TDK InvenSense

IMU PCB Design and MEMS Assembly Guidelines

for

ICM/IAM/IIM-4xxxx, 2xxxx and MPU-6xxx Products

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1 INTRODUCTION

This document provides high-level PCB design, sensor mounting, and handling guidelines for TDK IMU Products, which incorporate a combination of gyroscopes and accelerometers. Each sensor has specific requirements to ensure the highest performance in a finished product. For a layout assessment of your design, including placement and estimated temperature disturbances, please contact TDK.

The TDK IMU Products discussed in this document consist of 3-axis MEMS gyroscopes and 3-axis MEMS accelerometers.

This document replaces AN-IVS-0002A-00 MEMS Motion Handling and Assembly Guide.

2 PCB DESIGN GUIDELINES

2.1 GENERAL PCB GUIDELINES

- Non-Solder Mask Defined pads (NSMD) are recommended more than Solder Mask Defined (SMD) pads.
- Traces connected to pads should be as symmetrical as possible.
- Avoiding vias, traces, and copper pour under all the layers below the IMU Products is recommended.
- Always place VDD decoupling capacitors as close as possible to the VDD pin. The VDD pin fanout trace should be thicker than 6 mil. The noise level on the VDD must meet datasheet specifications.
- Always place VDDIO decoupling capacitors as close as possible to the VDDIO pin. The VDDIO pin fanout trace should be thicker than 6 mil.
- VDD and VDDIO can be from same power supply source, but each should have its own decoupling capacitor.
- Keep VDD, VDDIO, serial communication lines away from any battery charge and from components and signals related to the DC-DC switching power regulator when doing the layout. The high energy of a power signal can generate a lot of noise or spikes.
- Provide solid GND connections for GND pin (pin6). Any nearby GND via must be bigger than 8 mil drill and 12 mil copper. Multiple nearby GND vias are recommended for solid GND connections. The GND pin fanout trace should be thicker than 6 mil.
- Reduce serial communication bus capacitance by avoiding long trace. Always balance the bus lines as close as possible to trace length.
- For high-speed interfaces, such as I^2C, I3C, and SPI, all clock and data traces should be routed with the same length and away from other high-speed traces. Power traces should also be routed away from high-speed signals.
- TDK InvenSense IMU Products provide configurations for output pad slew rate (drive strength) adjustment. User can adjust the output pad slew rate to optimize signal integrity on PCB board. Stronger drive strength is recommended for long PCB trace or long jump wire connections. Do not apply unnecessary strong drive strength to avoid over/under shoot on signal. We recommend adjusting the slew rate setting by using scope to monitor the signals.

2.2 PCB LAND PATTERN

For the solder mask of each pin, Non-Solder Mask Defined pads (NSMD) are recommended more than Solder Mask Defined (SMD) pads. NSMD contact pads have the solder mask pulled away from the solderable metallization. NSMD contact pads have several advantages over SMD pads. They provide a tighter tolerance for copper etching, provide a larger copper pad area, and allow the solder to anchor to the edges of the copper pads, which improves solder joint reliability.

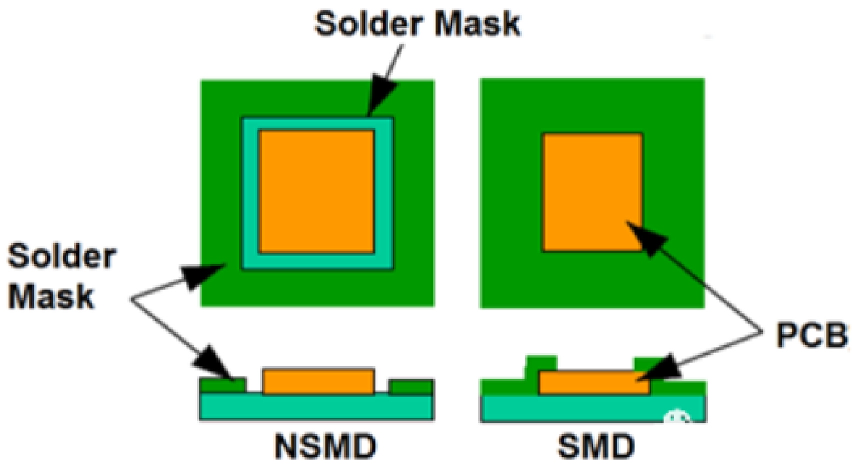
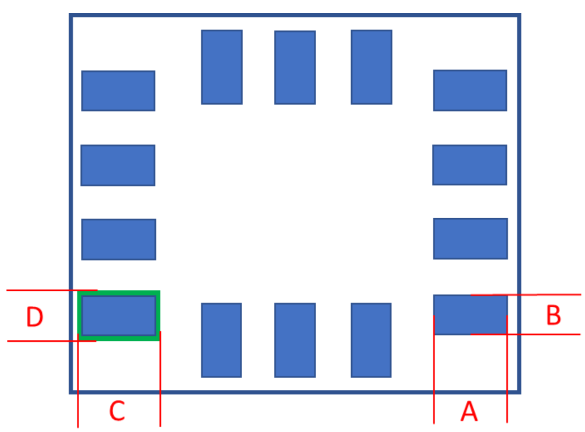


Figure 1. NSMD and SMD

To achieve optimal performance of MEMS motion devices, placing the solder mask below the MEMS component is not recommended. If this is not possible, placing the solder mask below the component will still work.

IMU Product package dimensions are provided in datasheets.

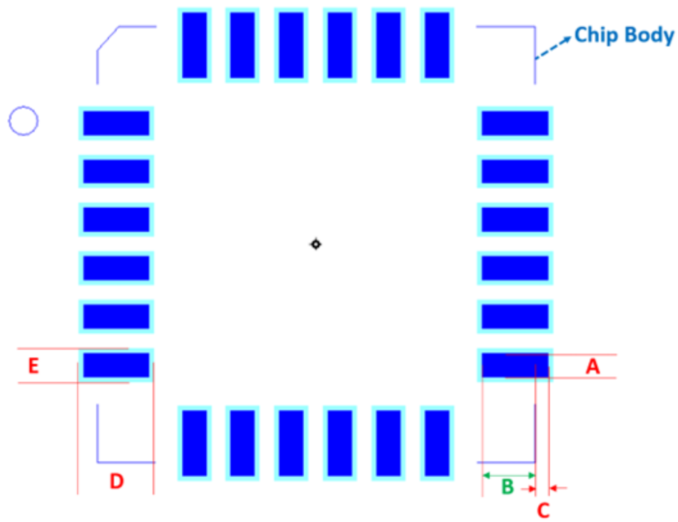
Figure 2 provides PCB land pattern recommendation for IMU Products in LGA package.



- A = PCB land length = LGA solder pin length
- B = PCB land width = LGA solder pin width
- C = Solder mask opening length = PCB land length + 0.1 mm
- D = Solder mask opening width = PCB land width + 0.1 mm

Figure 2. Recommended PCB Layout footprint for IMU Products in LGA package

Figure 3Figure 2 provides PCB land pattern recommendation for IMU Products in QFN and DQFN package.



- A = PCB land width = QFN solder pin width
- B = QFN solder pin length
- C = PCB land length extension = 0.15mm
- PCB land length = B + C
- D = Solder mask opening length = PCB land length + 0.1mm
- E = Solder mask opening width = PCB land width + 0.1mm

Figure 3. Recommended PCB Layout footprint for IMU Products in QFN and DQFN package

TDK InvenSense products have very low active and standby current consumption. The exposed center pad is not internally connected, and it is not required for heat sinking. The exposed center die pad must not be soldered to the PCB and must be left unconnected.

Please use the most recent revision of the datasheet for the device with which you are working. Dummy traces can be added on unused pins.

2.3 SOLDER PASTE PRINTING

Mechanical decoupling from the PCB to the sensor must be ensured to prevent any stress on the component. Contact from PCB resist and the package exposed pad must be avoided. Proper thickness definition for both solder paste and copper help set proper clearance below the package. No solder paste needs to be disposed below the exposed pad.

Solder paste disposition should be done by stencil screening. In standard conditions, TDK InvenSense recommends using a stencil opening to land ration of 90%. Stencil walls should be tapered to produce uniform release of the paste when the stencil is removed from the PCB. Stencil thickness should be at least 100 µm in respect to standard area ratio design rule.

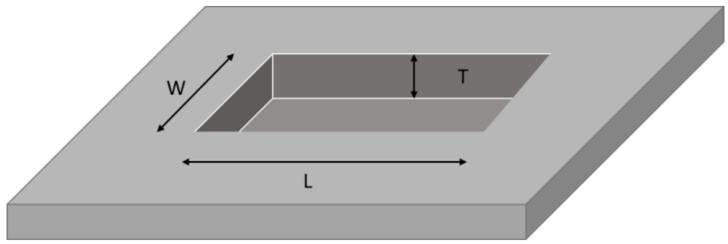


Figure 4. Stencil opening dimensions

Generic best practices for stencil design should be followed:

- Aspect Ratio
    - The width of aperture / thickness = W/T
    - The lowest acceptable aspect ratio is 1.5
- Area Ratio
    - Surface area of aperture / surface area of the aperture walls = (L x W)/ (2 x (L+W) x T)
    - The lowest acceptable area ratio is 0.66

Maximizing symmetry and balance for pad connection will help with component self-alignment and will lead to better control of solder paste reduction after reflowing. At the end of the soldering process, the solder paste must be as uniform as possible to avoid unbalanced stress on the component.

Solder paste volume to be printed is greater than the final solder joint volume because solder paste decreases during reflow. Solder paste volume reduction factor is typically between 0.45 and 0.55. Being the pad area constant, in the worst case, the solder paste thickness can be assumed to be about 45% after the reflow:

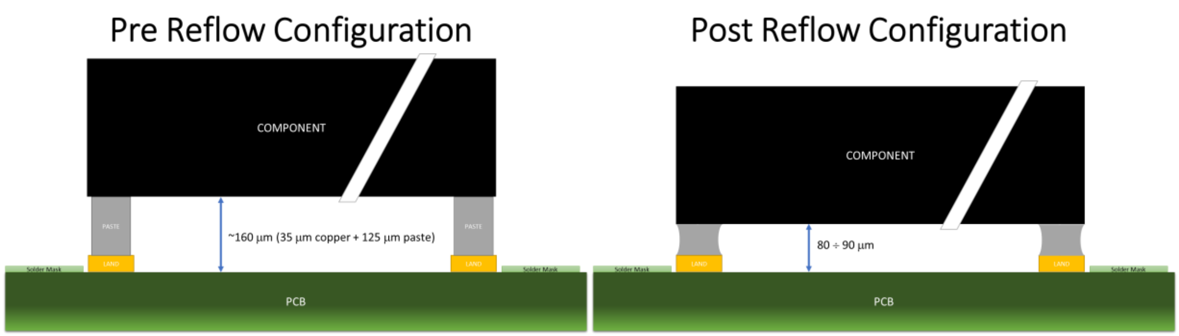


Figure 5. Pre and post reflow configuration

2.4 ROUTING

Traces connected to pads should be as symmetrical as possible. Symmetry and balance for pad traces will improve component self-alignment and lead to better control of solder paste reduction after the reflow process.

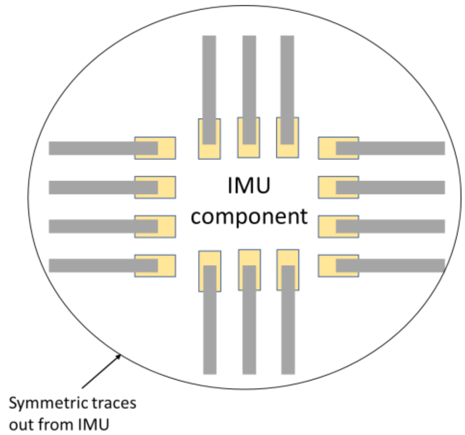


Figure 6. Symmetrical trace out

For high-speed interfaces, such as I^{2}C, I3C, and SPI, all clock and data traces should be routed with the same length, and away from other high-speed traces. Power traces should also be routed away from high-speed signals. Keep VDD, VDDIO, serial communication lines away from any battery charge and DC-DC switching power regulator related components and signals when doing the layout. The high energy of a power signal can generate a lot of noise or spikes.

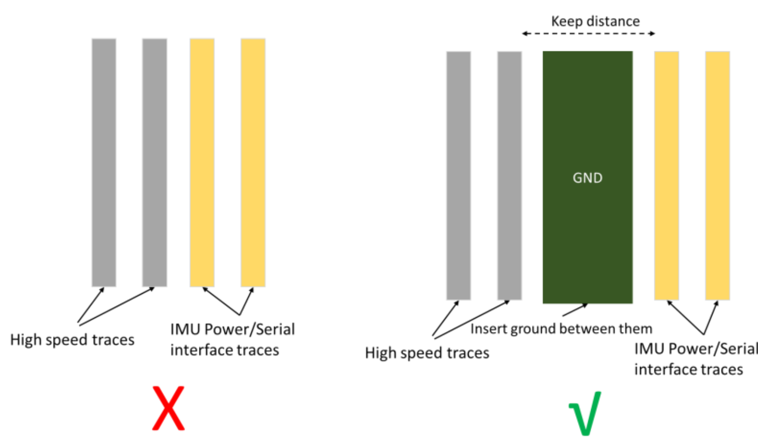


Figure 7. Keep distance from high-speed and high current traces

We recommend 6mil or thicker traces for a 0.5 oz or 1 oz copper PCB. Provide a solid ground return path, with traces 10mil or thicker for a 0.5 oz or 1 oz copper PCB. Do not use small vias for power and GND traces.

Trace, via, and filled copper are not allowed under the IMU chip directly, as they can cause elevation changes.

Do not place vias within the pad outline because vias and their related plating materials can contribute to an orientation offset and non-uniform mechanical package stress.

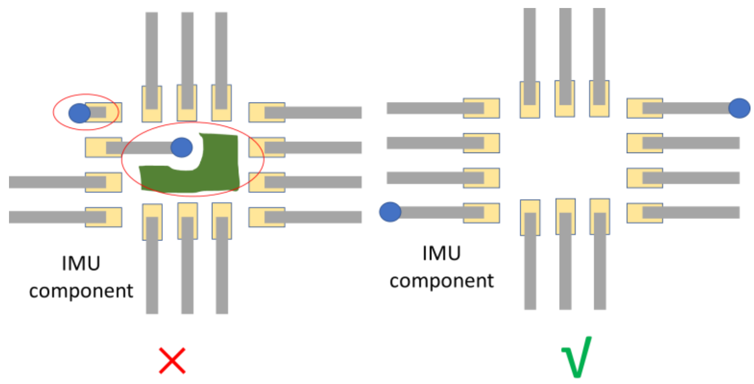


Figure 8. Avoid traces, vias and copper pour below IMUs

NC (No Connect) pins should be soldered to the board for mechanical stability, but those pads on the board should not be connected electrically.

2.5 FLEX PCB

Flex PCB should be avoided for MEMS sensor parts. If flex PCB must be used, a stiffener must be provided on the opposite side of the flex PCB. The thickness of the stiffener depends on its material. The goal is to prevent the flex PCB from bending, which will generate mechanical stress to MEMS.

2.6 MULTIPLE IMUS ON SAME PCB

Due to the nature of MEMS sensor operation, when having multiple IMUs on same PCB there will be mechanical force interferences between them. The interference can cause sensor noise increase and sensor offset change.
It is strongly recommended that not placing two or more IMUs on same PCB board.
When it is necessary to have multiple IMUs on same PCB, please try to cut their mechanical force transaction path as much as possible. Cutting PCB slot is one of many methods recommended here. The below drawing shows a cutting concept for board design reference.

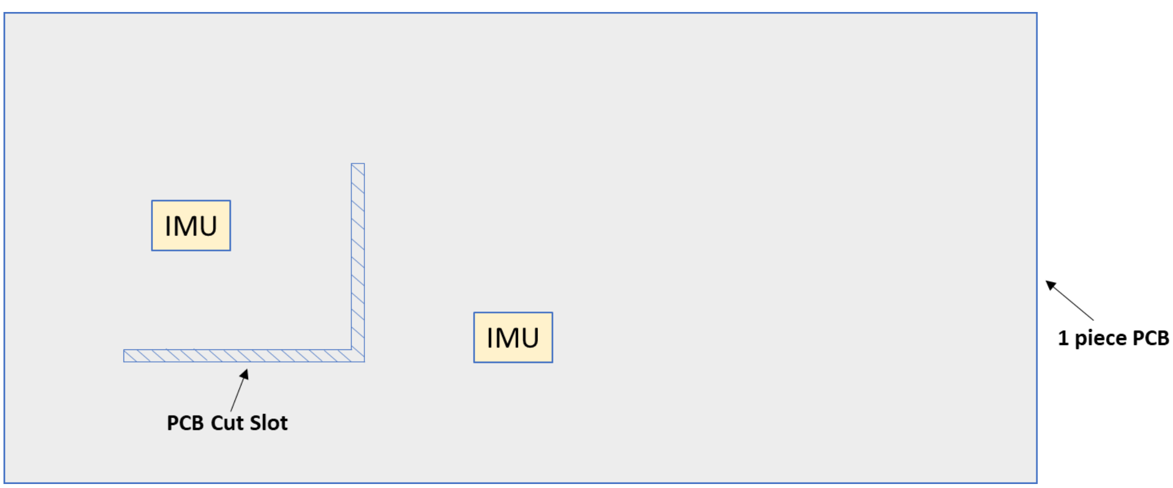


Figure 9 PCB Cut Slot for Multiple IMUs on the same PCB

3 PCB PLACEMENT AND BOARD MOUNTING GUIDELINES

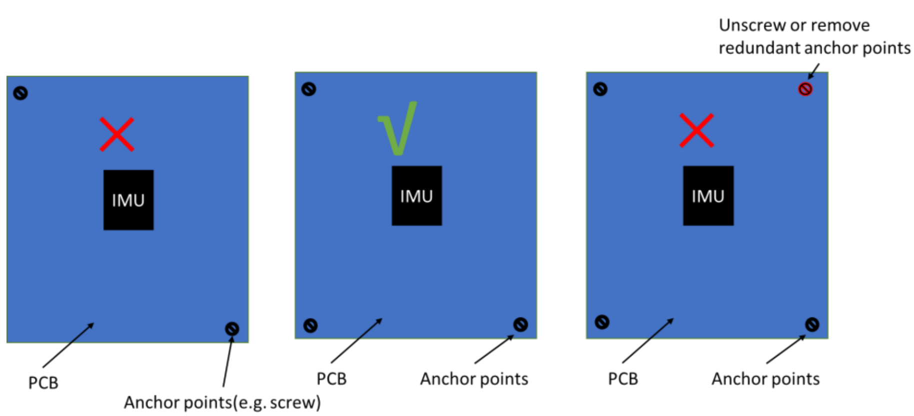
MEMS accelerometer and gyroscope sensors are general high accuracy motion measurement devices. TDK IMU Products are designed with precision, efficiency, and mechanical robustness. However, to achieve highest possible accuracy, the following recommendations should be considered when mounting IMU Products on a printed circuit board (PCB).

3.1 GENERAL RECOMMENDATIONS

- PCB should be installed with 3 anchor points; 2 anchor points or more than 3 is not recommended.
- The IMU Products should be placed on a location that minimizes mechanical stress. Keeping away from the connection of two anchor points is strongly recommended.
- The IMU Products should be kept a reasonable distance to the critical points: e.g., away from any fixed mounting location, screw hole, large insertion components, such as buttons, shielding boxes, connectors, etc.
- To avoid dynamic mechanical stress on pogo pins, placing the sensor direct under or next to pogo pins is not recommended.
- Keeping a reasonable distance between sensor and heat sources, which may include processors, power management circuitry, or high current devices, is recommended.
- Direct epoxy contact on the IMU Products is forbidden. The epoxy-seal should be placed a reasonable distance away from sensor.
- Keep a reasonable distance from the IMU Products to the edge of PCB or bridges for PCB separation by router. Deflection from a routing drill or saw can damage the MEMS device.
- Keeping a reasonable distance from vibration sources, such as speakers, vibration/haptic motors, fans, etc. is recommended.
- The MEMS device can be damaged when shock level is over the datasheet specification. Shock or impact should be avoided in manufacturing flow or device assembly process. It is not recommended to mount the sensor in areas where resonant amplitudes of the PCB are likely.

3.2 PCB ANCHOR POINTS

In theory, an ideal flat plane is determined by 3 anchor points. Two anchor points or more than 3 anchor points should be avoided. If 3 or more anchor points are used, removing any redundant PCB anchor points is recommended. Figure 10 shows different anchor scenarios. Two anchor points are expected to be unstable. The



PCB may shake from any movement or vibration and is expected to have PCB bending. Four or more anchors are assumed not exactly in plane. Removing the redundant anchor point can minimize mechanical stress significantly.

Figure 10. PCB anchor points

3.3 MECHANICAL STRESS ON PCB

Keeping a reasonable distance to the mechanical stress maximum point is recommended. As shown in Figure 11, keep away from the intersection of diagonal lines. Keeping away from any PCB anchors is strongly recommended. Keeping a distance larger than 3 mm to any PCB anchor is recommended.

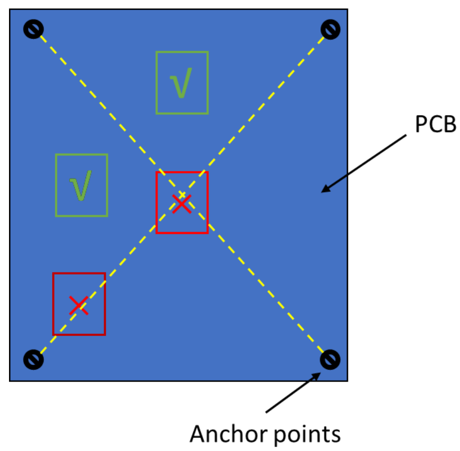


Figure 11. Keep away from anchor points and the intersection of anchor points

3.4 PCB WITH OVER HANGING BEAM

Avoid placing the IMU chip on the end of long hanging beam or the PCB with large span. The vibrations are very likely to be happen there. Keeping a reasonable distance is very important to avoid unexpected behavior from the MEMS component.

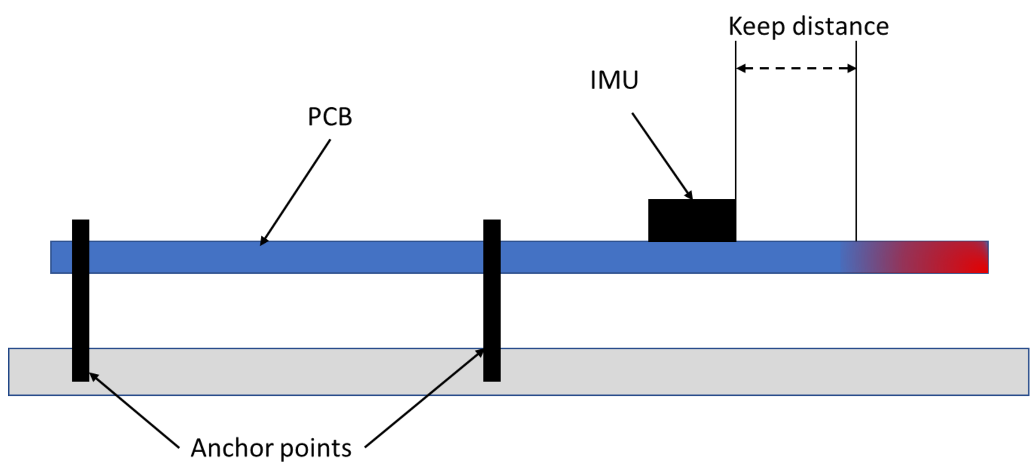


Figure 12. Avoid place IMU Products on long hanging beam

3.5 BUTTONS, POGO PINS, SHIELDING BOXES, CONNECTS, ETC.

Do not place connectors or test points for pogo pins on the PCB surface below the IMU Product location, as shown in Figure 13. Shock from snapping the connectors and pressure from the pogo pin during functional test on a production line may damage the MEMS part.

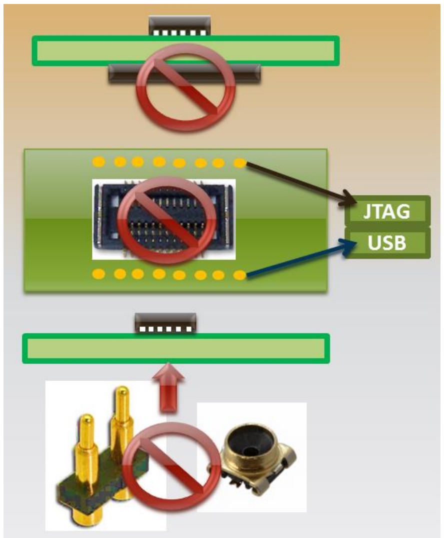


Figure 13. Avoid connectors directly behind the board

3.6 THERMAL REQUIREMENTS

Keeping a distance between the heat sources and IMU device is recommended to achieve a higher accuracy measurement. Do not place any heat source component on the opposite side of PCB under IMU device.

The heat sources may include processors, power management circuitry, or high current devices. The temperature gradient across the IMU device, especially the top to bottom gradient should be minimized for best measurement results. Refer to Figure 14.

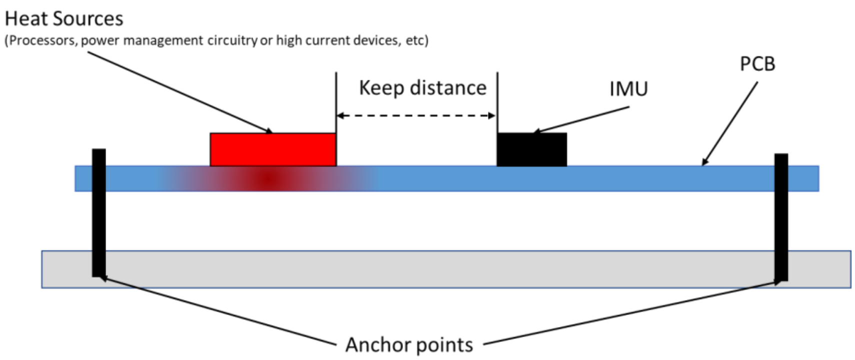


Figure 14. Keep distance from heat sources

3.7 SPEAKERS, FANS, ETC.

Moving parts that cause vibration and are not intended to be measured, such as speakers, vibration/haptic motors, fans, etc. (Figure 15), should be mechanically isolated from IMUs.

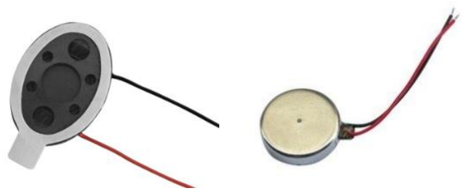


Figure 15. Speaker and Tactile Vibrations can be interpreted as Noise by the IMUs

Active signals may harmonically couple with the gyro MEMS devices, compromising gyro response. TDK IMU gyroscopic sensors operate at drive frequencies 25 kHz ~ 29 kHz. To avoid harmonic coupling, do not route active signals directly below or near the package. If the IMU device is stacked under an adjacent PCB board, design a ground plane to shield the IMU device from the adjacent PCB.

Electrical sources, such as a switched-mode power supply (SMPS) as shown Figure 16, can cause high frequency vibration. SMPS (switched mode power supply) with switching noise below 150 kHz (including Harmonics) can reduce device performance.

As mentioned in section 3.4, in addition to unwanted IMU device vibration output, mechanical vibration can damage MEMS if the vibration frequency matches the MEMS resonant frequency.

Place any acceleration or vibration sources as far away as possible from MEMS devices. If placement is uncertain, consult the local FAE to provide a more detailed analysis.

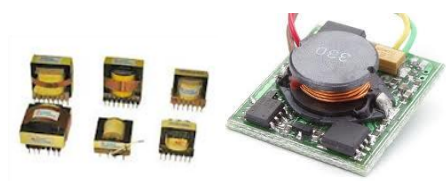


Figure 16. Keep distance to high frequency Electrical sources

3.8 SHOCK AND VIBRATION

The MEMS device can be damaged when shock level is over datasheet specification. Shock or impact should be avoided in manufacturing flow or device assembly process.

During the design phase the IMU device should be placed in the location where it will receive the least amount of shock possible. Various measurements can be taken to reduce shock transfer, such as location of the IMU device, shock absorbing washers/gaskets for screw points, and shock absorbing foam pads. From a systems level, the placement of other PCBs/Modules/Sub-systems should be placed away from the top/bottom of where the IMU device is located, to prevent possible contact.

If there are concerns regarding a system level design, the systems should be tested with the IMU device as a whole system or tested beforehand and double checked to ensure there is no damage to the IMU device.

MEMS gyro and accel have multiple internal resonance points. System level designs should avoid vibrations with the same frequency of IMU device resonances. Please consult TDK technical support for each device's resonant frequency.

The TDK InvenSense IMU gyroscope sensors operate at specific drive frequencies. Please refer to each part's datasheet for the exact drive frequency number. Any vibration within these frequency ranges will cause extra gyro noise or even damage the gyro.

The most common PCB board level vibration source is from power circuits such as a wireless charger, and buck and boost power regulation circuits. Powered devices that may generate acceleration or vibration to the MEMS structures can cause damage to MEMS devices. Examples of such components are inductors, capacitors, PMIC, haptic motors, speakers, etc.

The shock experienced from a PCB level versus device level may and will be different. To attain the most accurate data reading, the measurements should be taken from a PCB level as close to the IMU device as possible. If there is a source of acceleration or vibration in the vicinity of the MEMS device, we recommend testing on the PCB with a vibrometer to confirm that the MEMS device is not being excited by any resonance frequency.

3.9 EPOXY-SEAL AND OTHER MATERIAL

Please make sure that the sensor is not partially or fully covered and not in contact with any epoxy coating or other material. Direct contact with the sensor should be avoided so that the sensor would not show any unexpected output because of this. The contact material includes any type of material.

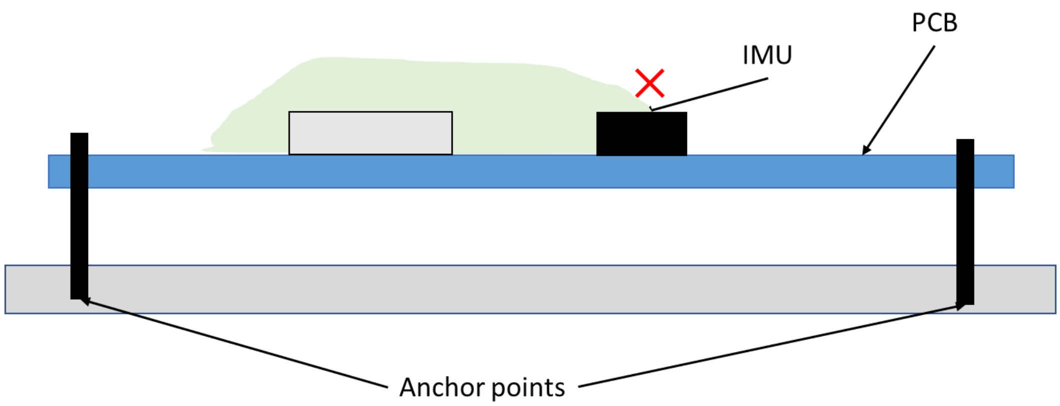


Figure 17. Keep distance from epoxy coating

3.10 EDGE OF PCB

Keep the IMU device away from the edge of the PCB or bridges for PCB separation by router see picture below) Deflection from a routing drill or saw can damage the MEMS device. Similarly, dull router bits and saw blades can cause excessive mechanical vibration, which should be avoided. Do not snap apart panelized boards, since snapping apart the PCB boards may introduce severe bending forces and mechanical shock, which may damage the IMU device.

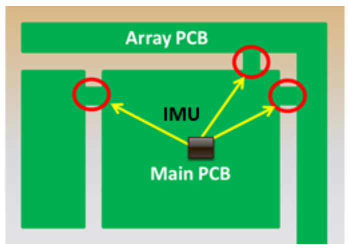


Figure 18. Panelized PCB Bridges

4 IMU PRODUCTS HANDLING GUIDELINE

4.1 MEMS HANDLING INSTRUCTIONS

Unlike conventional IC products in similar packages, MEMS devices contain moving micromechanical structures. Therefore, MEMS devices require different handling precautions than conventional ICs prior to mounting onto PCBs.

All TDK InvenSense IMU products react negatively to Helium gas.

TDK InvenSense products have been qualified to an unpowered shock tolerance of 10,000g or 20,000g. Information for each component is available in the corresponding product datasheet. Furthermore, the products are shipped in cushioned tape and reel packing (ref.: EIA-481) to protect them from potential damage induced by abnormal handling and shipping.

- Do not drop individually packaged sensors or trays of sensors. Components placed in trays could be subject to excessive *g*-forces and stress.
- PCBs that incorporate mounted sensors should not be separated by manually snapping them apart. This could create excessive *g*-forces and stress.
- Do not clean MEMS sensors in ultrasonic baths. Ultrasonic baths can induce MEMS damage if the bath energy causes excessive drive motion through resonant frequency coupling.
- Do not open and remove MEMS devices from the moisture barrier bag until you are ready to use them. The moisture barrier bag provides protection to the MEMS sensors during storage and transfer.
- Do not use any devices that are dropped inadvertently during handling.

4.2 IMU PRODUCTS ASSEMBLY GUIDELINES

Any material used in the surface mount assembly process of the MEMS gyroscope should be free of restricted RoHS elements or compounds. Pb-free solders should be used for assembly.

Although the sensor products can withstand up to three reflow soldering cycles, it is strongly recommended to avoid multiple reflow steps for the MEMS component. The PCB side where the MEMS is placed must be soldered the last.

Use a typical pick-and-place machine with reflow equipment like an oven. Avoid any manual soldering process. Machines should be set to the lowest and slowest pick and place operations, as much possible, to help minimize any shock impact to the device.

The sensor has been designed and tested to survive shock up to the Absolute Maximum Rating (AMR) reported into the datasheet. Shock above the AMR could damage the MEMS structure. Parts subjected to shock greater than AMR must be discarded.

4.3 PICK AND PLACE GUIDELINES

Pick and place (PnP) equipment key factors to be considered to minimize mechanical stress on MEMS structures are the following:

- Nozzle gap to the package upper surface
- Nozzle shape (larger inner diameter preferred)
- Nozzle material
- Air vacuum release pressure level

4.3.1 Target parameters

The following table summarizes the best choice of the above listed parameters for some of the packages available for TDK-InvenSense IMUs.

PnP equipment parameter		Package size	
		4.5x4.5x1.1 mm	3x3x0.75 mm
Nozzle height(1)		≤0.15 mm	≤0.15 mm
Nozzle material		Plastic or rubber	Plastic or rubber
Nozzle shape(1)(2)	Outer diameter	≥3.95 mm	≥2.8 mm
	Inner diameter	≥2.6 mm	≥1.5 mm
Pick-up vacuum pressure		≥ -64 kPa	≥ -64 kPa
Release air pressure		≤1.3 kPa	≤1.3 kPa

Table 1. PnP equipment recommended parameters

Notes:

1. tolerances are included (example for nozzle height ≤0.15 mm: 0.1mm ±0.05mm)
2. preferred square shape - limited to package size

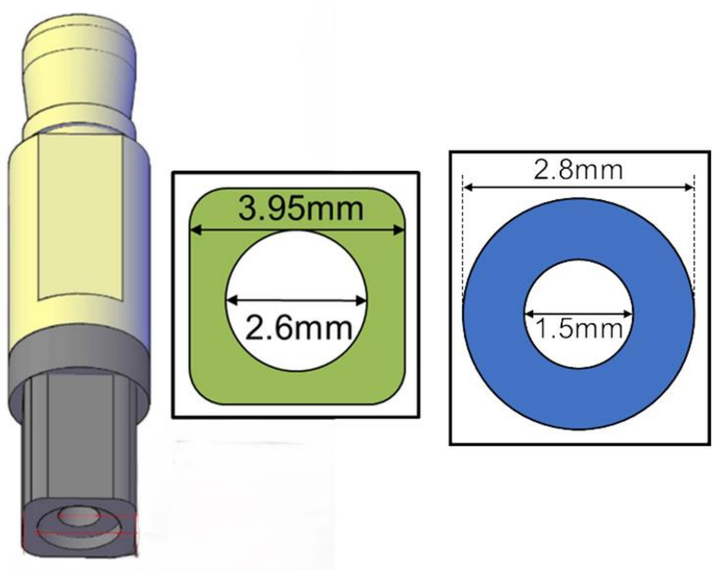


Figure 19. Examples of nozzle designs, reference for different packages

4.4 ESD CONSIDERATIONS

Establish and use (Electrostatic Damage) ESD-safe handling precautions when unpacking and handling ESD-sensitive devices.

- Store ESD-sensitive devices in ESD-safe containers until ready for use. The tape and reel moisture-sealed bag is an ESD approved barrier. The best practice is to keep the units in the original moisture sealed bags until ready for assembly.
- TDK InvenSense products are qualified to meet HBM (Human Body Model) 2000V and CDM (Charged Device Model) 500V. Restrict all device handling to ESD protected work areas that measure less than 200V static charge. Ensure that all workstations and personnel are properly grounded to prevent ESD.

4.5 REFLOW SPECIFICATION

TDK InvenSense products are qualified in accordance with IPC/JEDEC J-STD-020F. This standard classifies proper packaging, storage, and handling to avoid subsequent thermal and mechanical damage during the solder-reflow attachment phase of PCB assembly.

The peak-solder reflow temperature for package qualification is 260°C maximum for IMU products. The TDK InvenSense products can withstand up to three reflow soldering cycles.

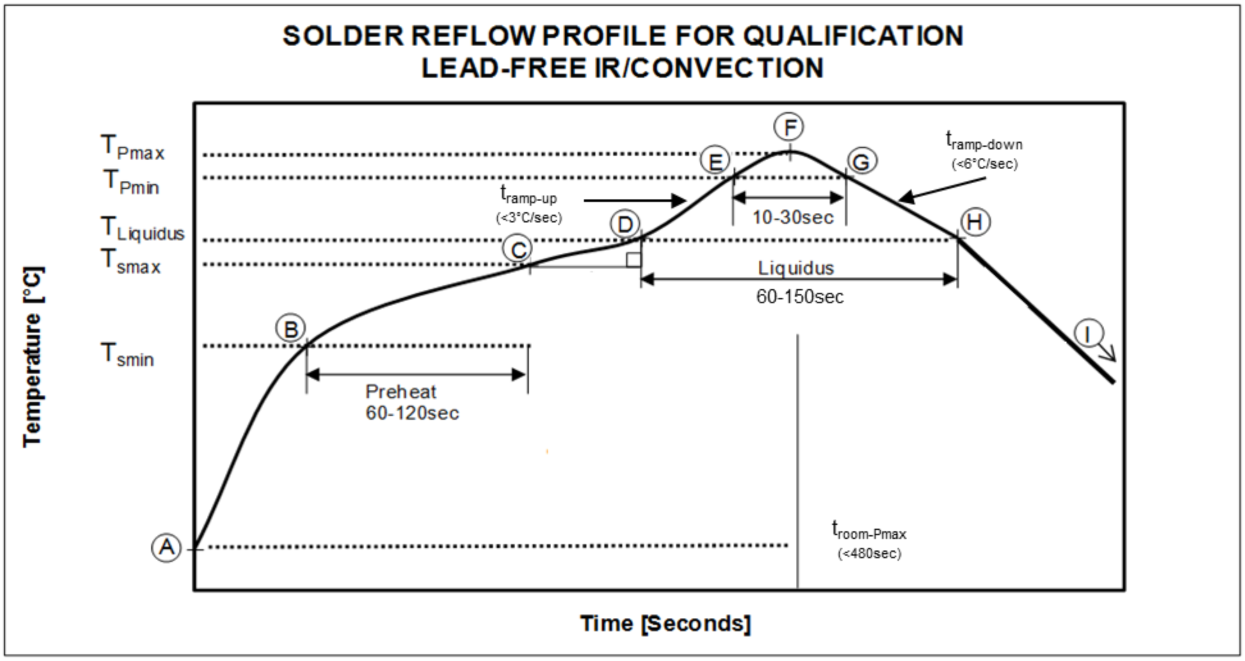


Figure 20. Solder Reflow Profile for Qualification

Step	Setting	CONSTRAINTS		
		Temp (°C)	Time (sec)	Max. Rate (°C/sec)
A	T_{room}	25		
B	T_{Smin}	150		
C	T_{Smax}	200	$60 < t_{BC} < 120$	
D	$T_{Liquidus}$	217		$r_{(TLiquidus\text{-}TPmax)} < 3$
E	T_{Pmin}	255		$r_{(TLiquidus\text{-}TPmax)} < 3$
F	T_{Pmax}	260	$t_{AF} < 480$	$r_{(TLiquidus\text{-}TPmax)} < 3$
G	T_{Pmin}	255	$10 < t_{EG} < 30$	$r_{(TPmax\text{-}TLiquidus)} < 6$
H	$T_{Liquidus}$	217	$60 < t_{DH} < 150$	
I	T_{room}	25		

Table 2. Temperature Set Points Corresponding to Reflow Profile Above

Notes: Customers must never exceed the Classification temperature (T_{Pmax} = 260°C).
All temperatures refer to the topside of the package, as measured on the package body surface.

4.6 REWORK PROCEDURE

If a MEMS component is needed to be removed from PCB for analysis or other purpose and replaced by a new one, care should be taken during removal and installation of the component, so that no damages are added to the component and PCB.

The general process for rework using a rework system consists of the following steps:

(1) Prepare the board for rework.

(2) Apply low-solids content liquid flux along one edge of the component; then tip the board slightly to flow flux under component.

(3) Preheat the board from both top and bottom sides (100°C to 120°C, 60s to 150s) to prevent warping the PCB.

(4) Then heat the board to (240°C - 255°C) to remove the component; use a shield to apply the heat to the component from above, while isolating other adjacent components from the heat.

(5) Prepare the site for installation of a replacement component by cleaning the site thoroughly. Remove excessive solder from the pads. Level the pad, clean the site thoroughly, inspect for damage to solder mask or pads, apply flux or solder paste.

(6) Install the replacement component by locally heating the component to (240°C - 255°C). The installing process needs to be finished in 20s.

(7) Clean the assembly, visually inspect, and X-ray for voiding and solder bridges;

(8) Perform full functional test.

The following recommendations are intended to reduce the chances of damaging a component during removal: Temperature profile – Hot air heat transfer to the package and PCB is strongly recommended. Typically, heating nozzles are used to increase the temperature locally. Temperature and air flow for heating the device should be controlled. The maximum temperature of the package body must not exceed the maximum allowed temperature for reflow soldering (260 °C).

It is important to minimize the chance of overheating neighboring devices during the removal or installation of the component. A heat shield should be utilized to protect adjacent components on the PCB.

4.7 STORAGE SPECIFICATIONS

TDK InvenSense products conform to the storage specifications of IPC/JEDEC J-STD-020 Moisture Sensitivity Level (MSL) 1.

Calculated shelf-life in moisture-sealed bag	12 months -- Storage Conditions: <40°C and <90% RH
After opening moisture-sealed bag	Unlimited hours -- Storage Conditions: Ambient ≤30°C at 85%RH

TDK InvenSense products conform to the storage specifications of IPC/JEDEC J-STD-020 Moisture Sensitivity Level (MSL) 3.

Calculated shelf-life in moisture-sealed bag	12 months -- Storage Conditions: <40°C and <90% RH
After opening moisture-sealed bag	168 hours -- Storage Conditions: Ambient ≤30°C at 60%RH

TDK InvenSense products conform to the storage specifications of IPC/JEDEC J-STD-020 Moisture Sensitivity Level (MSL) 5.

Calculated shelf-life in moisture-sealed bag	12 months -- Storage Conditions: <40°C and <90% RH
After opening moisture-sealed bag	48 hours -- Storage Conditions: Ambient ≤30°C at 60%RH

Notes: A attain the MSL rating for motion-based products, please refer to the qualification report for the details.

4.8 TAPE AND REEL HANDLING INSTRUCTIONS

TDK InvenSense IMU devices are shipped in tape and reels. They are packaged to protect them from potential damage induced by normal handling and shipping. These are handling guidelines for the tape and reels populated with MEM's motion devices:

- Tape and reels (with devices) should not be dropped at any time or un-reeled manually.
- Precautions should be taken to minimize the amount of vibration that tape and reels (with devices) are subjected to while in pick and place machines.
- The slowest settings possible should be used on pick and place machines during the SMT process.
- Tape and reels should be kept in packaging for as long as possible, until ready for use on pick and place machines
- Any carts used for internal transportation of tape and reels (with devices) should be padded with bumpers and have shock absorbing features.

4.9 TAPE & REEL SPECIFICATION

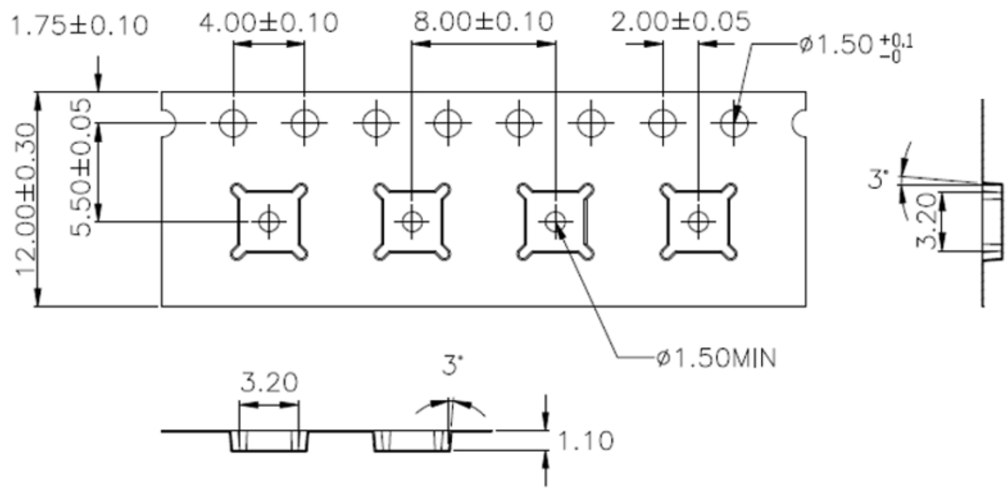


Figure 21. 3 x 3 mm² Tape Dimensions (TYPE I) (for package height 0.9 mm and 1.0 mm)

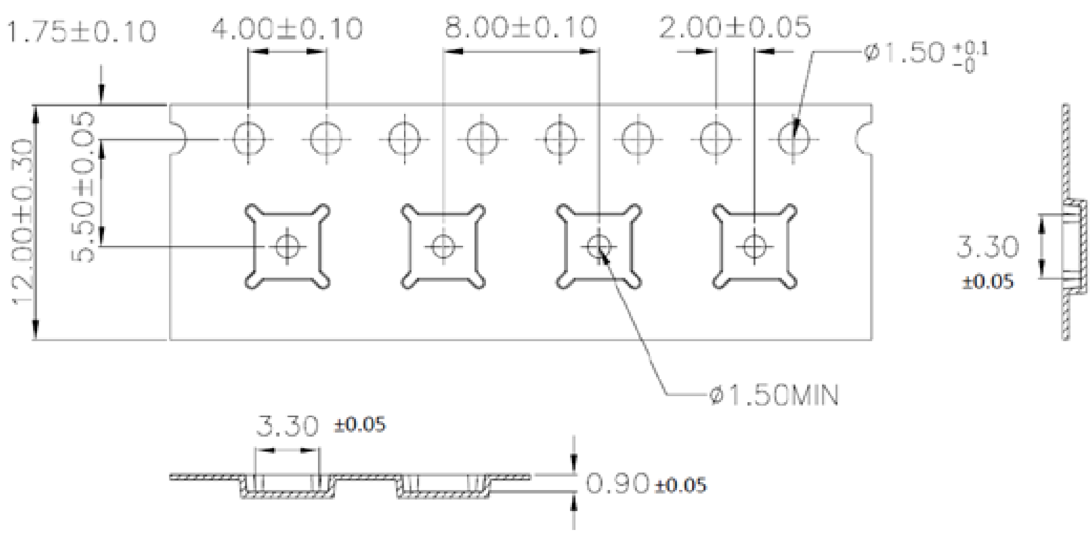


Figure 22. 3 x 3 mm² Tape Dimensions (TYPE II) (for package height 0.75 mm)

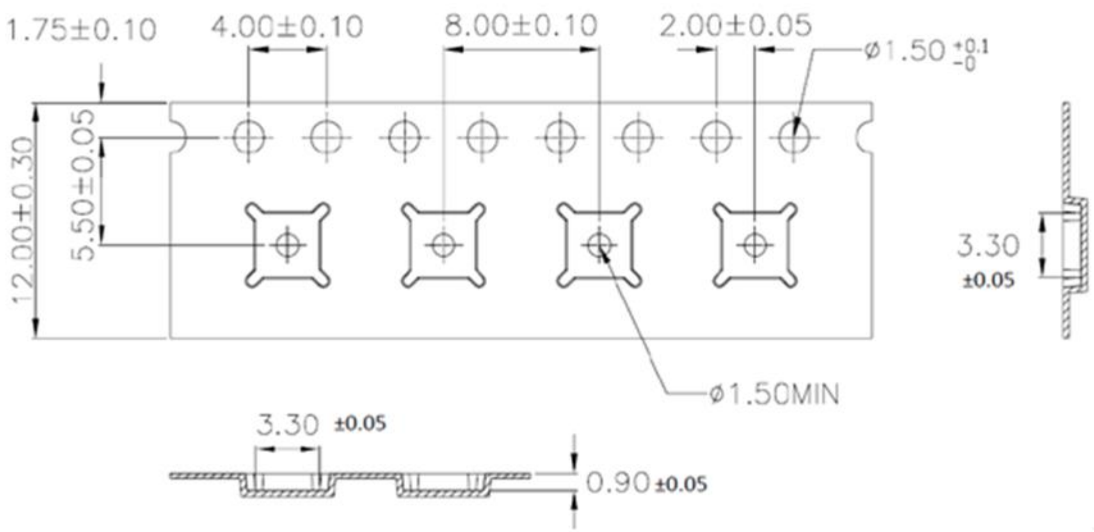


Figure 23. 2.5 x 3 mm² Tape Dimensions (TYPE I)

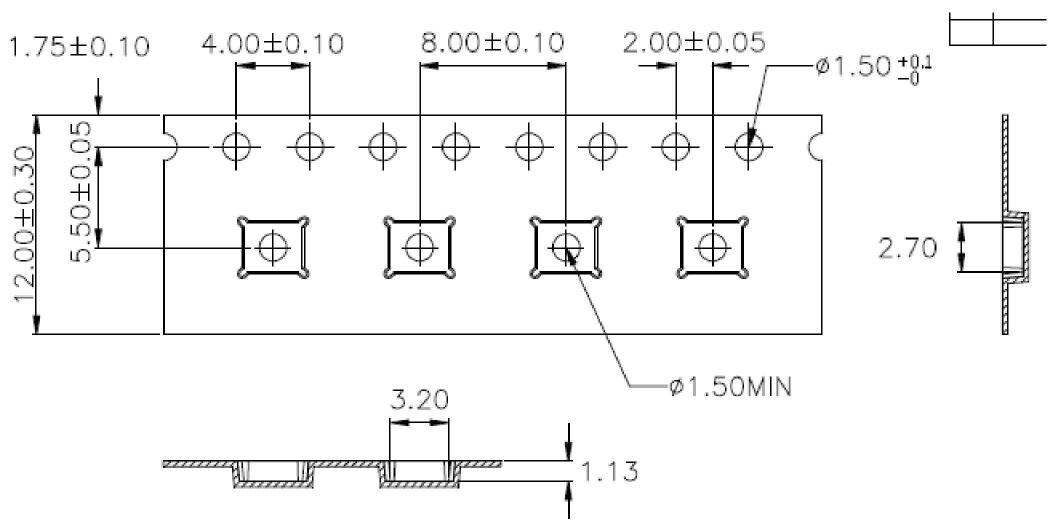


Figure 24. 2.5 x 3 mm² Tape Dimensions (TYPE II) (for special request only)

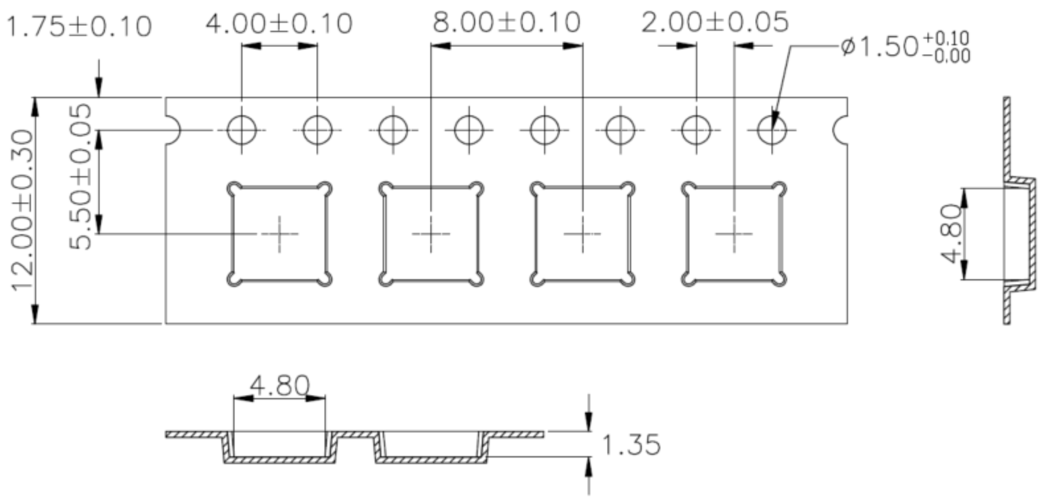


Figure 25. 4.5 x 4.5 mm² Tape Dimension (for package height 1.1 mm)

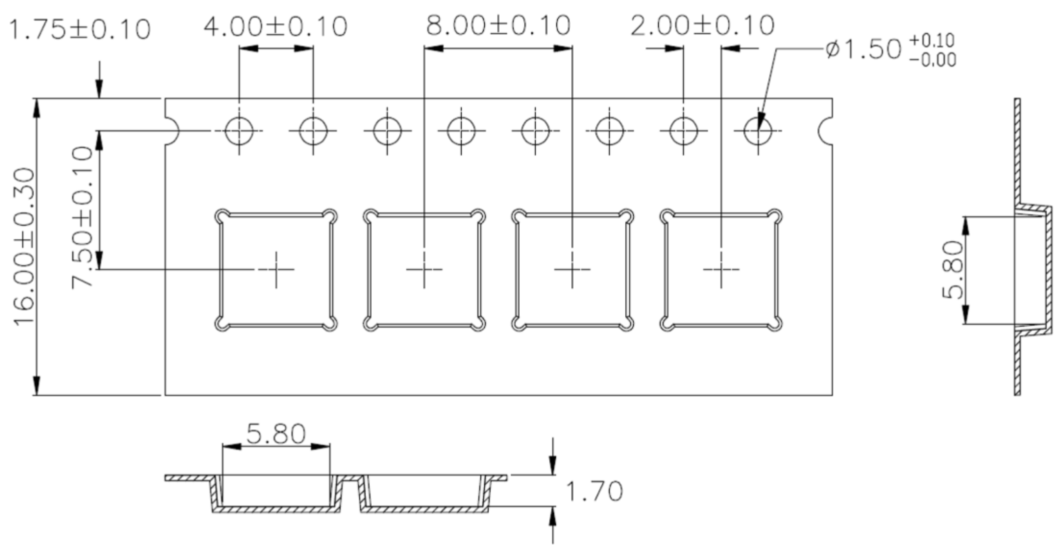


Figure 26. 5.5 x 5.5 mm² Tape Dimension (for package height 1.4 mm)

4.10 TAPE AND REEL PACKAGING

Please refer to each product's datasheet for IMU product Pin-1 orientation in tape.

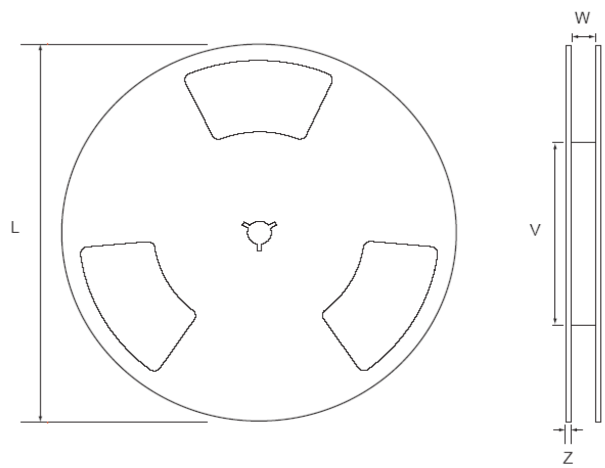


Figure 27. Reel Outline Drawing

REEL (mm)			
L	**V**	**W**	**Z**
330	**102**	**12.8**	**2.3**

Table 3. Reel Dimensions

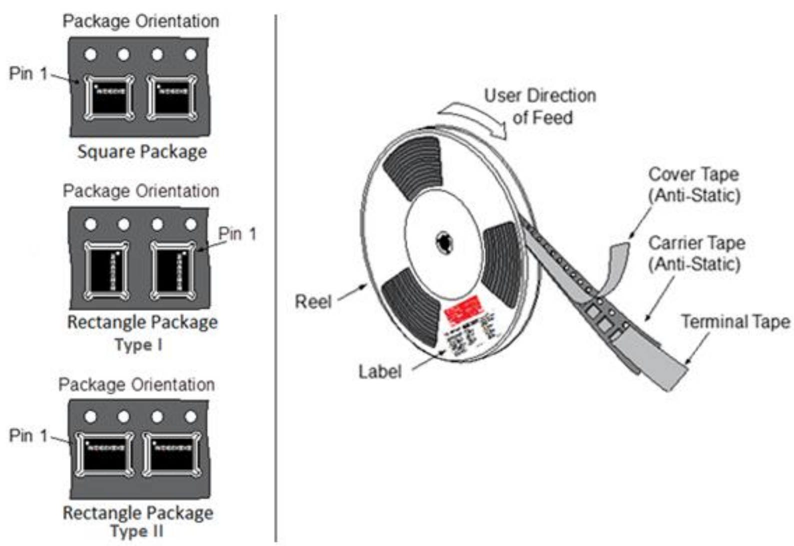


Figure 28. Tape and Reel – Package Orientation

Quantity Per Reel	5,000
Reels per Box	1
Boxes Per Carton (max)	5
Pcs/Carton (max)	25,000

Table 4. Reel Specifications for Packages smaller than 4.5 x 4.5 mm^2

Quantity Per Reel	4,000
Reels per Box	1
Boxes Per Carton (max)	5
Pcs/Carton (max)	20,000

Table 5. Reel Specification for Package 4.5 x 4.5 mm^2

Figure 29. Barcode Label

Figure 30. Location of Label on Reel

Figure 31. Barcode Label on the Left of the Box

4.11 PACKAGING

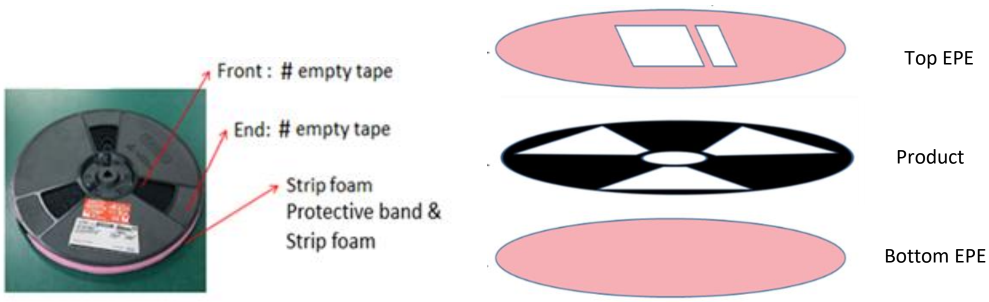


Figure 32. REEL Packing

CAUTION — LEVEL **1**

This bag contains
MOISTURE-SENSITIVE DEVICES

If Blank, see adjacent bar code label

1. Calculated shelf life in sealed bag: 12 months at < 40 C and < 90% relative humidity (RH)
2. Peak package body temperature: Not to exceed 260 °C (If Blank, see adjacent bar code label)
3. After bag is opened, devices that will be subjected to reflow solder or other high temperature process must
   a) Mounted within: unlimited hours of factory conditions ≤ 30 ° C/85% RH,OR
   b) Stored per J-STD-033
4. Devices require bake, before mounting, if:
   a) Humidity Indicator Card is > 10% when read at 23 ± 5° C
   b) 3a or 3b not met.
5. If baking is required ,refer to IPC/JEDEC J-STD-033 for bake procedure

Note: If device containers cannot be subjected to high temperature or shorter bake times are desired, reference IPC/JEDEC J-STD-033 for bake procedure

Bag Seal Date: ____ (If Blank, see adjacent bar code label)

Note: Level and body temperature defined by IPC/JEDEC J-STD-020

CAUTION — LEVEL **3**

This bag contains
MOISTURE-SENSITIVE DEVICES

If Blank, see adjacent bar code label

1. Calculated shelf life in sealed bag: 12 months at < 40° C and < 90% relative humidity (RH)
2. Peak package body temperature: Not to exceed 260 °C (If Blank, see adjacent bar code label)
3. After bag is opened, devices that will be subjected to reflow solder or other high temperature process must
   a) Mounted within: 168 hours of factory (If Blank, see adjacent bar code label) conditions ≤ 30 °C/60% RH,or
   b) Stored per J-STD-033
4. Devices require bake, before mounting, if:
   a) Humidity Indicator Card reads >10% for level 2a-5a devices or >60% for level 2 devices when read at 23± 5 °C
   b) 3a or 3b are not met
5. If baking is required ,refer to IPC/JEDEC J-STD-033 or bake devices for 9 hours at 125 ±5 °C in trays or in Tape-and-Reel bake devices for 312 hours at 40 °C, ≤ 5% RH

Bag Seal Date: ____ (If Blank, see adjacent bar code label)

Note: Level and body temperature defined by IPC/JEDEC J-STD-020

CAUTION — LEVEL **5**

This bag contains
MOISTURE-SENSITIVE DEVICES

If Blank, see adjacent bar code label

1. Calculated shelf life in sealed bag: 12 months at < 40° C and < 90% relative humidity (RH)
2. Peak package body temperature: Not to exceed 260 °C (If Blank, see adjacent bar code label)
3. After bag is opened, devices that will be subjected to reflow solder or other high temperature process must
   a) Mounted within: 48 hours of factory (If Blank, see adjacent bar code label) conditions ≤ 30 °C/60% RH,or
   b) Stored per J-STD-033
4. Devices require bake, before mounting, if:
   a) Humidity Indicator Card is > 10% when read at 23 ± 5° C
   b) 3a or 3b are not met
5. If baking is required , devices may be baked for 12 hours at 125± 5 °C

Note: If device containers can not be subjected to high temperature or shorter bake times and desired , reference IPC/JEDEC J-STD-033 for bake procedure

Bag Seal Date: ____ (If Blank, see adjacent bar code label)

Note: Level and body temperature defined by IPC/JEDEC J-STD-020

Figure 33. MSL Label (MSL 1, MSL 3 and MSL 5)

Figure 34. Reel Packing for shipping

Caution Label

ESD Label

Figure 35. Labels

Standard Packing Box

Boxes Placed in Foam Lined Shipper Box

Outer Shipper Label

Figure 36. Boxes and Labels

Representative Shipping Carton Label:

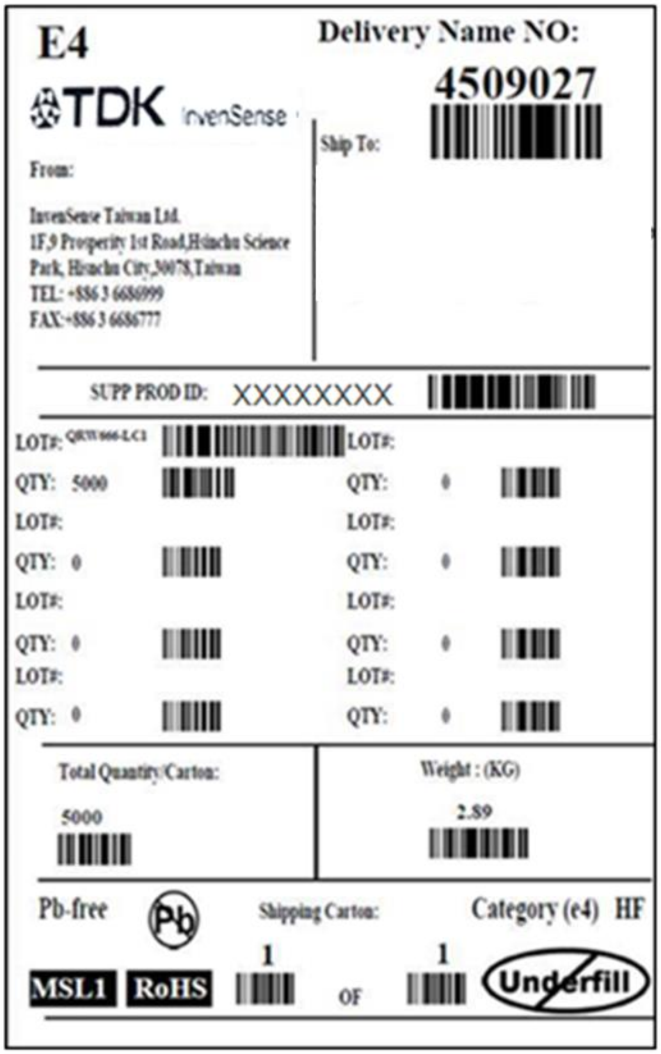


Figure 37. Outer Shipping Carton Label

5 REVISION HISTORY

REVISION DATE	REVISION	DESCRIPTION
11/01/2022	1.0	Initial release
12/22/2022	1.1	Cosmetic updates
09/15/2023	1.2	Updated section 2.2, added QFN package PCB land pattern Updated section 4 and rest of the document to cover all IMU Products
12/06/2023	1.3	Changed chapter 3 title. Updated chapter 4.1 and 4.2
01/29/2024	1.4	Updated introduction chapter 1, and legal information; and fixed typo in the footer
02/26/2024	1.5	Added section 4.3. Updated section 4.5: Added Figure 20 and Table 2.
04/24/2024	1.6	Added sections 2.6, 4.6 & 4.9
07/01/2024	1.7	Added section Figure 28 Package Orientation and Table 4~5 Reel Q'ty Specification
07/25/2024	1.8	Pick and Place target parameters revised (section 4.3.1)