



Introduction

Maxlinear's LGA / BGA module makes the second level interconnection with an array of solderable pads. This array may consist of a layout similar to that of a BGA, however it may also have an arbitrary arrangement of solderable pads that include large pads for high current paths, grounding or thermal dissipation and smaller pads for signals.

Some benefits of the LGA package include:

- LGA has a lower mounted height than BGA. This allows for more space for a heat sink or for a smaller form-factor.
- LGA devices can be used for either lead containing or lead-free assemblies depending on SMT solder paste used.
- LGA eliminates risk of customers receiving units with missing or damaged spheres due to shipping or handling.
- Board-level reliability can significantly exceed customer requirements with optimal PCB design and SMT process.

PCB Design Guidelines

Type		LGA	BGA	
Package	Solder mask defined	SMD		
	Pad opening size (mm)	0.63	0.60	
Pad layout on board (PCB)	Pads Preference		NSMD but SMD OK to use	
	SMD	Pad opening size (mm)	0.63	0.60
		Metal pad size (mm)	0.73 ~ 0.88	
		Vias	Vias between pads (on the planes) on top layer	
	NSMD	Pad opening size (mm)	min. 0.73	
		Metal pad size (mm)	0.63	0.60
		Vias	Vias should be placed outside of pad layout	
Pad finish		OSP, ENIG recommended		

Note: OSP: Organic Solderability Preservative, ENIG: Electroless Nickel Immersion Gold

Table 1: PCB Design Guidelines

LGA

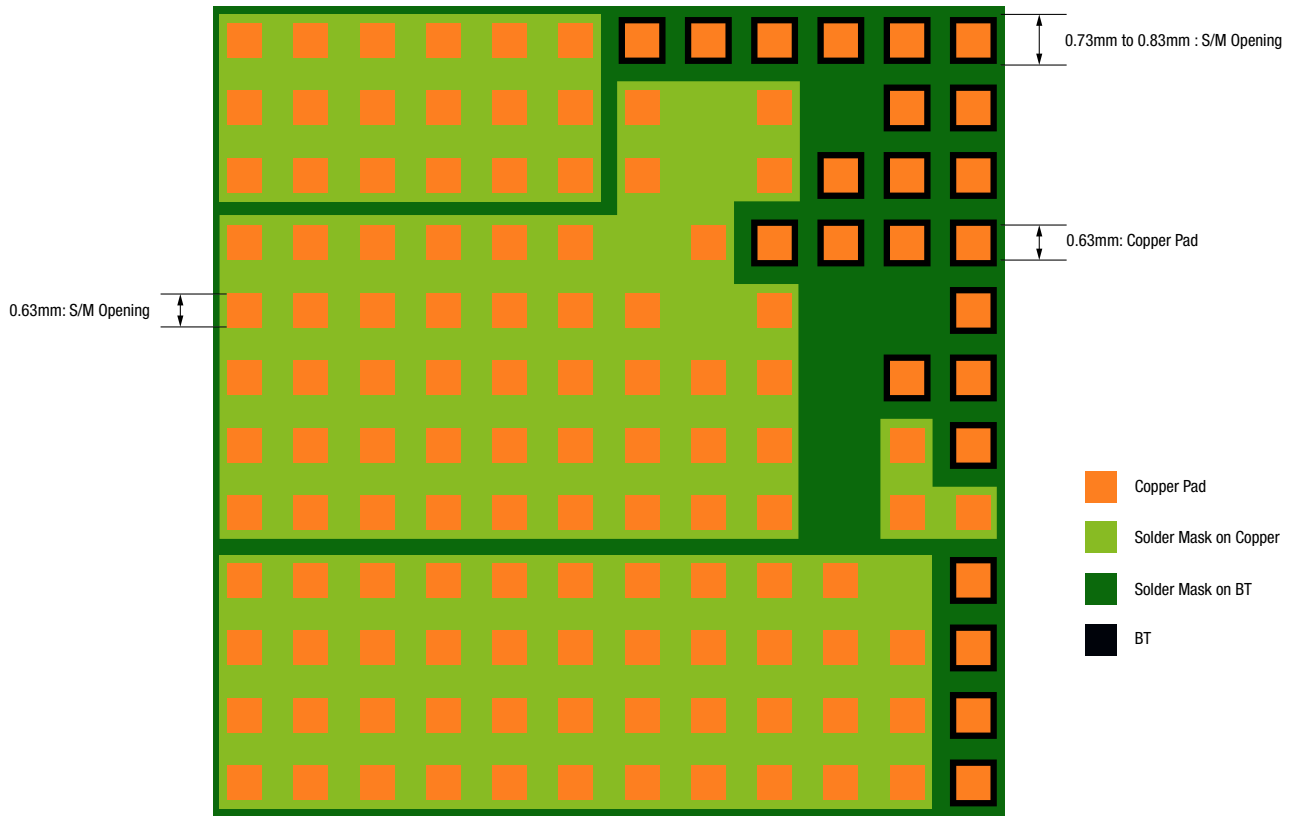


Figure 1: Mixed Pads (SMD and NSMD) for LGA

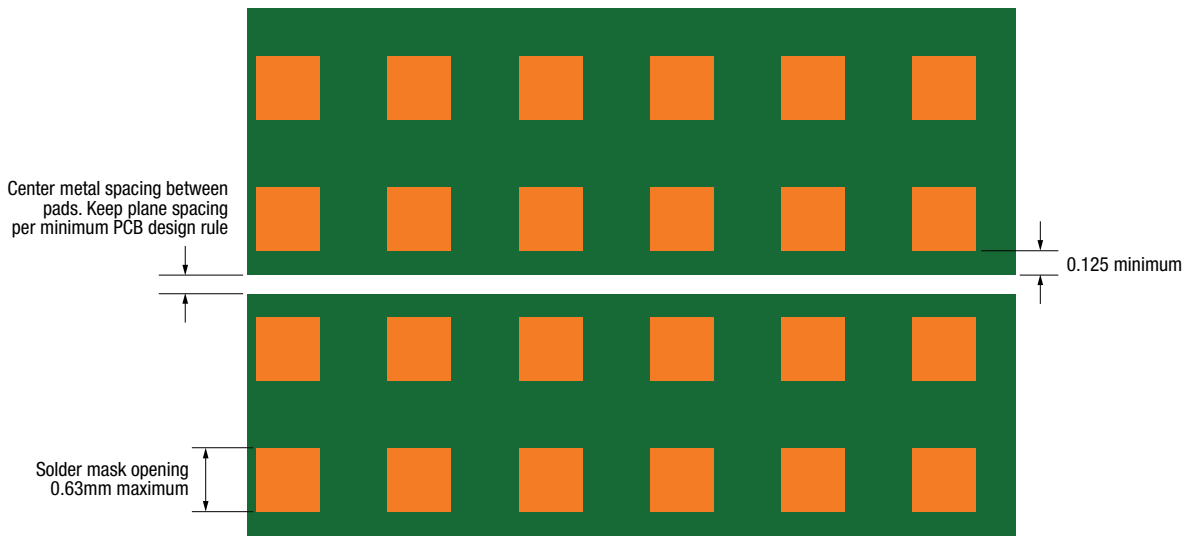


Figure 2: PCB Plane Separation (LGA)

BGA

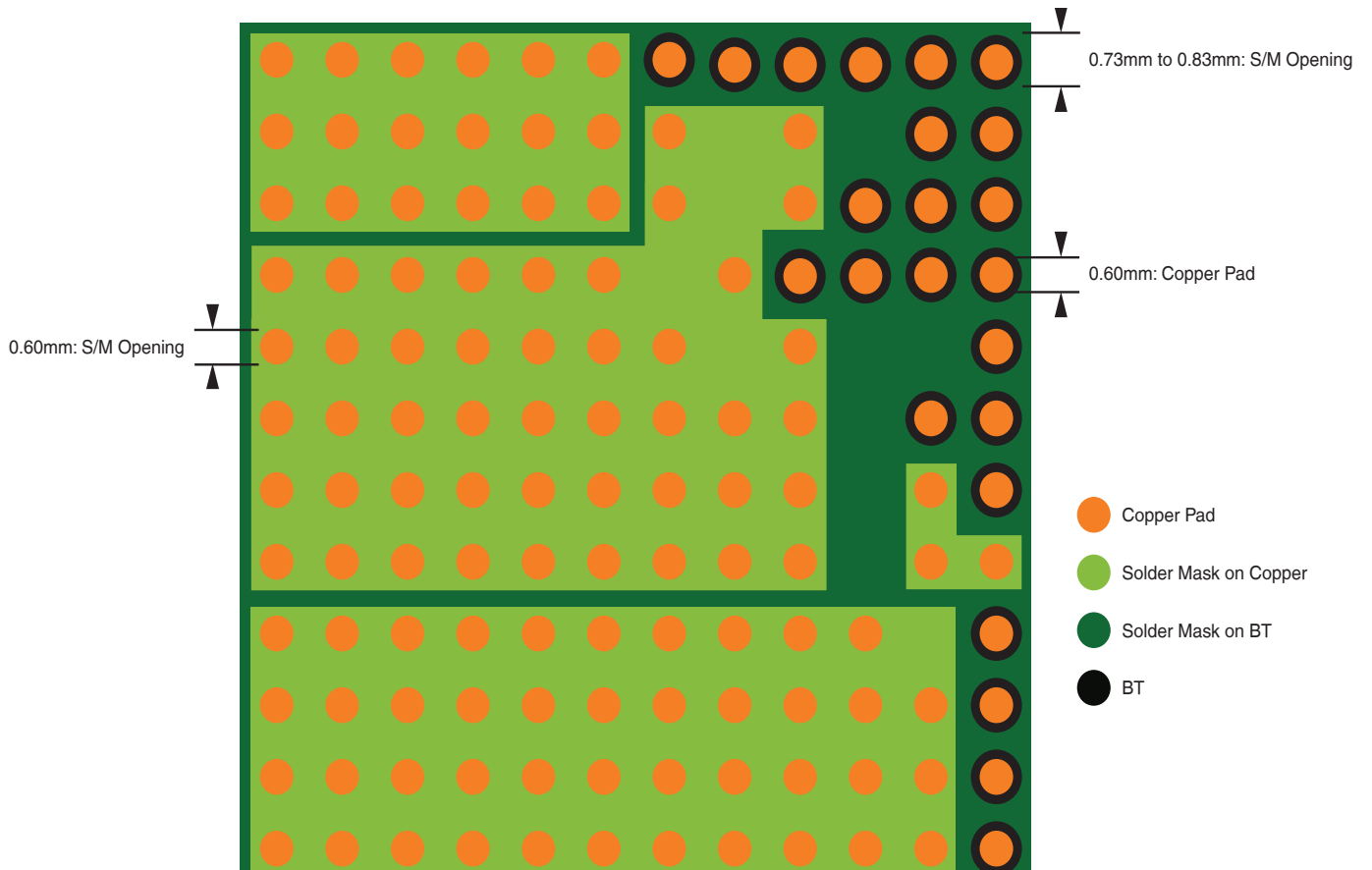


Figure 3: Mixed Pads (SMD and NSMD) for BGA

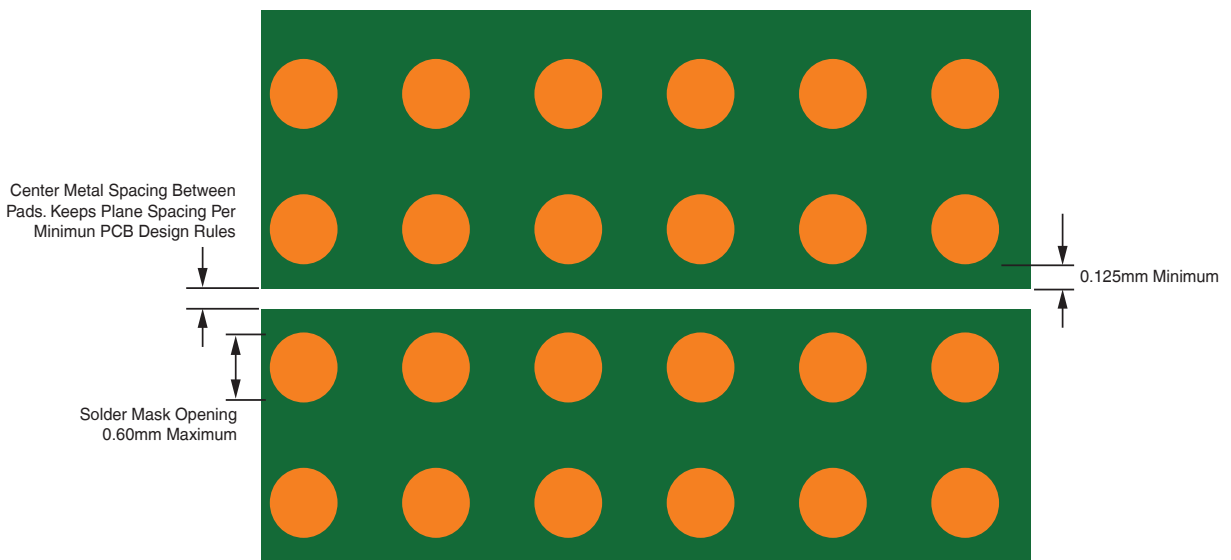


Figure 4: PCB Plane Separation (BGA)

MSL (Moisture Sensitivity Level) Classification

- MSL-3. Refer to JEDEC J-STD-020D.1 for detail.
- Dry packed and vacuum sealed in trays.
- If any of the above packing methods are encountered and moisture indicator shows a pink color, or punctured seal of the bag is observed, bake the packages per the following conditions:
 - 125°C for 48 hours or 150°C for 24 hours
 - If this step is not followed, there is a possibility of delamination of the mold compound from the substrate (solder mask)
 - If the part is heated above 245 deg C, the internal solder in the module may remelt and will spread through the delaminated areas
 - When the part is baked properly, the solder does not spread and is contained within the soldered pad

Land Pattern

- An NSMD (non-solder mask defined) land pattern is recommended, but SMD is OK to use.
- For NSMD, it is recommended to place vias outside the module lands.
- The same pad size is recommended to avoid solder leaking issues during reflow. (No thermal pad with large size.)
- Refer to Figure 7 for the LGA144 15x15 recommended land pattern and stencil design. Refer to Figure 8 for the BGA144 15x15 recommended land pattern and stencil design.

Stencil Design Recommendations

- Stainless steel laser cut stencils are recommended.
- Recommended stencil thickness is 0.125mm (5mils)
 - A slightly smaller stencil aperture than the pad opening is recommended (especially for SMD pads)
 - Stencil Opening: refer to the table below
 - To prevent paste from contacting solder mask
 - Solder volume: refer to the table below
 - 4mil thick stencil is not recommended (due to low stand off)
 - Corner radius of 0.06mm on the aperture is recommended
- Stencil area ratio ($W/4*t$) > 0.66 (not an issue for this aperture size) where W = pad width and t = stencil thickness.

	Pad Size (mm)	Stencil Opening (mm)	Stencil Area (mm ²)	% Area	% Linear	Paste Volume (mm ³)
Square Pads (LGA)	0.63	0.6	0.36	91	95	0.045
Round Pads (BGA)	0.60	0.57	0.25	90	95	0.031

Table 2: Stencil Design

Solder Paste

- Use lead-free (SAC) solder paste – Type 3 or Type 4.

Solder Joint Voiding

- No IPC standard for LGA pad voiding criteria
- A 25% maximum void criteria for solder joints is recommended
- PCB baking at 125°C for at least 4 hours will minimize solder joint voiding

Reflow Profile Design

- A 9 zone or greater reflow oven is recommended.
- Refer to JEDEC J-STD-020D.1 and reflow recommendations from the solder paste manufacturer.
- It is recommended to use a long soak profile with the thicker LGA module with the square size as in Figure 5.
- 2 sided reflow is not recommended.
- Peak temperatures must not exceed the temperatures listed below (Pb free).

Package Thickness	Volume		
	< 350mm ³	350mm ³ - 2000mm ³	≥ 2000 mm ³
< 1.6mm	260 + 0°C	260 + 0°C	260 + 0°C
1.6mm - 2.5mm	260 + 0°C	250 + 0°C	245 + 0°C
≥ 2.5mm	250 + 0°C	245 + 0°C	245 + 0°C

Table 3: Maximum Peak Reflow Temperature - Pb-free Process

Package Thickness	Volume	
	< 350mm ³	≥ 350 mm ³
< 2.5mm	240 + 0/ - 5°C	225 + 0/ - 5°C
≥ 2.5mm	225 + 0/ - 5°C	225 + 0/ - 5°C

Table 4: Maximum Peak Reflow Temperature - SnPb Eutectic Process

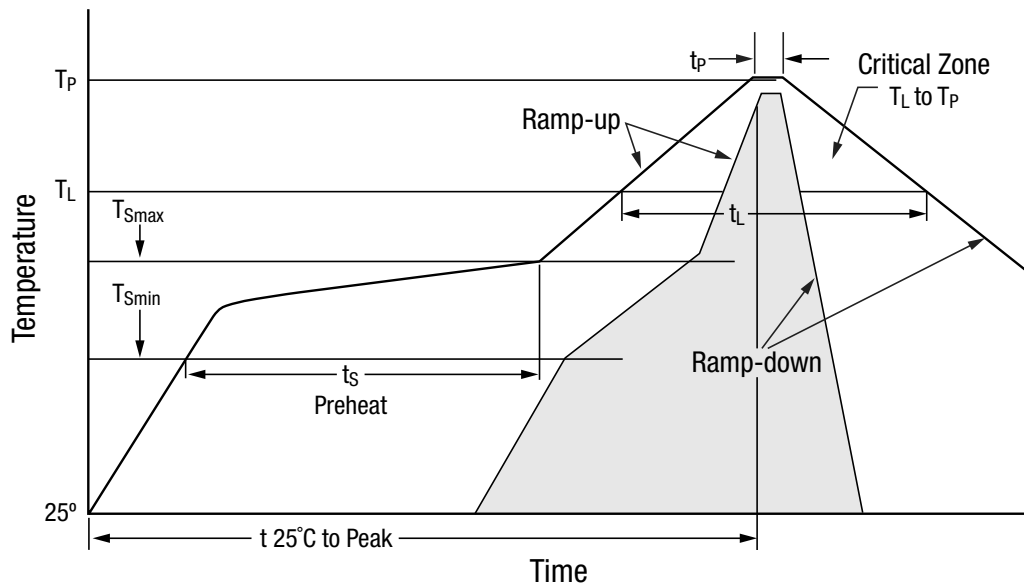


Figure 5: Reflow Profile

Profile Feature		Lead-Free Solder	Leaded Solder
Pre-heat	Min Soak Temperature (T_{Smin})	150°C	100°C
	Max Soak Temperature (T_{Smax})	200°C	150°C
	Soak Time (t_s)	60 - 120 seconds	60 - 120 seconds
Reflow	Liquidus Temperature (T_L)	217°C	183°C
	Time Above Liquidus	30 - 90 seconds	30 - 90 seconds
Peak Package Body Temperature (T_p)		See Table 3	See Table 4
Time within 5°C of peak temp(T_p)		30 seconds max	30 seconds max
Average Ramp up Rate (T_{Smax} to T_p)		2.5°C/second max	2.5°C/second max
Ramp Down Rate		2.5°C/second max	2.5°C/second max
Time 25°C of peak temp.		8 minutes max	8 minutes max
Do not exceed		See Table 3	See Table 4

Table 5: Reflow Profile



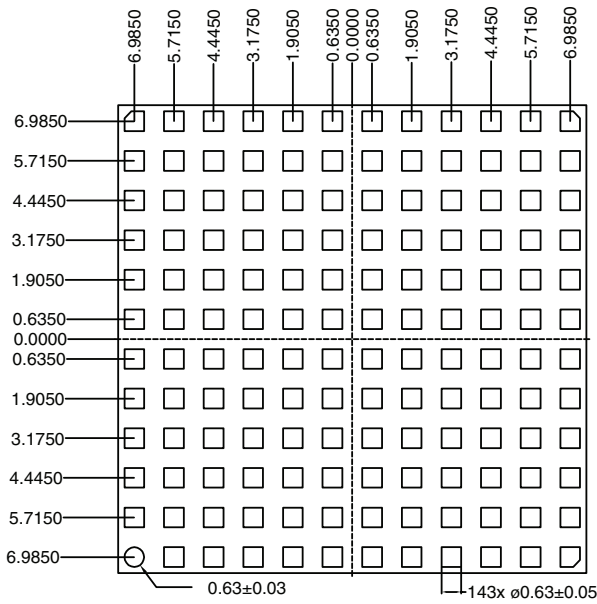
Figure 6: Recommended Reflow Temperature Profile

Rework and Component Removal

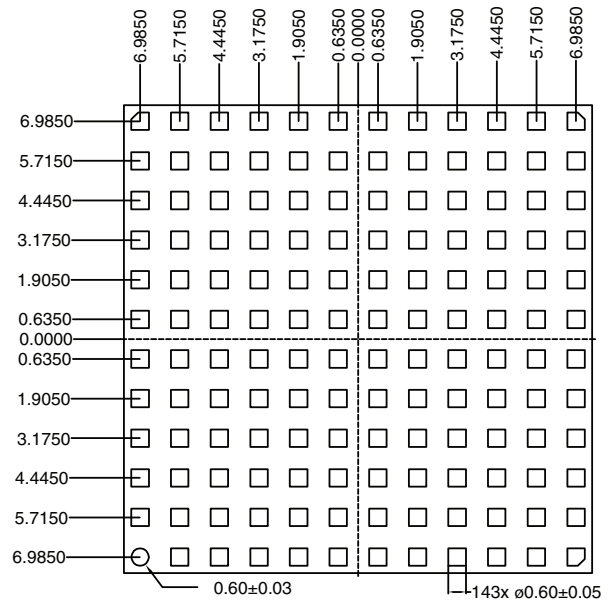
- Determine the failure mode from the board and at what operation the defect occurred. After assembly:
 - Is it open or short?
 - Open: Check solder joint quality, partial joint, no joint, cold solder
 - Short: X-ray to check (pad design, stencil design)
 - Electrical test
 - No output: Check output caps next to the module
 - Shorting
 - » Need X-ray to verify short location internal or external to package
 - » X-ray checks need to be done on the board
- Remove the component from the board within 168 hours of the moisture barrier bag opening prior to assembly or after baking the PC board assembly. Bake the PCB for 24 hours at 125°C.

Recommended PCB Layout

LGA144 15 x15



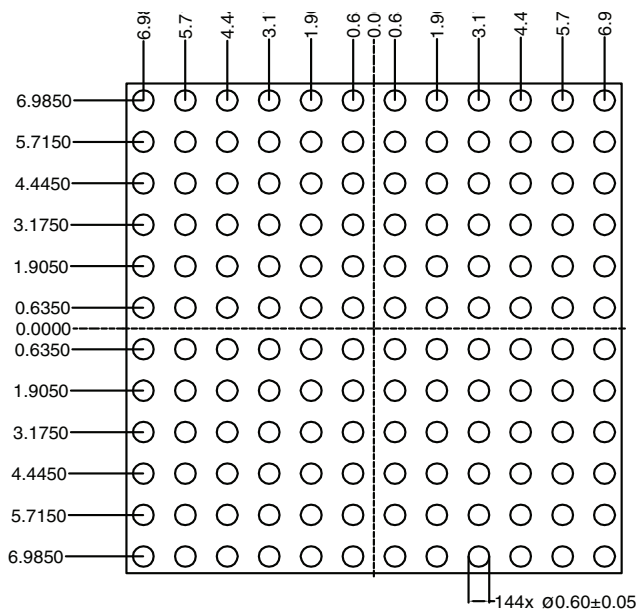
Land Pattern



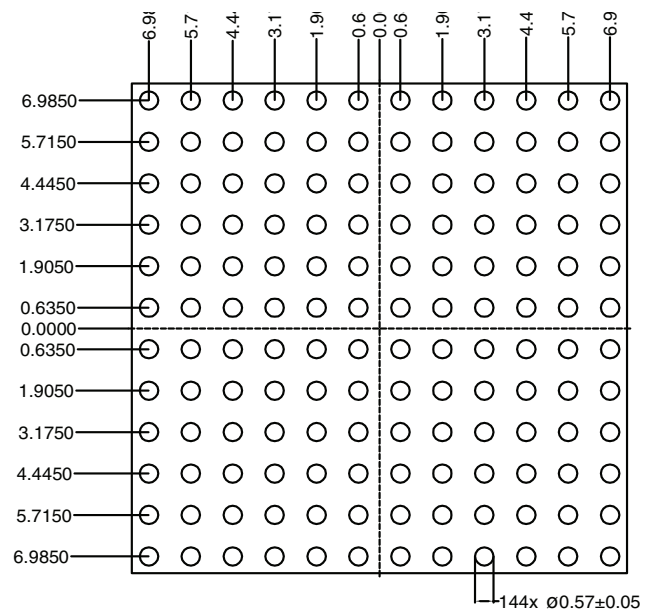
Stencil

Figure 7: Recommended Landing Pattern and Stencil Design for LGA144 15x15

BGA144 15 x15



Land Pattern



Stencil

Figure 8: Recommended Landing Pattern and Stencil Design for BGA144 15x15

Revision History

Revision	Date	Description
A	5/16/14	Initial Release
B	6/2/14	Updated PCB layout and stencil design guideline sections.
C	9/2/15	Added introduction, POD land pattern and stencil opening table.
D	2/5/19	Converted to MaxLinear format. Removed LGA 12x12, 45-lead and added LGA 15x15, 144-lead package and data. Updated MSL, PCB design guidelines, stencil design and reflow profile design sections (including new reflow profile info). Added LGA 15x15, 144-lead land pattern and solder joint voiding section. Added BGA144 15x15.



Corporate Headquarters:
 5966 La Place Court
 Suite 100
 Carlsbad, CA 92008
 Tel.: +1 (760) 692-0711
 Fax: +1 (760) 444-8598
www.maxlinear.com

The content of this document is furnished for informational use only, is subject to change without notice, and should not be construed as a commitment by MaxLinear, Inc. MaxLinear, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in the informational content contained in this guide. Complying with all applicable copyright laws is the responsibility of the user. Without limiting the rights under copyright, no part of this document may be reproduced into, stored in, or introduced into a retrieval system, or transmitted in any form or by any means (electronic, mechanical, photocopying, recording, or otherwise), or for any purpose, without the express written permission of MaxLinear, Inc.

MaxLinear, Inc. does not recommend the use of any of its products in life support applications where the failure or malfunction of the product can reasonably be expected to cause failure of the life support system or to significantly affect its safety or effectiveness. Products are not authorized for use in such applications unless MaxLinear, Inc. receives, in writing, assurances to its satisfaction that: (a) the risk of injury or damage has been minimized; (b) the user assumes all such risks; (c) potential liability of

MaxLinear, Inc. is adequately protected under the circumstances.

MaxLinear, Inc. may have patents, patent applications, trademarks, copyrights, or other intellectual property rights covering subject matter in this document. Except as expressly provided in any written license agreement from MaxLinear, Inc., the furnishing of this document does not give you any license to these patents, trademarks, copyrights, or other intellectual property.

Company and product names may be registered trademarks or trademarks of the respective owners with which they are associated.

© 2014 - 2019 MaxLinear, Inc. All rights reserved