

# Soldering and Mounting Guidelines for the LGA Accelerometer Sensor to a PC Board

by: Kimberly Tuck, Cheol Han  
Sensors and Actuator Solutions Division  
Tempe, AZ

## INTRODUCTION

MEMS based sensors are sensitive to Printed Circuit Board (PCB) reflow processes. For optimal zero-g offset after PCB mounting, care must be taken to PCB layout and reflow conditions. This application note is a guideline for soldering and mounting the LGA package inertial sensors. The purpose of these guidelines is to minimize the stress on the package after board mounting. Both the MMA73x1L 3-axis analog output family of accelerometers and the MMA745xL digital output accelerometer use the Land Grid Array (LGA) package platform. This application note describes suggested methods of soldering these devices to the PC board for consumer applications. Figure 1 shows the bottom view of LGA 14 lead, 3 x 5 mm individual sensor device. Figure 2 shows the recommended PCB land pattern for the package.

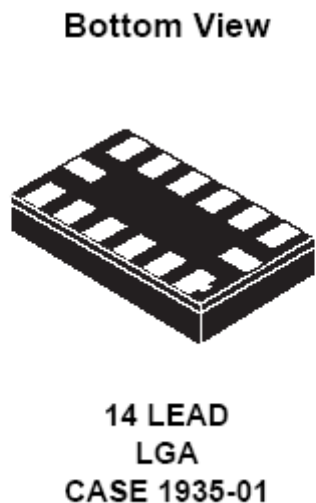


Figure 1. LGA 14-Lead, 5 x 3 mm Die Sensor

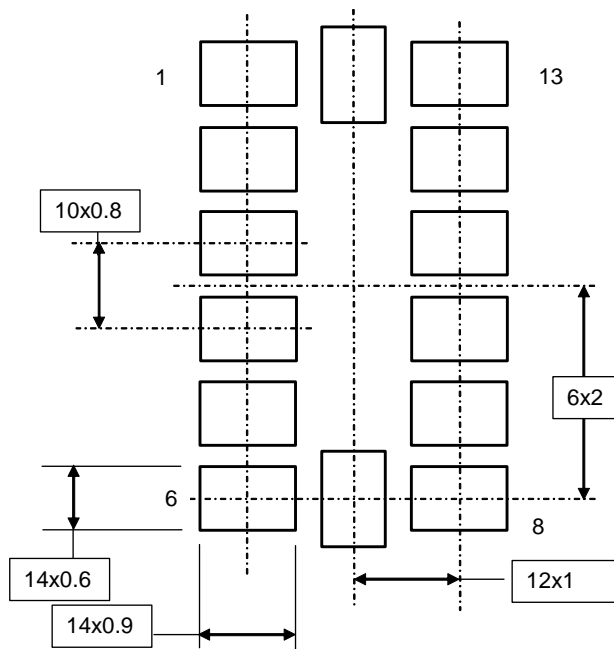


Figure 2. Recommended PCB Land Pattern for the 5 x 3 mm LGA Package

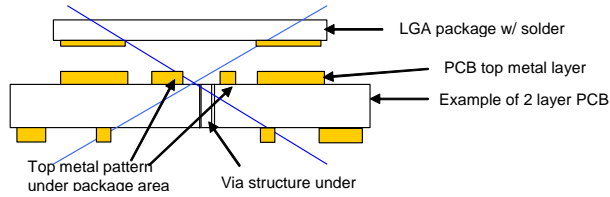
## OVERVIEW OF SOLDERING CONSIDERATIONS

Information provided here is based on experiments executed on LGA devices. They do not represent exact conditions present at a customer site. Hence, information herein should be used as a guidance only and process and design optimizations are recommended to develop an application specific solution. It should be noted that with the proper PCB footprint and solder stencil designs the package will self-align during the solder reflow process.

The following are the recommended guidelines to follow for mounting LGA sensors for consumer applications.

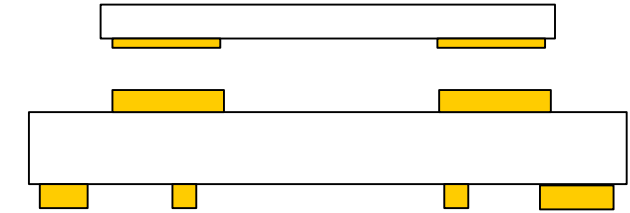
**PCB Mounting Recommendations**

1. The PCB land should be designed with Non Solder Mask Defined (NSMD) as shown in [Figure 5](#).
2. No additional metal pattern underneath package as shown in [Figure 4](#).



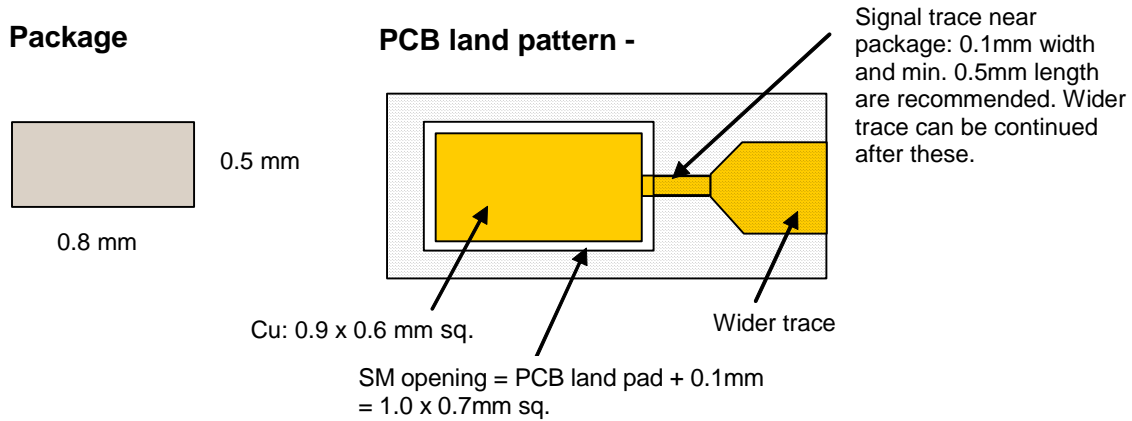
**Figure 3. Incorrect PCB Top Metal Pattern Under Package**

3. PCB land pad is 0.9mm x 0.6mm which is the size of the package pad plus 0.1mm as shown below in [Figure 5](#).

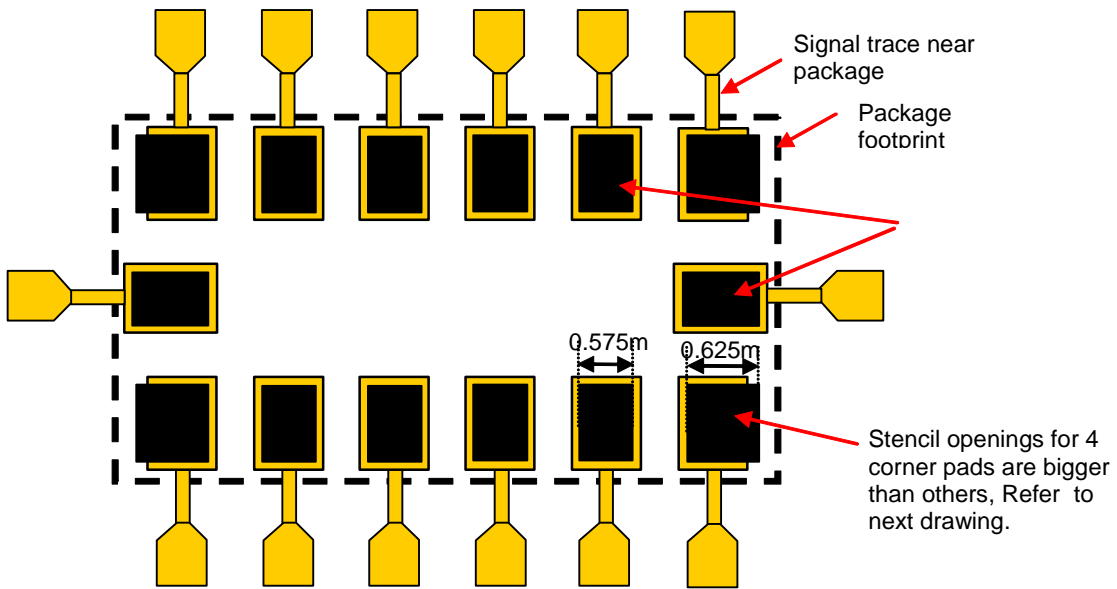


**Figure 4. Correct PCB Top Metal Pattern Under Package**

4. The solder mask opening is equal to the size of the PCB land pad plus an extra 0.1mm as shown in [Figure 5](#).
5. The stencil aperture size is equal to the PCB land pad – 0.025mm. Also note that for the 4 corner pads the aperture size must be larger for solder balancing as shown in [Figure 6](#) and [Figure 7](#). A 6mil thick stencil is recommended.



**Figure 5. Recommended PCB Land Pad, Solder Mask, and Signal Trace Near Package Design**



**Figure 6. Stencil Design Guidelines**



