the MPS proprietary TQFN package.

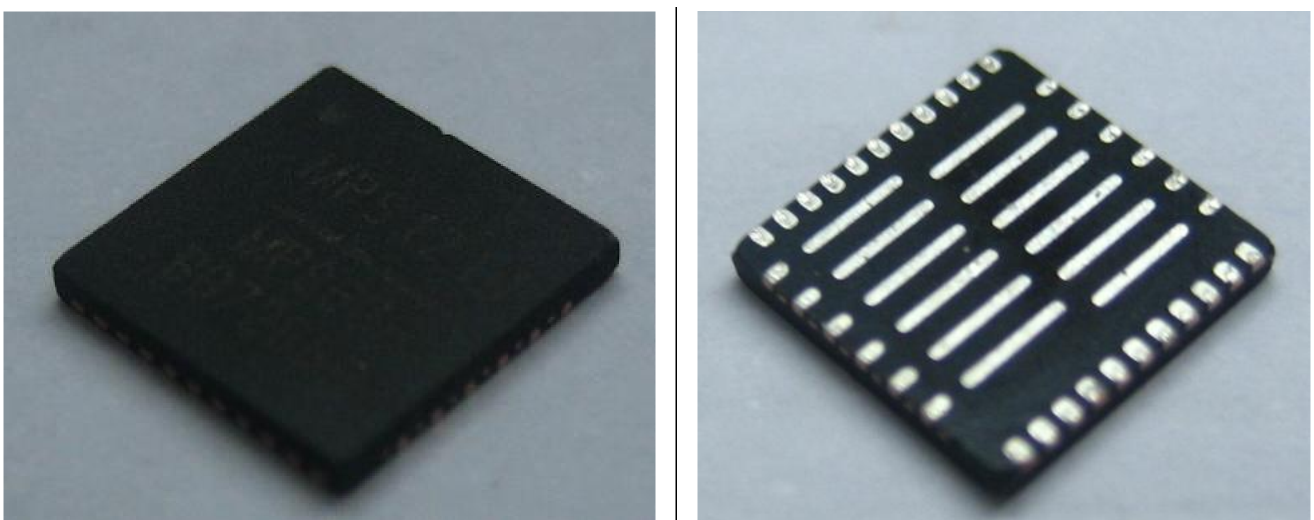
INDEX

TQFN PACKAGE INTRODUCTION

MPS proprietary TQFN package is an enhanced version of the standard QFN package. The biggest difference between the MPS TQFN and standard QFN package is that the big thermal pad on the QFN package is split into many small ones on TQFN package. Each small thermal pad has electric connection to the inner power devices. A picture of the TQFN package is shown as Figure 1. Its key attributes are given in Table 1.

The TQFN package provides lots of advantages shown as below:

- Superior electrical and thermal performance compared to QFN and leaded plastic packages.
- Easy for PCB layout.
- Small footprint results in significant PCB space savings.
- Can utilize the standard surface mount assembly technology.
- Proved Board-Level Reliability per “IPC-9701-Performance Test Methods and Qualification Requirements for Surface Mount Solder Attachments”.



A) Package Top View

B) Package Bottom View

Figure 1—Picture of MPS TQFN Package

Table1—Attributes of TQFN Package

Typical Lead Pitch	**Control 0.5mm/ Power 0.7mm**
Package Total Height	0.9mm
Lead Coplanarity	Leadless

REWORK PROCESS

For the TQFN parts, if there is any defect underneath the package, the whole package has to be removed. Rework of TQFN packages can be a challenge due to their small size. In most applications, TQFN parts will be mounted on smaller, thinner, and denser PCBs that introduce further challenges due to handling and heating issues. Since reflow of adjacent parts is not desirable during rework, the proximity of other components may further complicate this process. Because of the product dependent complexities, the following only provides a guideline and a starting point for the development of a successful rework process for these packages.

Usually, the rework of TQFNs includes the following steps:

- Thermal profiling;
- Removal of defective component;
- Site redress;
- Solder paste replenishment;
- New component placement;
- Reflow soldering.
- Final inspection.

Thermal Profiling

Thermal profiling is a very important step when reworking TQFNs. A suitable thermal profile ensures that the components and board are not overheated and all the solder joints are reflowed. An improper profile may result in the lifting of pads if the solder is not molten before removal is attempted. On the other hand, if the component is heated extensively, damage can occur to the inner structures of TQFNs and/or the surrounding components on the PCB.

The rework removal and replacement profiles are similar to the assembly reflow profile. It is necessary to monitor the temperatures at critical locations on the assembly to ensure that the temperature of the solder joints, the temperature difference across the site, and the temperature of the adjoining components should also be monitored.

The recommended reflow profile guidelines are shown in Table 2 and Figure 2.

Table 2—Temperature Profile Guidelines

Reflow Profile	Description	Process Windows
Preheat	Initial heating of component leads	A linear ramp rate of 0.5°- 2.0°C/second allows gradual evaporation of volatile flux constituents and helps minimize defects such as solder balling and/or beading and bridging resulting from hot slump.
Thermal Soak	Solder paste dries out and flux activates	150°C to 180°C, 0.3°C/second.
Reflow	Time above 230°C/Peak reflow temperature	30 to 40 seconds/260°C maximum
Cooling	Cooling rate	A rapid cool down is desired to form a fine grain structure. Slow cooling will form a large grain structure, which typically exhibits poor fatigue resistance. The acceptable cooling range is 0.5°C-6.0°C/second

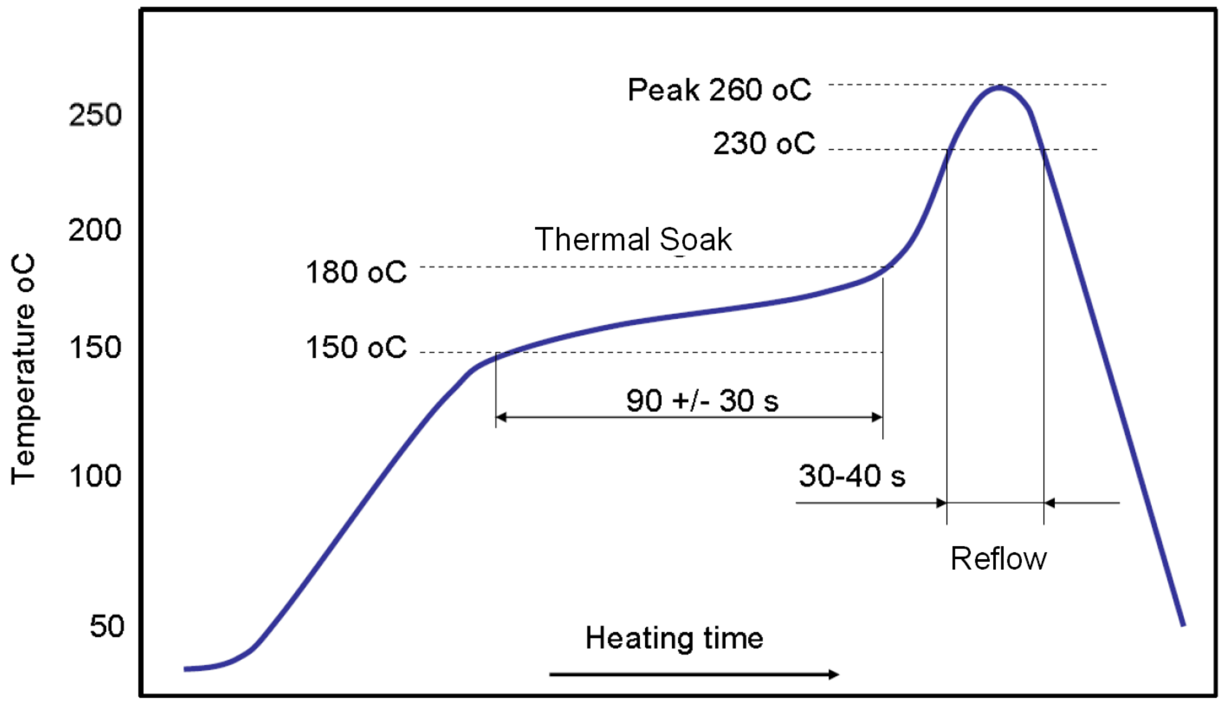


Figure 2—Recommended Temperature Profile

Removal of Defective Component

Don’t remove the defective components by hand. Do it by the rework station which have a split light system, an XY table for alignment, and a hot air reflow system with a top and bottom heater.

To remove the defective component from the board, hot air should be applied from the top and bottom heaters. An air nozzle of correct size should be used to conduct the heat to the TQFN components according to the temperature profile shown in Figure 2. Then use the vacuum pick up tool to remove the component properly. The pictorial procedure is shown in Figure 3. Many assembly sites have extensive in-house knowledge on rework, and their experts should be consulted for further guidance.

Figure 3—Removal of Defective Component

Site Redress

The residual solder on the reworked site has to be removed to obtain a flat surface for the placement of the new component. The lack of a flat surface can result in open joints, odd-shaped joints, and component displacement during soldering of the new component. Hence, it is important that a consistent amount of solder be left on the pad and the surface is flat for successful component replacement.

Use soldering iron and braid to clean the residual solder. Place a copper braid on the residual solder and heat it using a soldering iron. The temperature of soldering iron should be low enough not to cause any damage to the circuit board. The residual solder melts and wicks up to the copper braid thus leaving a thin layer of solder on the PCB pads. Care should be taken to ensure that the integrity of the board is not compromised in any way. The width of the solder wick and the tip of the soldering iron are to be chosen carefully. Inappropriate size of the solder wick and soldering iron tip may damage the solder mask and rip the pads off the PCB.

Once the residual solder has been removed, the lands should be cleaned with a solvent. The solvent is usually specific to the type of paste used in the original assembly and paste manufacturer's recommendations should be followed.

Solder Paste Replenishment

Design a mini-stencil according to the guideline given in AN060 and then place the mini-stencil in the component site. Use a mini-metal squeegee blade to deposit solder paste in the specific area. The printed pad should be inspected to ensure even and sufficient solder paste before component placement.

In situations where neighboring parts are at close proximity with the TQFN components, and the mini-stencil method is not an option, apply solder paste carefully on each pad using a syringe. The volume of solder paste will be difficult to control.

Selection of the appropriate powder size for a specific solder paste application is an elemental step that will ultimately affect the printability of the solder paste with regard to the mini-stencil design.

Area Ratio is an essential component to the printing process and powder size choice. Calculating the Area Ratio and choosing the correct powder size can assist in proper stencil release. The Area Ratio calculation method is introduced in the previous section. Once the proper aperture size has been determined, using the Area Ratio, the appropriate powder size can then be chosen. For all apertures, it is important to choose the correct powder size so that a minimum of 4-5 solder particles (the large particle size of the range) can be maintained across the aperture.

Type 3 and Type 4, no lean and lead-free solder paste are recommended for MPS TQFN parts. The maximum power size of the solder paste should be no greater than 45um. Table 3 lists the key dimensions of the pad, stencil and solder powder. It can be seen clearly from this table that there are minimum 4.7 solder particles across the aperture.

Table 3—The Key Dimensions of the Pad, Stencil and Solder Powder

Printing Parameters					Power Size	
Pitch	Pad Width	Aperture Width	# of small spheres	# of large spheres	Diameter Range	
0.5	0.25mm	0.25mm	12.5	5.6	20um	45um
0.5	0.25mm	0.21mm	10.5	4.7	20um	45um

New Component Placement

A vacuum nozzle is used to pick the new package up. The split light system displays images of both the TQFN leads and the footprint on the PCB. The two superimposed images are aligned manually by adjusting the XY table. Once the PCB and the package are aligned, the package is placed down on the PCB.

Reflow Soldering

The reflow profile developed during original attachment or removal should be used to attach the new component. Since all reflow profile parameters have already been optimized, using the same profile will eliminate the need for thermocouple feedback and will reduce operator dependencies.

Inspection

Post-reflow inspection of TQFN parts on PCB is highly recommended, and is typically accomplished by using transmission-type X-ray equipment. X-ray can be used for reflow process monitoring and as a failure analysis tool. A 2-D X-ray equipment with Oblique View at Highest Magnification (OVHM) is preferred due to its capability to detect solder bridges, opens and voids.

REFERENCES:

[1]. "Power Choice Stencil Design Guidelines", Indium application note. www.indium.com.

[2]. "TQFN Package Design, Fabrication, and Assembly Guidelines", MPS application note.