E-PAD product PCB layout guideline

Scope: PCB design guidelines for products with thermo PAD

1. PCB designer comply with IPC-7525 and IPC-7351 and IPC-7095 requirements is recommended.

2. QFN/DFN /SOP E-pad package PCB layout guideline:

2.1. Because of the rich solder paste volume under QFN/DFN/SOP thermo pad area which will raise the QFN/DFN package during reflow process, this kind of effect will cause QFN/DFN terminal floating and solderbility fail issue. To prevent the QFN/DFN floating and become solderability open short issue during SMT process, the solder past coverage between 15% to 30 % of thermo pad area is recommended. The blue color areas of Figure QFN/DFN is a just sample for stencil design. Anyway, stencil design should be fine tune by SMT house if the poor contact issue happened.

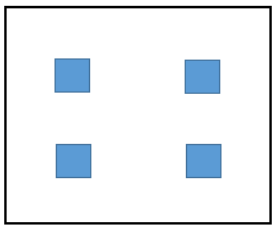

Figure QFN/DFN

3. PCB process

3.1. Stencil Design Guidelines: Refer to IPC-7525 Stencil Design Guidelines process.

3.2. Reflow Oven: Forced convection reflow with nitrogen is recommended for Pb-free and Green package.

3.3. Reflow profile: Using more than 8 zone oven is recommended for Pb-free and Green package.

3.4. To use IPC-A-610 is recommended for soldered electrical and electronic assemblies.

4. Rework and Repair Guide

4.1. Reballing BGA/CSP is not recommended for production applications if there are no special reball fixtures and tools. There are many rework system on market, however, special reball fixtures and tools have been designed to simplify and help control this process. For additional information, refer to IPC -7711/21A Rework and Repair Guide or IPC-7095A (Design & Assembly Process Implementation for BGA's) or search IPC website.

4.2. QFN/DFN Rework and Repair Guide: The Rework and Repair Guide of QFN/DFN package is the same with BGA products which need special re-work fixtures and tools, and use IR-reflow process.