SMT ASSEMBLY FOR LEADLESS PACKAGES

Introduction

This document outlines standard recommended practices for surface mount assembly of Hittite Microwave Corporation no-lead quad flatpack (QFN) and leadless ceramic chip carrier (LCCC) style packages.

General Description of QFN and LCCC Packages

Hittite standard products that make use of QFN and LCCC style packages carry the suffix "LP" and "LC", respectively.

LP style package construction consists of a copper alloy leadframe with plastic overmolding. Available finishes are 100% matte-tin and tin-lead solder. Outline drawings and suggested PCB land patterns for available LP style packages can be accessed through the following link:

http://www.hittite.com/support/plastic-smt-packages.html

LC style package construction consists of an air-cavity alumina substrate with tungsten interconnections. Pad metalization is gold plating over nickel. Outline drawings and suggested PCB land patterns for available LC style packages can be accessed through the following link:

http://www.hittite.com/support/ceramic-glass-metal.html

Component Outlines

LP style packages are available in nominal sizes ranging from (2.0 mm x 2.0 mm x 1.0 mm) to (6.0 mm x 6.0 mm x 1.0 mm), and with pin-outs ranging from 6 to 40 peripheral (i/o) pads.

LC style packages are available in nominal sizes ranging from (3.0 mm x 3.0 mm x 1.0 mm) to (5.0 mm x 5.0 mm x 1.0 mm), and with pin-outs ranging from 12 to 32 peripheral (i/o) pads.

Both LP and LC package styles have an exposed ground-pad, which must be attached to an RF ground plane with a sufficient thermal path to ensure reliable device operation.

PCB Solder Mask and Land Pattern Design

Due to the tight pitch and small pad geometry of LP and LC style packages, adherence to the manufacturer recommended PCB land pattern geometry is critical. The application of a solder mask is required, and must be applied to the substrate between the peripheral pads and the ground pad. A minimum solder mask width of 0.010 inches [0.25 mm] between the peripheral pads and the ground pad should be used in order to prevent solder-bridging. To ensure maximum ease of assembly, it is recommended that all vias in the ground pad be epoxy-resin filled and overplated ("capped").

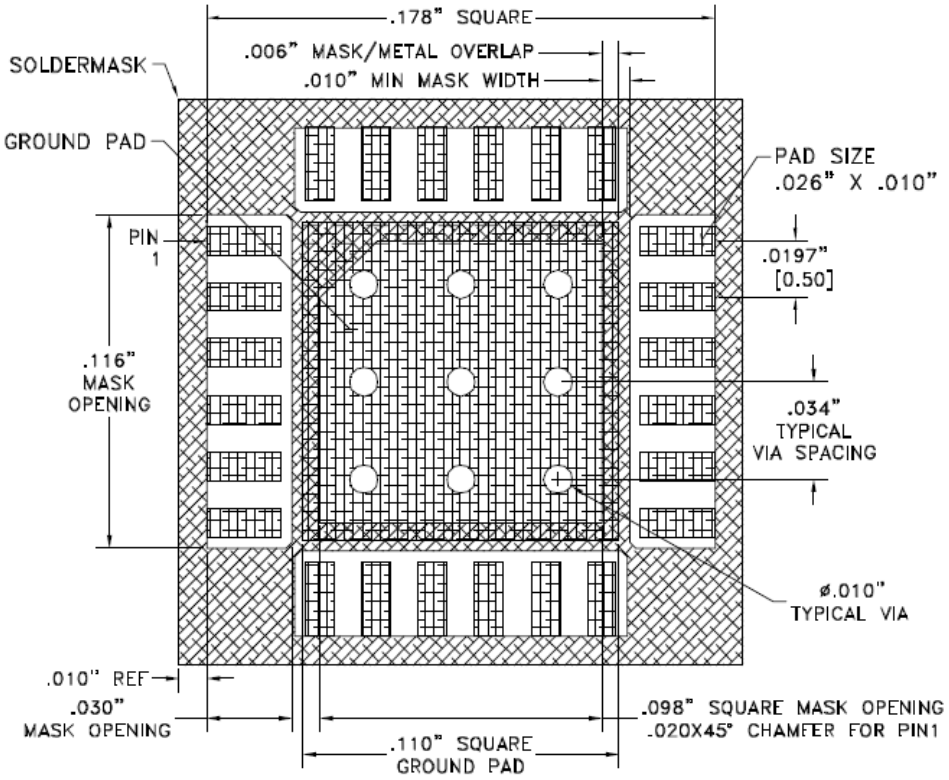
Solder Paste Application and Stencil Design

Industry standard surface mount assembly of fine-pitch devices requires reflow solder assembly in which solder paste is applied to the PCB, followed by device placement and reflow. Typically, solder paste application in high-volume manufacture employs a contact stencil printing process. The stencil is a metal mask with apertures allowing solder paste to be deposited on the pads. Critical stencil printing process parameters are alignment of the stencil apertures to the PCB pads, as well as control of the resulting solder paste volume. The volume of solder-paste deposited by a stencil printing process is a function of many secondary variables (solder paste temperature, stencil finish, paste age, relative humidity of the manufacturing facility, etc.) but is primarily dictated by stencil aperture size and stencil thickness. Recommended stencil parameters are indicated by the example

SMT ASSEMBLY FOR LEADLESS PACKAGES

calculations below. Automated optical inspection of solder paste is recommended to confirm that adequate paste volume is being deposited by the stenciling process.

Recommended Stencil Aperture Calculation (.005" Stencil)

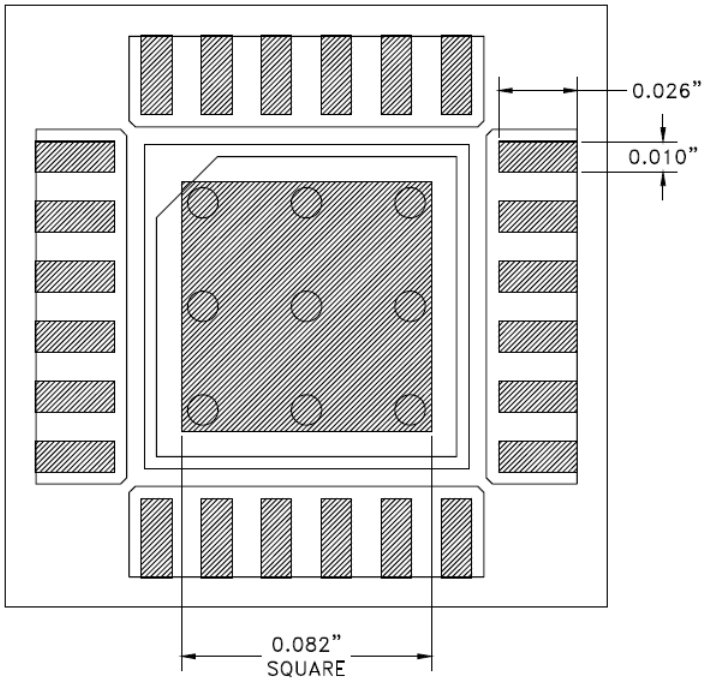


Example Land Pattern (LP4, LC4)

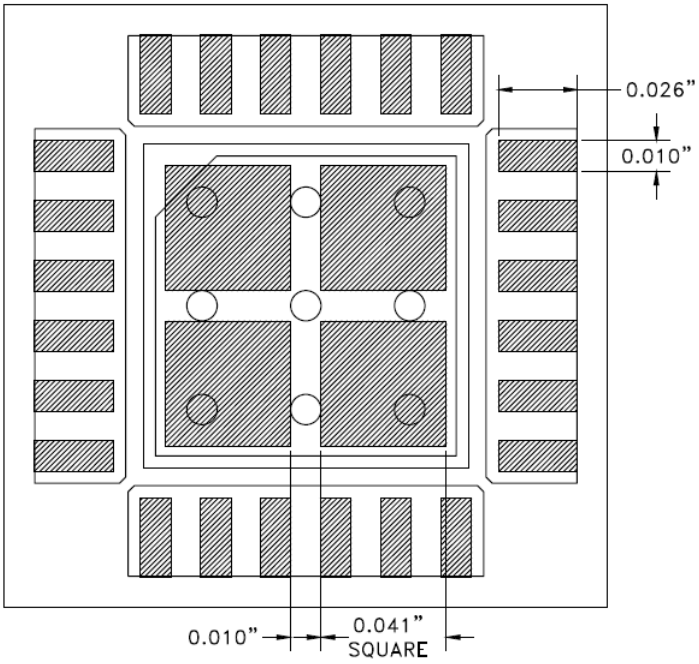
Line	PCB Footprint		
A	Peripheral Pad Size	.010 x .026	in
B	Ground Pad Size, Solder mask defined pad	0.098 x .098	in
C	Ground Pad Area (=B^2)	0.00960	in^2
	Stencil Aperture		
D	Pad Aperture (=A)	.010 x .026	in
E	70% Area Calculation (=.7*C)	0.0067	in^2
F	Ground Pad Aperture (=Square Root (E))	0.082	in

Stencil Aperture Calculation (Assumes 70% paste deposition on ground pad by area)

SMT ASSEMBLY FOR LEADLESS PACKAGES



Stencil aperture indicated by shaded region



Optional .010" windowpane on ground pad stencil aperture
Note: Windowpane stencil aperture pattern is intended to be implemented on devices with outer dimensions >3mm only, and should not affect 70% paste deposit by area.

2 Elizabeth Drive Chelmsford, MA 01824 Phone: 978-250-3343 Fax: 978-250-3373

SMT ASSEMBLY FOR LEADLESS PACKAGES

Package Style	Solder Mask Defined Ground Pad Dimension	Ground Pad Aperture Size	Nominal Pad/Pad Aperture Size
LC3	0.060	0.050	.010 x .026
LC3B	0.058	0.049	.010 x .026
LC3C	0.060	0.050	.010 x .026
LC4	0.098	0.082	.010 x .026
LC4B	0.098	0.082	.010 x .026
LC5	0.138	0.115	.010 x .026
LP2	.031 x .055	.026 x .046	.010 x .024
LP3	0.058	0.049	.010 x .026
LP4	0.098	0.082	.010 x .026
LP4C	0.081	0.068	.013 x .035
LP5	0.138	0.115	.010 x .026
LP6	0.173	0.145	.013 x .028
LP6C	0.173	0.145	.010 x .028

Tabulated Calculation of Required Solder-Stencil Aperture Sizes for HMC Devices

Component Placement

Hand placement of leadless packages with bottom-terminations introduces a high defect rate, and is not recommended. Use of a pick and place machine with integrated vision system is recommended for this operation, as it ensures sufficient X, Y, and theta alignment. Z axis pressure must be limited during placement. To minimize solder paste smearing due to excess displacement, the base of the component should be brought to a height of approximately .003" above the surface of the PCB, which will result in a depression of .001-.002" of the component into the solder paste when using the recommended .005" stencil.

Reflow Soldering and Profiling Process

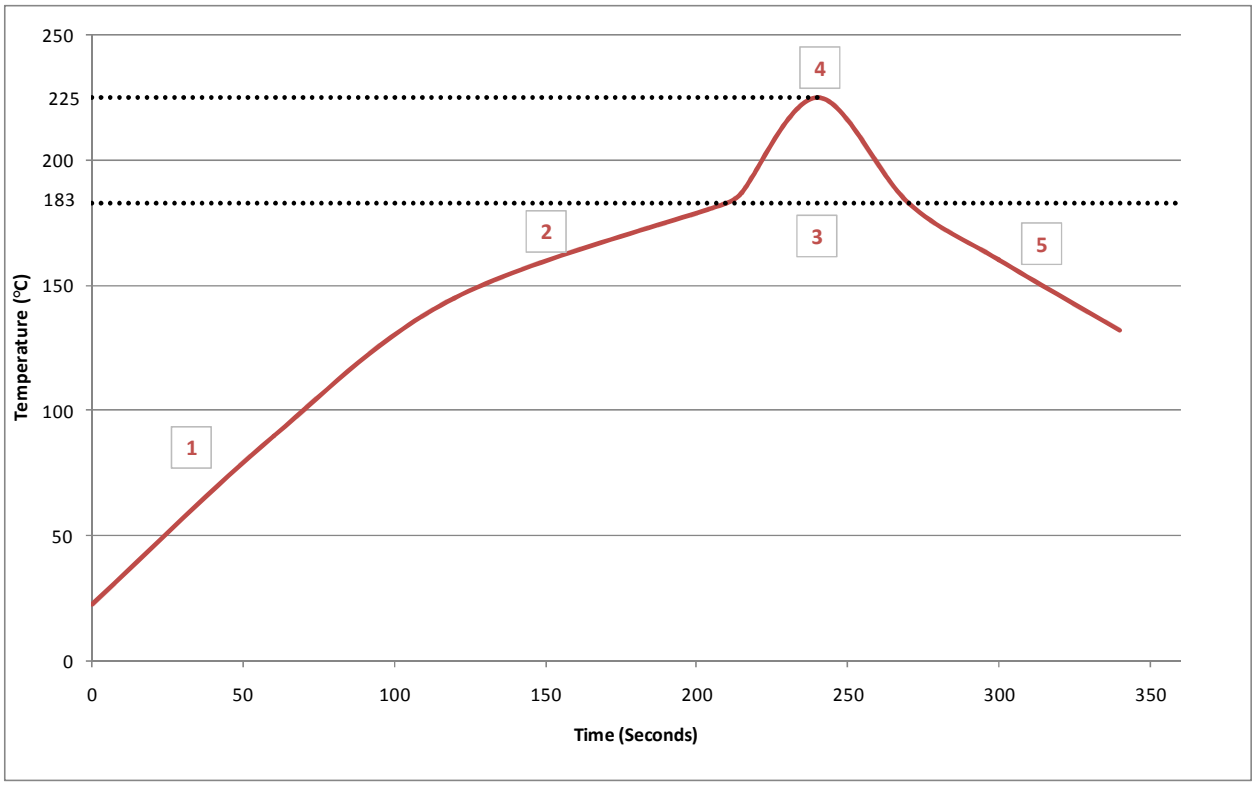
Reflow oven temperature profiles should characterized using an accurate thermal mass, and should be confirmed at all critical component locations (exposed lands) on the board using a shielded thermocouple. A unique reflow profile may be required for each assembly. Adhere to all solder paste manufacturer recommendations. Additional reflow soldering assembly guidelines can be referenced in IPC-7093. Starting process development profiles for tin-lead and RoHS compliant tin-silver-copper (SAC) alloys are shown below.

Critical Profile Parameter		Function
1	Pre Heat	Heat assembly to flux activation temperature
2	Soak	Heat assembly to liquidus temperature while allowing adequate flux activation
3	Time Above Liquidus	Allow soldering to occur
4	Maximum Temperature	Completely reflow solder paste without exceeding peak device temperature
5	Ramp Down Rate	Allow formation of fine grain structure without damaging thermally sensitive components

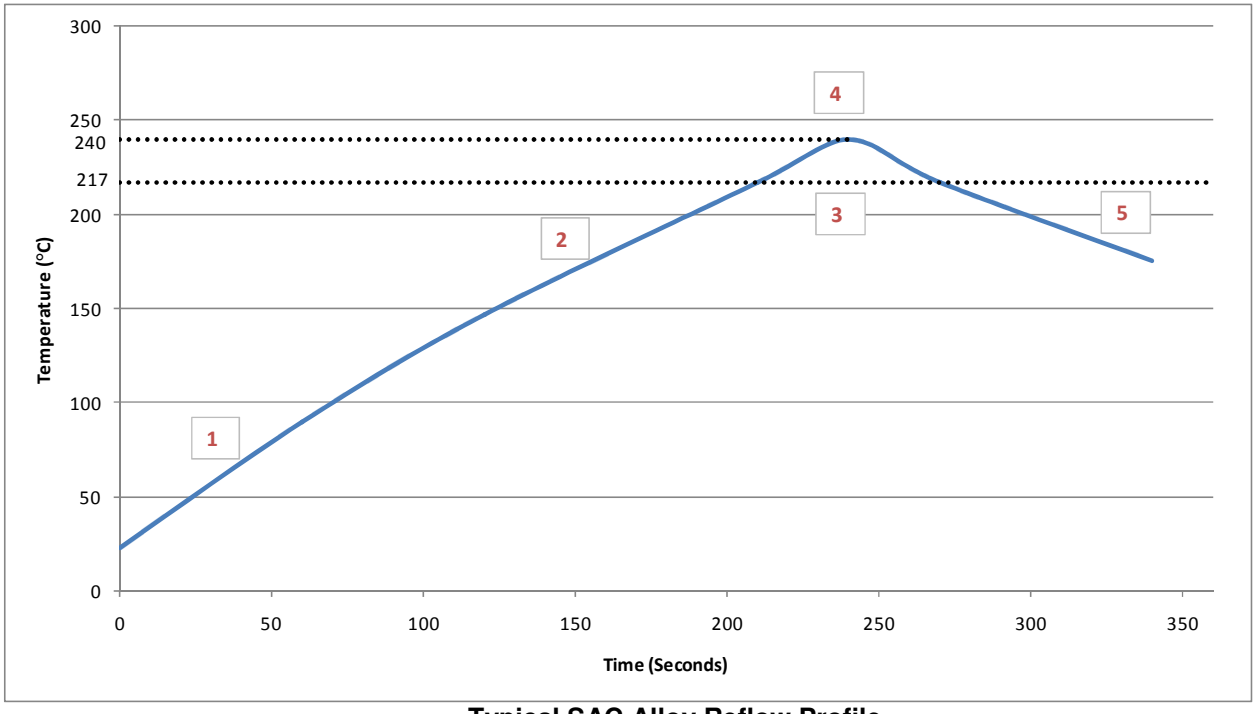
Solder Profile Parameters and Function

2 Elizabeth Drive Chelmsford, MA 01824 Phone: 978-250-3343 Fax: 978-250-3373

SMT ASSEMBLY FOR LEADLESS PACKAGES



Typical Tin-Lead Alloy Reflow Profile



Typical SAC Alloy Reflow Profile

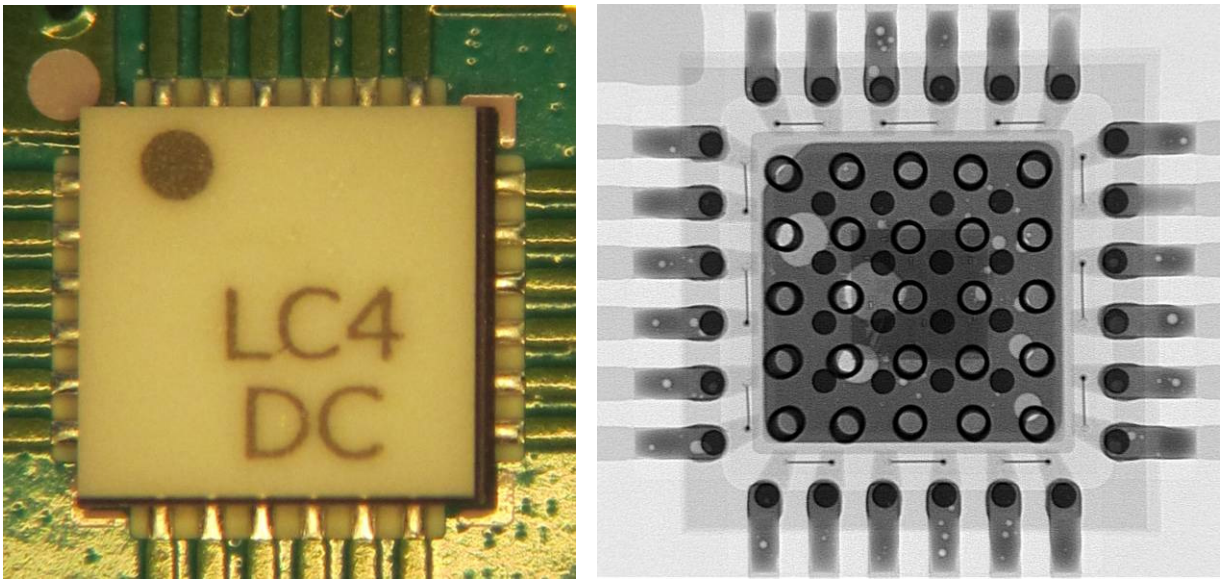
2 Elizabeth Drive Chelmsford, MA 01824 Phone: 978-250-3343 Fax: 978-250-3373

SMT ASSEMBLY FOR LEADLESS PACKAGES

Inspection Techniques

Utilization of standard optical inspection is limited on QFN and LCCC style packages, due to the location of solder joints on the bottom surface of the component. Some inspection criteria such as planarity and alignment can be confirmed through external visual inspection, however, transmission x-ray is the primary method for inspection of solder joints on leadless, "bottom terminated components". Although an x-ray method alone may not be sufficient to demonstrate solder bonding, it is useful in identifying solder volume related defects such as insufficient solder, or excessive solder which may result in bridging between pads. X-ray is typically employed in addition to other measures such as environmental stress screening (ESS) and electrical test, as a means of ensuring SMT process control.

One measure of process control in SMT assembly of devices with thermal/electrical ground pads is the percentage exhibiting voiding, by area. When utilizing these guidelines, the typical percentage of voiding on the ground slug is less than 30% by area. The resulting impact to device reliability and expected lifetime is negligible when voiding is held to below this threshold.



Installed Component, External Visual (Left) and Radiographic Inspection (Right)

Rework

A localized hot air rework station is recommended for removal of components. The localized heat-source provided by the rework station limits the exposure of adjacent components, which may shift if the liquidus temperature of the solder is exceeded.

Once the intended component has been sufficiently heated to reflow the solder joints, it should be removed from the board with tweezers. It is not recommended that the removed part be used in and subsequent assemblies. Excess solder remaining on the PCB lands should be removed using solder braid/soldering iron. Flux residues should be removed in accordance with solder paste manufacturer recommendations, and ionic content should be measured to confirm that no contamination risk exists.

SMT ASSEMBLY FOR LEADLESS PACKAGES

A localized rework stencil should be used for controlled solder paste deposition on the necessary PCB pads. Component placement can be achieved through the use of a split-vision placement system, which superimposes views of the component leads over the PCB pads, allowing for precise manual alignment.

Solder reflow can be accomplished by using the hot air rework station. Sufficient control should be taken in order to ensure that the maximum device reflow temperature is not exceeded, and that adequate time above liquidus is achieved. A thermocouple can be used to verify that the temperature is held to an acceptable limit during rework.

Other guidelines for SMT rework can be referenced in IPC-7711/7721.

Conclusion

This document details PCB layout and assembly processes which are essential guidelines in ensuring successful SMT assembly of leadless packages.

For questions on this application or any other application, please contact Hittite Microwave Corporation for further assistance.

2 Elizabeth Drive Chelmsford, MA 01824 Phone: 978-250-3343 Fax: 978-250-3373