Introduction

This technical note is intended to provide information about Kionix's 3 x 3 mm LGA packages and guidelines for developing PCB land pattern layouts. These guidelines are general in nature and based on recommended industry practices. The user must apply their actual experiences and development efforts to optimize designs and processes for their manufacturing techniques and the needs of varying end-use applications. It should be noted that with the proper PCB footprint and solder stencil designs, the package will self-align during the solder reflow process.

3x3 LGA Package Marking

- Marking font type : Arial
- Font size : 1.5 Point (0.56 mm height)
- Line space : 0.1 mm
- Text information
  - 1st line – Device name
  - 2nd line – Assembly Build Lot code
  - 3rd line – Date code (WWYY)
  - 4th line – Pin #1 Dot (0.3 mm diameter)

Note - All text lines shall be right justified.

Figure 1. 3x3 mm LGA package marking information

36 Thornwood Dr. – Ithaca, NY 14850
tel: 607-257-1080 – fax:607-257-1146
www.kionix.com - info@kionix.com

3x3 LGA Package Outline and Dimensions

The following diagrams show the outline of the Kionix's LGA packages with dimensions and tolerances. All dimensions and tolerances conform to ASME Y14.5M-1994. All dimensions are in millimeters and angles are in degrees.

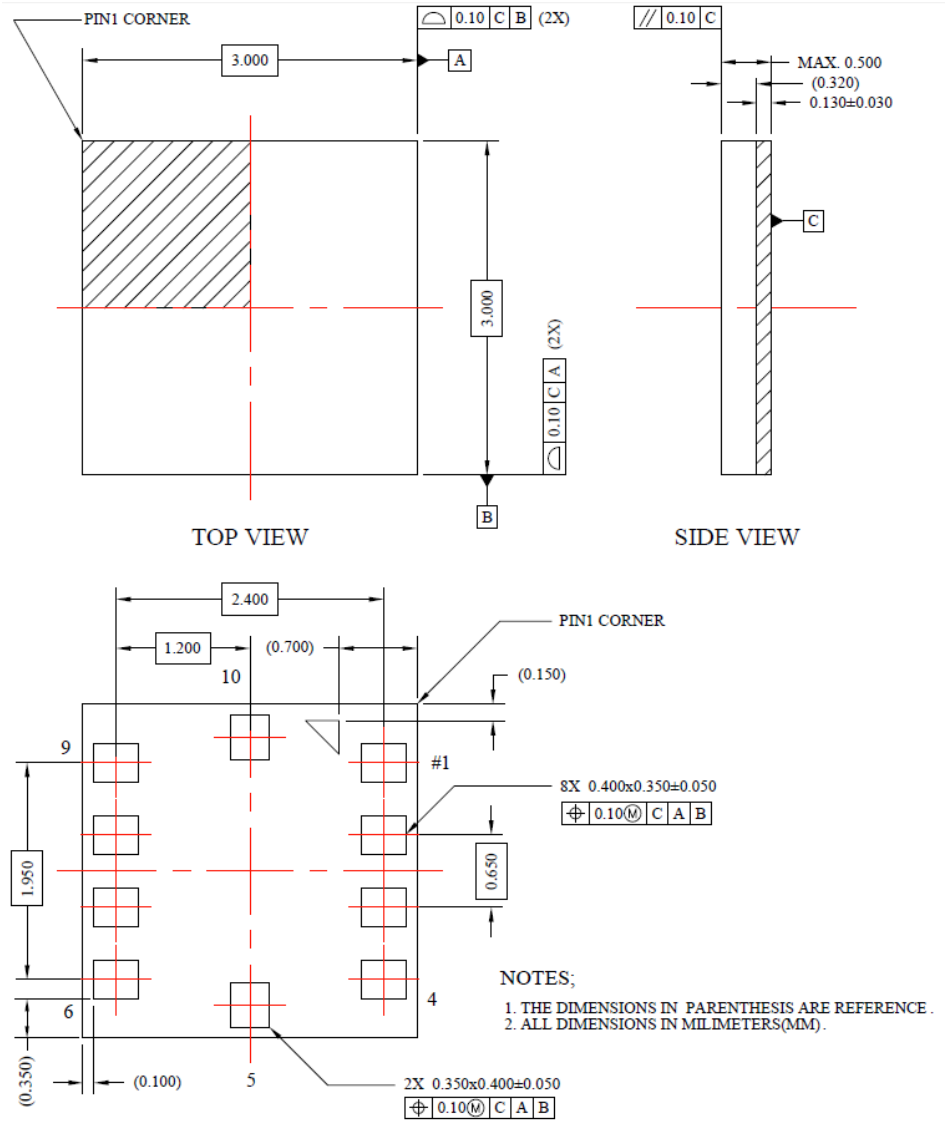


Figure 2. 10-pin 3 x 3 mm LGA package outline diagram with dimensions

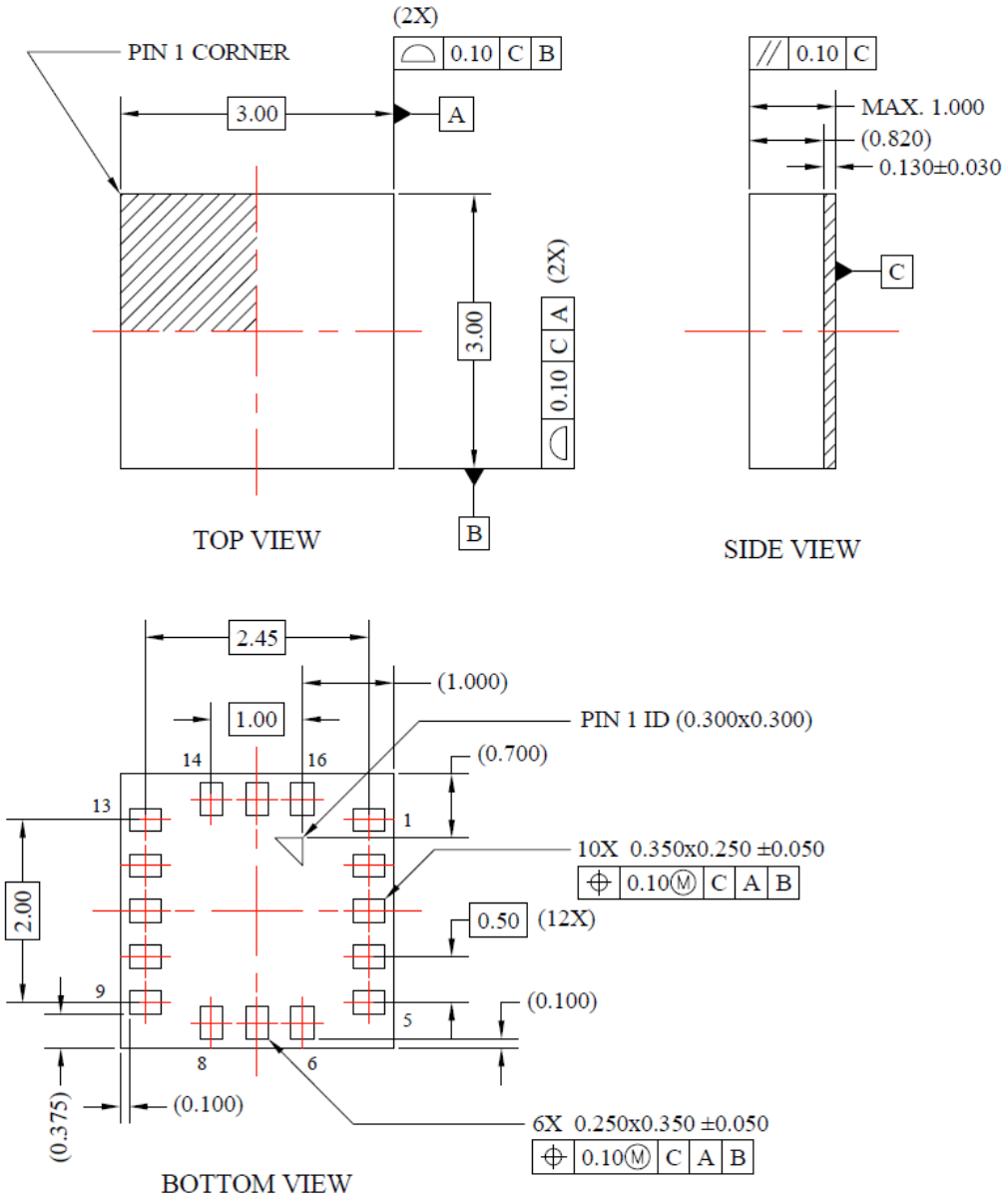


Figure 3. 16-pin 3 x 3 mm LGA package outline diagram with dimensions

Typical LGA packages expose metal traces on the package sides; so no solder material should be allowed to contact the package sides.

There may be variation in the location and shape of the “PIN 1 ID”, for various product packages with the same pin layout.

Solder Pad Layer Dimensions

The solder pocket is defined by dimensions of the metal layers behind the solder pad and the solder mask around the pad. There are two varieties of solder pad design in the Kionix 3x3 LGA.

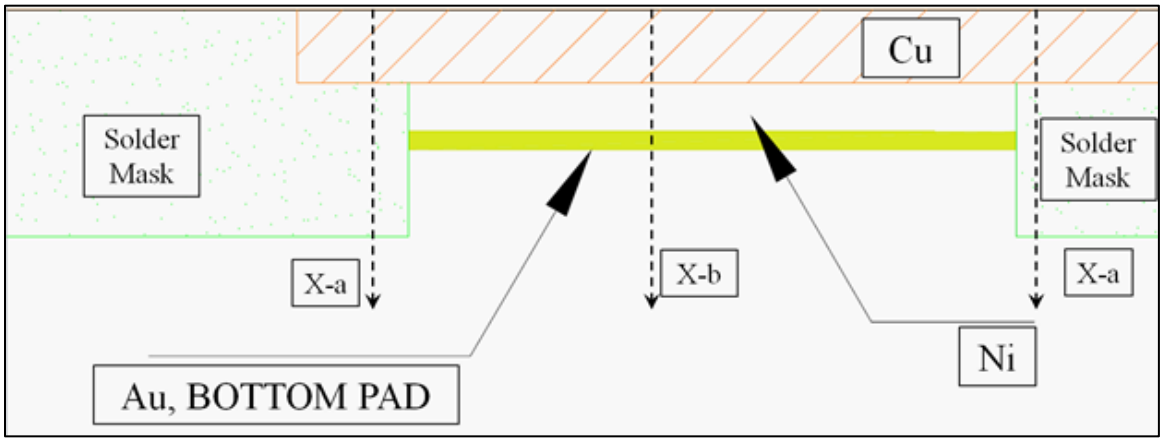


Table 1. For 3x3 products with 0.13mm substrates (KX023, KX23H, KXCJA)

Solder Mask (Cross Section "X-a")(µm)				Solder Pad (Cross section "X-b")(µm)			
Layer	Min	Nominal	Max	Layer	Min	Nominal	Max
S/M	10	20	30	NI	3	-	12
				Au	0.3	0.3	0.5*

* Absolute physical limit is 1.0 µm

Table 2. For 3x3 products with 0.21mm substrates (parts not in table 1)

Solder Mask (Cross Section "X-a")(µm)				Solder Pad (Cross section "X-b")(µm)			
Layer	Min	Nominal	Max	Layer	Min	Nominal	Max
S/M	15	30	45	NI	3	-	12
				Au	0.3	0.3	0.5*

* Absolute physical limit is 1.0 µm

LGA PCB Layout Recommendations

Given the above 3 x 3 mm package dimensions, the following guidelines are recommended:

The PCB should be designed with SMD (Solder Mask Defined) openings for the LGA lands. These openings should be an identical mirror image of the bottom pattern of the LGA package. The land size on the PCB should be 100% of the device land.

The pin 1 indicator triangle that is exposed on the LGA substrate does not need to be soldered to the PCB and should be left floating.

LGA Solder Stencil Guidelines

A laser-cut, stainless steel stencil with electro-polished trapezoidal walls is recommended.

Solder stencil thickness: 8-pin and 10-pin should be 0.125mm thick while the 16-pin should be 0.102mm thick.

Solder stencil opening: 8-pin and 10-pin should be 0.05mm smaller per side than corresponding pad openings while the 16-pin should be 0.25 x 0.35 mm.

If improved solder release is required, aperture walls can be trapezoidal and the corners rounded.

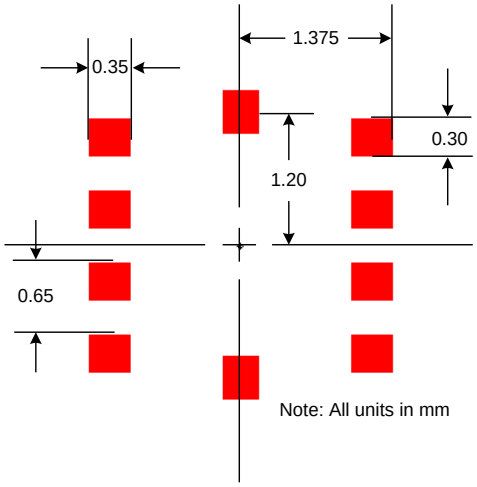


Figure 4. Example of a 10-pin 3 x 3 mm LGA solder stencil layout

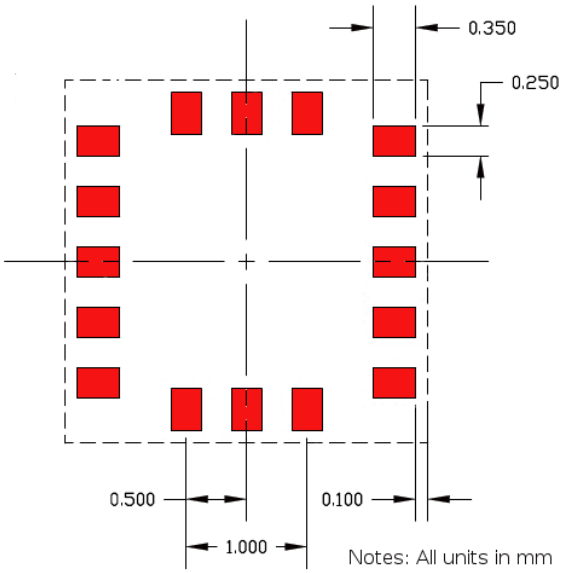


Figure 5. Example of a 16-pin 3 x 3 mm LGA solder stencil layout

PCB Via and Trace Placement

Vias are not needed for thermal dissipation, as our part doesn't generate much heat. Therefore, only electrical vias are needed. If vias are not in the land pads, capped, plugged, tented, un-capped or un-plugged vias can be used. To ensure optimal performance, vias and traces should not be placed on the top layer directly beneath the accelerometer. The following figures illustrate an example of proper PCB via and trace placement. Obviously, each product will present its own physical limitations for accelerometer placement and trace routing. Therefore, these guidelines are general in nature. Engineering judgment should be used to try to avoid placement directly beneath the accelerometer.

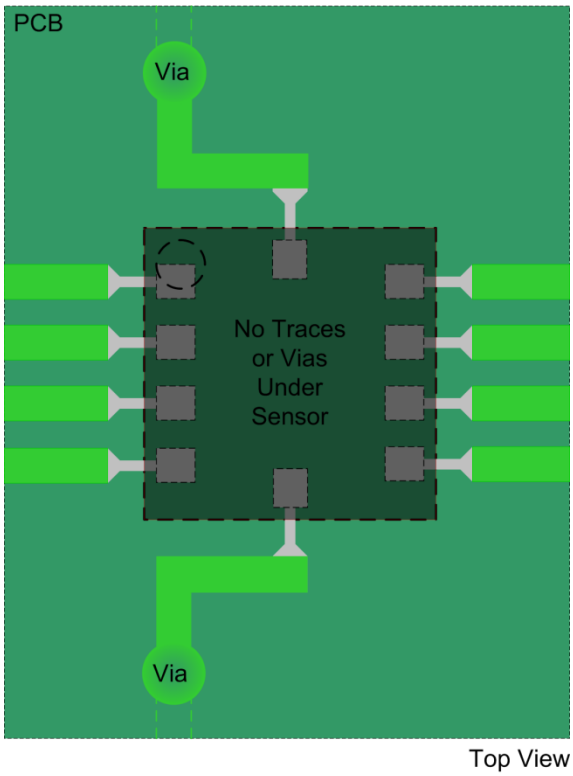


Figure 6. Via and Trace “Keepout” (Top Vew)

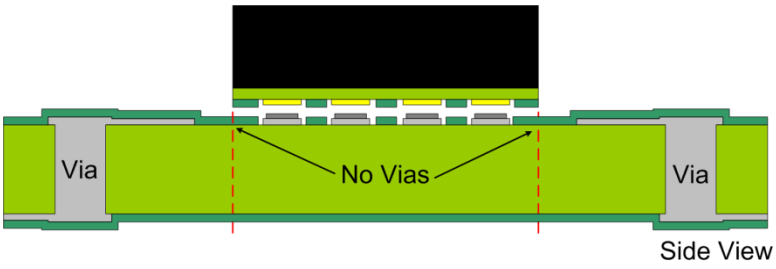


Figure 7. Via and Trace “Keepout” (Side View)

Tape and Reel Dimensions

The following section provides information on the tape and reel used for shipping Kionix's 3 x 3 mm LGA accelerometers.

Package	Tape Width	Component Pitch	Hole Pitch	Reel Diameter
LGA (3x3)	12mm	8mm	4mm	330mm

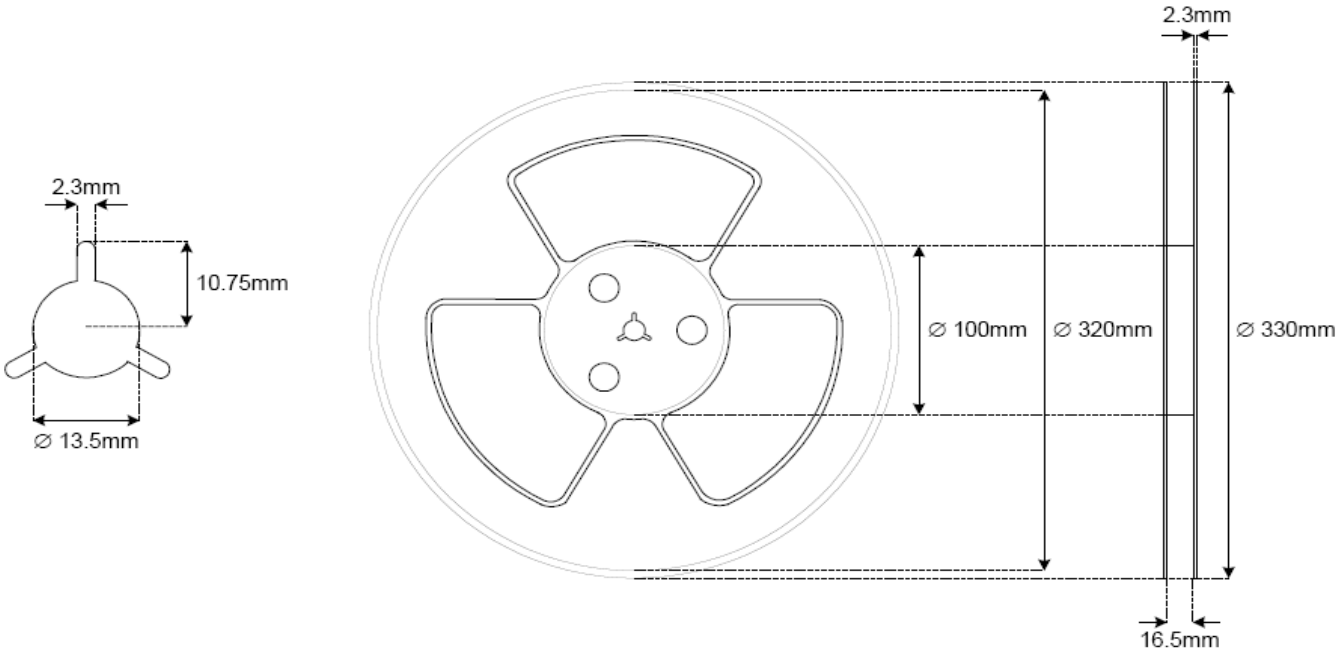
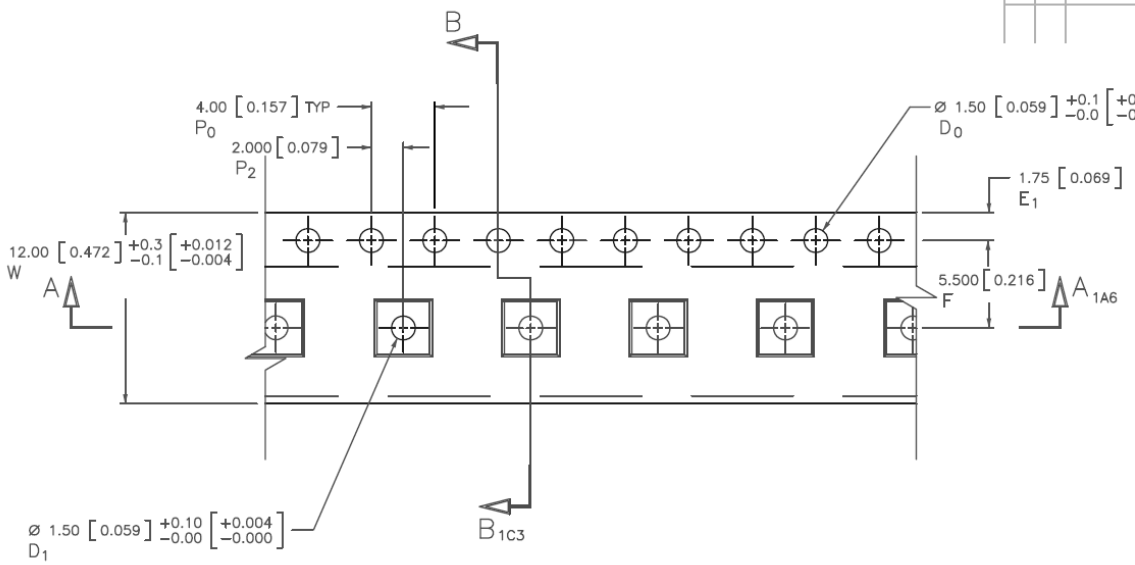
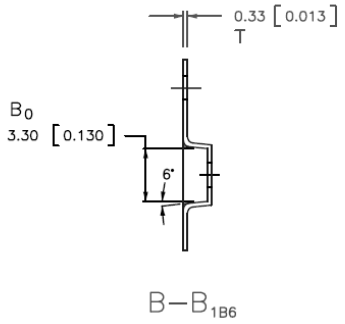


Figure 8. Dimensions of the reel

REVISION LOG							
ZONE	SYM	DESCRIPTION	DATE	APPROVAL	DATE	VALIDATION	DATE
	IR	INITIAL RELEASE	2/18/08	M. KEATTS	2/18/08	M. KEATTS	2/18/08





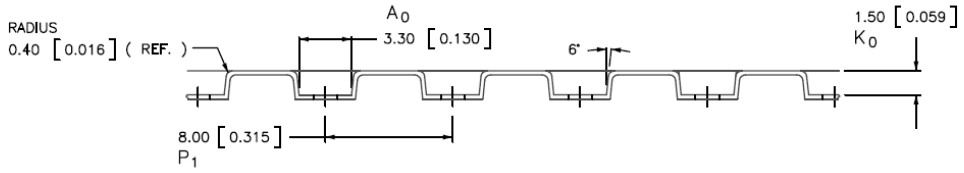


TABLE 1
POCKET SIZE
A_0 = 3.30mm [0.130]
B_0 = 3.30mm [0.130]
K_0 = 1.50mm [0.059]

NOTES:

1. ALL DIMENSIONS ARE MILLIMETERS [INCHES]
2. 10 SPROCKET HOLE PITCH CUMULATIVE TOLERANCE ± 0.2mm
3. MATERIAL 0.33 [0.013] CARBON POLYSTYRENE
4. POCKET POSITION RELATIVE TO SPROCKET HOLE MEASURED AS TRUE POSITION OF POCKET, NOT POCKET HOLE
5. 110 METERS / 22" REEL

PART CONFORMS TO E.I.A. STANDARDS

UNLESS OTHERWISE SPECIFIED	DESIGN	DATE	KEACO
DO NOT SCALE DRAWING	DRAWN J. DEREK WYATT	2/18/08	
ANGLES ±1°	CHECKED MARK KEATTS	2/18/08	TITLE: 12mm CARRIER TAPE
TOLERANCES .XX ± .10mm .XXX ± .05mm	VALIDATION	DATE	
	APPROVAL MARK KEATTS	2/18/08	SIZE D · DWG. NO. CT12–1300–01 · REV IR
			F–144 REV. A · SCALE: NONE · SHEET 1 OF 1

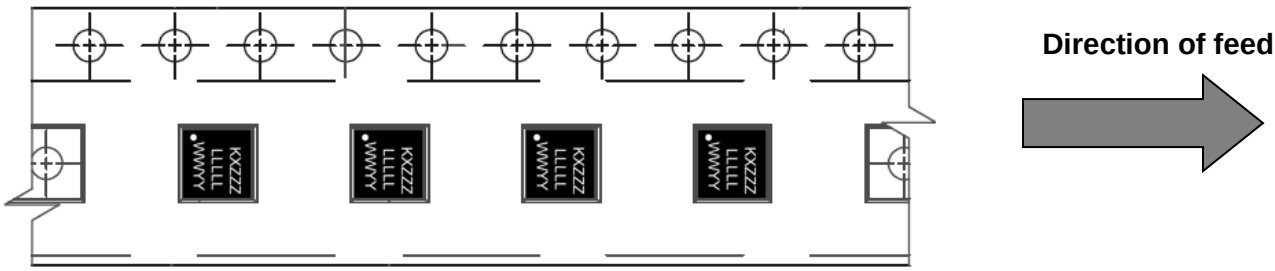


Figure 9. Orientation of the parts in the carrier tape and direction of feed

Revision History

Rev	Date	Description of Change
-	03-Jul-08	Initial release
2	29-Sep-11	Include 16 pin devices
3	3-Jan-12	Included 16 pin solder mask schematic
4	22-Mar-12	Added “Detail A” to illustrate metal lead dimension to Figures 2,3,4
5	12-June-14	Added Solder Pad Layer Dimensions
6	15-Oct-14	Added No solder on side of package recommendation.
7	07-May-15	Updated Solder Mask Defined feature drawings, removed 8 pin
8	10-July-15	Renamed the document