

mCube

LGA Package Application Note

AN-001

Rev.02

mCube, Inc.



Guidelines for Printed Circuit Board (PCB) Design and Assembly with mCube Land Grid Array (LGA) Package Sensors

The following document provides the general guidelines for the PCB design, device handling, soldering and assembly of mCube's LGA package sensors. The purpose is to minimize the impact from PCB design and SMT handling by using optimized practices. Through the best practice of PCB design and SMT handling that can reduce unnecessary stress induced sensor offset errors. The guidelines provided here are based on design experiments for the packages shown. They do not represent exact conditions present at real customer implementations. The information recommended in this document should be used as guidelines while developing an application specific solution. Actual experience as well as development efforts are still needed to optimize the process per individual device requirements.



Table of Contents

1. Device Handling.....	4
1.1 Mechanical Shock	4
1.2 Moisture Sensitivity Level Control (MSL)	5
1.3 Electrostatic Discharge (ESD).....	5
2. Accelerometer Sensor Placement	6
3. LGA Package Introduction.....	9
3.1 Package Description	9
3.2 Package Cross Section.....	10
4. PCB Design	11
4.1 PCB Land Pattern	11
4.2 PCB Surface Finish.....	12
5. Stencil Design and Solder Paste Material.....	13
5.1 Stencil Design	13
5.2 Solder Paste Material.....	14
6. Surface Mount Technology (SMT) Process Flow.....	15
7. Solder Paste Printing	16
8. Component Placement	17
9. Solder Paste Reflow Guidelines	158



1. Device Handling

1.1 Mechanical Shock

mCube's mechanical sensor are designed to handle medium-low-g shock events, so direct mechanical shock to the package will generate high g forces and should be avoided. SMT assembly houses should use automated assembly equipment with either plastic nozzles or nozzles with compliant (soft i.e. rubber or silicone) tips.

Recommendations:

- Do not use metal or ceramic nozzle tips during assembly.
 - These cause excessive g forces and therefore are NOT recommended.
- Place the sensor with minimal direct force during assembly.
 - This is achieved by optimizing the placement force control in the g-sensor library for chip shooters or IC placers.
- Place the sensor with minimum pick and place assembly speeds.
 - This is achieved by optimizing the speed settings control in the g-sensor library for chip shooters or IC placers.
- Discard mishandled sensors.
 - If the sensor is dropped from a height of 50mm or greater it should be discarded and not used.
 - Direct impact to a hard surface can also generate high g forces. The best practice is to discard any part that has been dropped.
- Handle partially finished PCB assemblies carefully.
 - Transport assemblies in shock-absorbent carriers.
 - Do not expose partial assemblies to bending, flexing or excessive shocks during product assembly.
- Do not use ultrasonic welding



1.2 Moisture Sensitivity Level Control (MSL)

The typical factory floor life of mCube LGA Package sensors is 168 hours (1 week) floor life at ambient conditions of $\leq 30^{\circ}\text{C}/60\%\text{RH}$ (MSL3).

Refer to IPC/JEDEC J-STD-020D.1 “Joint Industry Standard: Moisture/Reflow Sensitivity Classification for Non-hermetic Solid State Surface Mount Devices” and IPC/JEDEC J-STD-033A “Joint Industry Standard: Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices.”

Recommendations:

- Store the carrier tape and reel in the dry pack unopened until required on the assembly floor.
- If the product reel has been removed from the dry pack, reseal the product reel inside the dry pack with a black protective belt to avoid crushing the carrier tape from the reel or store openly it in a controlled humidity condition.
- Store the pizza box in the vertical position.

1.3 Electrostatic Discharge (ESD)

- Refer to the JEDEC standard JESD625-A “Requirements for Handling Electrostatic-Discharge-Sensitive (ESDS) Devices” for the correct handling techniques.



2. PCB Design and Accelerometer Sensor Placement

Recommendations (see Figure 1 through Figure 5):

- Use a rigid PCB material, as thick as possible.
 - A PCB thickness of 0.6mm or more is recommended.
 - Use materials with a low coefficient of thermal expansion.
 - PCB layout should be as symmetrical as possible to avoid unintentional components movement as a result of unbalanced solder wetting forces.
 - Solder paste must be as thick and as uniform as possible.
- Place the sensor in a part of the PCB which is mechanically stable as shown in Figure 1. Keep distance from point A to reduce the diving board effect and any impacts on the sensor operation.
 - It should not be exposed to any flexing or twisting during device operation.

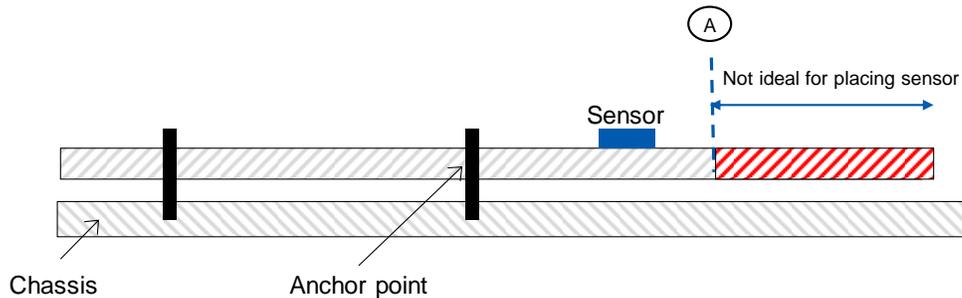


Figure 1 - Place Sensor in a Mechanically Stable Area of PCB



- Do not place the sensor near (see Figure 2):
 - Heat source (battery, power amp., microprocessor, backlighting circuitry, etc.)
 - Switching power supply
 - Push button
 - Strong magnetic or electric fields (speaker, vibrator, motor, camera module infrared radiation and charging sensor)
- Heat can cause the PCB to flex, causing PCB stress which leads to offset errors.

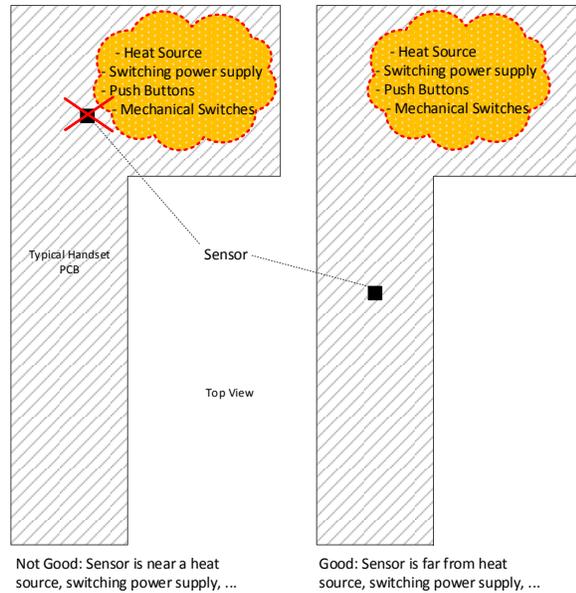


Figure 2 - Do Not Place Sensor Near Heat Source or Switching Power Supply



- Do not place the sensor too close to PCB screw holes and anchor points.
 - The screw can flex the PCB and stress the sensor package.

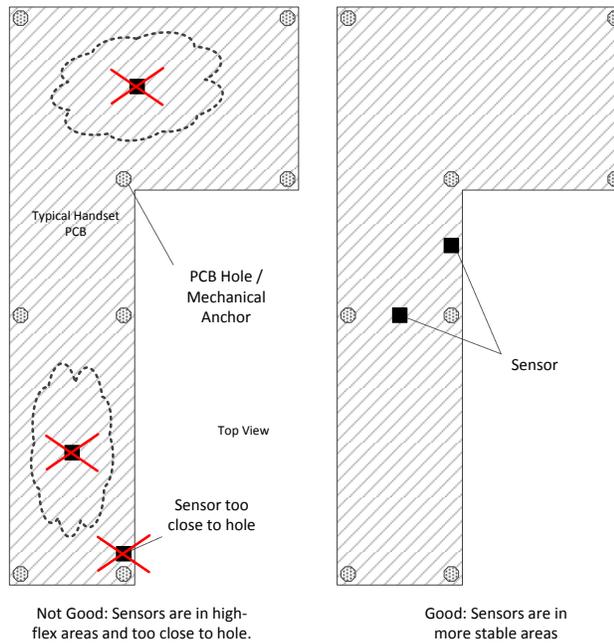


Figure 3 - Be Careful of Mechanical Anchor Points of Chassis

- Keep at a minimum of $d \geq 3\text{mm}$ from the PCB edge (for application $\leq 6''$)
Keep at a minimum of $d \geq 10\text{mm}$ from the PCB edge (for application $> 6''$)

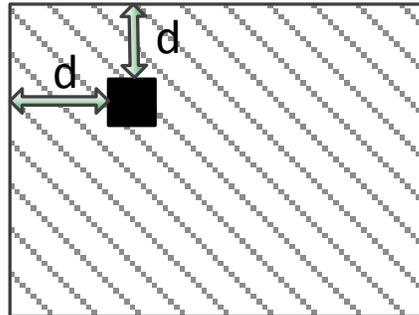


Figure 4 - Minimum of distance from PCB edge



- Place the sensor away from mechanical connectors as shown in Figure 5.
 - These produce a mechanical ‘snap’ force when the connector is used. This can cause the sensor to see excessive g-forces and is not recommended.
 - Do not place the sensor in places where using the connector will bend the PCB underneath the sensor.

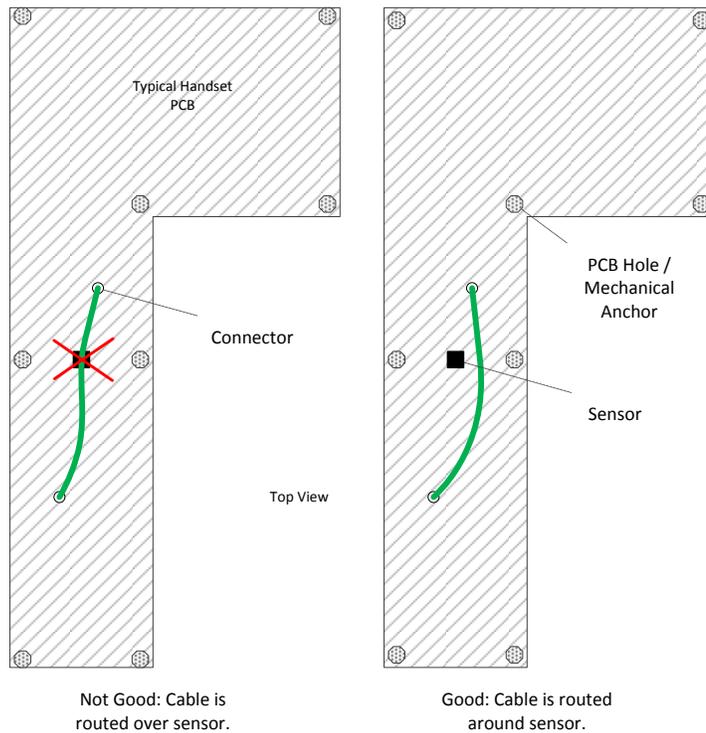


Figure 5 - Be Careful of Cable Routing

- Outer case direct touch or cover on top of the sensor is prohibited.



3. LGA Package Introduction

3.1 Package Description

- LGA package offer a square grid array terminal pads on the bottom surface. Reflow soldering provide physical connection between the packaged bottom terminal pads and PCB land pads. LGA packages similar as the ball grid array (BGA) type package but without the solder balls underneath with following features:
 - Typically apply with organic laminate as an interposer with nickel-gold (NiAu) plated terminations.
 - Lower stand-off height of about 60~100um.
 - Compatible to standard SMT process.
 - No solder ball underneath eliminates the risk of missing or damaged balls during the shipping and handling.
 - The periphery bottom terminal pads design sometimes referred as the quad flat no leads (QFN) package

Figure 6 shows the example of the top side and bottom side of a mCube LGA package.

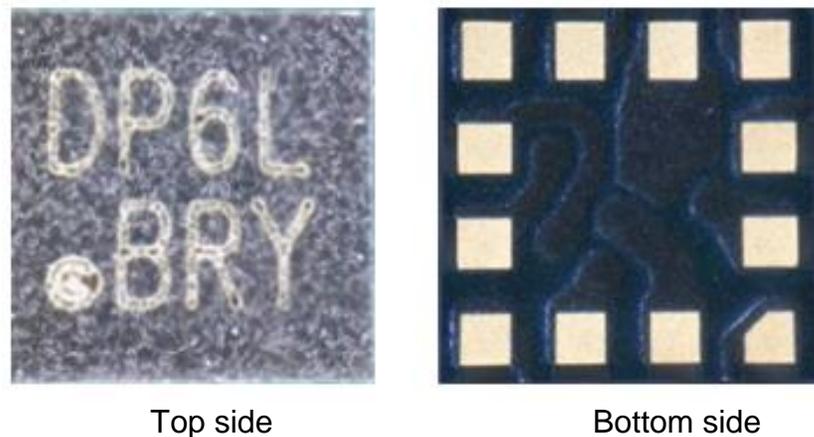


Figure 6 – LGA top side and bottom side of a mCube LGA package

mCube offers various industry LGA size and thickness with options of terminal pad counts and pitch. Package size from 1.6mmx1.6mm to 3mmx3mm with terminal pad pitch from 0.4mm to 0.5mm. Refer to mCube device datasheet to obtain the detailed package relevant dimensions.



3.2 Package Cross Section

- Figure 7 shows a typical LGA package cross sectional illustration with single die and two layer substrate configuration. The LGA package can also be assembled on multi-layers laminations with multi-dies as a module. A standard wire bonding technology with Cu or Au type wires is used for the electrical connection between die and substrate. Encapsulation process afterward to protect the connecting wires and dies inside the package.

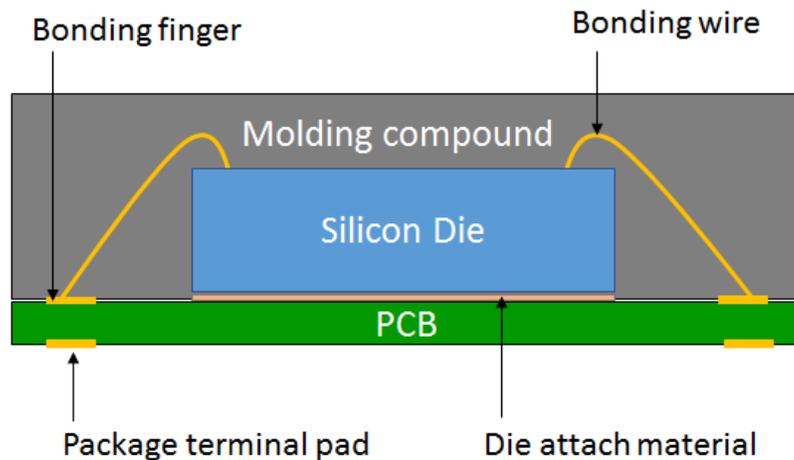


Figure 7 – LGA cross-sectional illustration with single die and two-layer substrate configuration



4. PCB Design

4.1 PCB Land Pattern

There are many factors to decide the selection of solder mask defined (SMD) or non-solder mask defined (NSMD) pads. Although both types are used successfully in application, the NSMD pad design which showed the tighter tolerance on copper etching than on solder masking is recommended for all PCB land patterns as shown in figure 8.

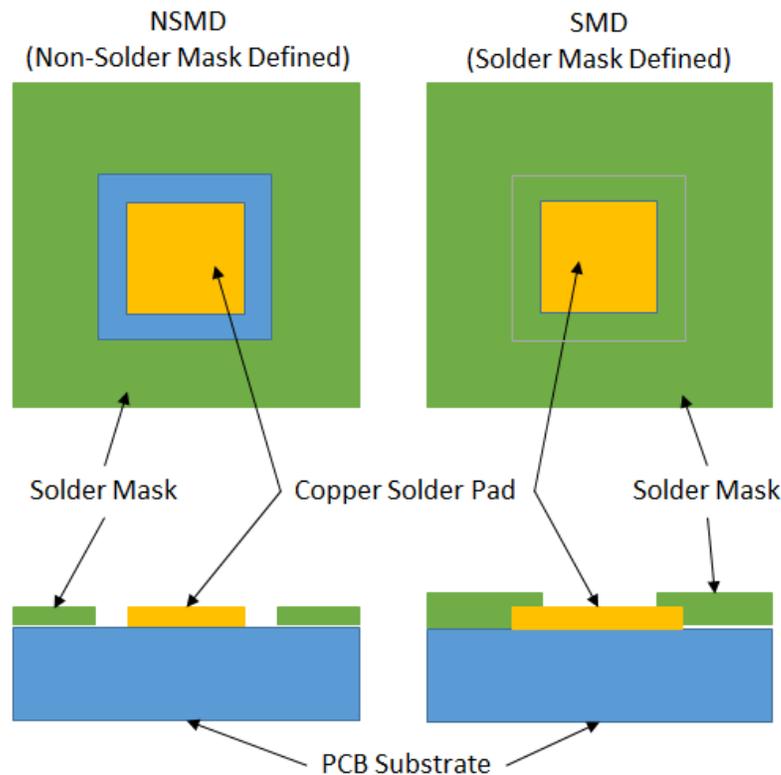


Figure 8 – NSMD and SMD land pattern illustrations

- NSMD lands: the copper pad is smaller than solder mask opening.
- SMD lands: the solder mask opening is smaller than the copper pad
- The solder mask registration is suggested to be less than or equal to 50um.
- Less than 1 OZ copper pad thickness is recommended.
- Typically, the PCB land pattern should be 1:1 to 1:1.15 to the package terminal pads for fine pitch LGA package.
- As a general rule to have a good solder fillet, the PCB land pattern dimension is most likely to be larger than that of package terminal pad dimension. The recommendation is to extend the land pattern about 0.025mm on outer/inner side along the centerline



of the package and 0.015mm on the right/left side along the pitch direction of the package as shown in figure 9.

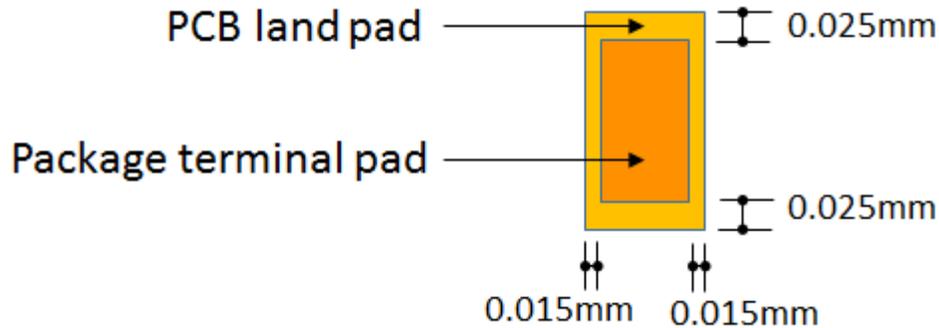


Figure 9 – PCB land pattern recommendation

However, mCube strongly suggest that end user to conduct a design of experiment (DOE) to determine the optimized PCB land pattern design.

With the use of proper PCB footprint design, the package will undergo minimal stress and will self-align.

- Make signal traces near sensor with smaller sizes.
 - Signal traces near the package should have minimal width as design allows and drawn straight away from the package. 100um max. recommendation. Prefer 75um in the NSMD design exposed area inside the solder mask opening.
 - Wider traces can be used after 0.5mm from the package.
 - A tear drop is recommended at the trace entry to lower the risk of trace crack.
- Do not place vias or traces under the package.
 - These can cause uneven assembly during reflow.
 - These can cause the PCB to bend and flex unevenly with temperature changes.
 - These can cause voids in the solder joint affect the reliability.

4.2 PCB Surface Finish

PCB surface finishes with Organic Surface Preservatives (OSP), Ni-Au Electroless Nickel Immersion Gold (ENIG), and Electroless nickel / Electroless Palladium / Immersion Gold (ENEPIG) are used in application. However, OSP is recommended.



5. Stencil Design and solder paste

A proper stencil design and use of proper solder paste will reduce package stresses. Therefore, the stencil material, manufacture, thickness, aperture, and solder paste selection must be taken into consideration to obtain the optimal solder paste volume on PCB land pad.

5.1 Stencil Design

Recommendations:

- Both a laser cut stainless steel or electroformed stencil with trapezoidal walls and corners rounded are used in industry. However, mCube recommend to use a laser cut stencil with electropolish or nano-coating on cleaning side stencil surface for well quality control.
 - Trapezoidal walls (5° tapering) with bottom opening larger than the top to ensure uniform release of solder paste and reduce smearing as shown in figure 10.
 - Corners rounded to minimize clogging.
- The stencil thickness and size should be as follows:
 - For mCube LGA packages, the stencil thickness can be between 0.1mm~0.15mm. Typically 0.125mm thickness as a guide for 0.5mm pitch package and 0.1mm thickness as a guide for 0.4mm pitch package.
 - The same aperture size to PCB land pad 1 to 1 for SnAgCu alloys is recommend. For finer pitch, it might be necessary to reduce the aperture by 20% to prevent from shorting beneath the LGA.
 - Aperture ratio (width of aperture opening / stencil thickness) > 1.5.
 - However, stencil thickness and size can be optimized at the production line for best solder release and yield.
- mCube recommend to inspect the stencil openings for burrs and any other quality issues prior to use.
- Accurately align the stencil and PCB using automated equipment prior to application of the solder paste.
 - With proper PCB trace and pad layout, the package should self-align.



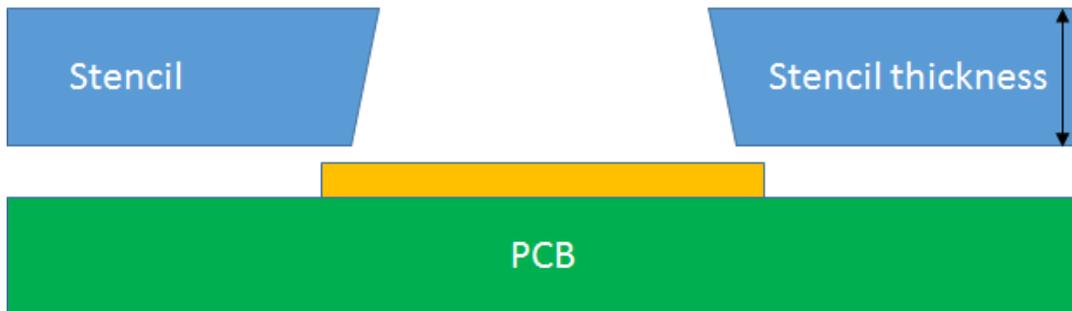


Figure 10 – Stencil cross sectional view

5.2 Solder Paste Material

Recommendations:

- Use solder paste appropriate for the pad size
 - Use type 4 or finer paste is recommended. Storage and use conditions should follow solder paste supplier's specification.
 - i.e. Solder paste should be reach room temperature and be mixed thoroughly.
 - Due to lower stand-off height of the LGA package, low residue, non-clean solder paste is commonly used to reduce to the cleaning operation is recommended.
 - Use only Pb-free solder paste (SnAgCu family for lead-free application). SnPb solder paste should not be used for mCube LGA package.



6. Solder Mount Technology (SMT) Process Flow

A typical SMT process flow is shown in figure 11.

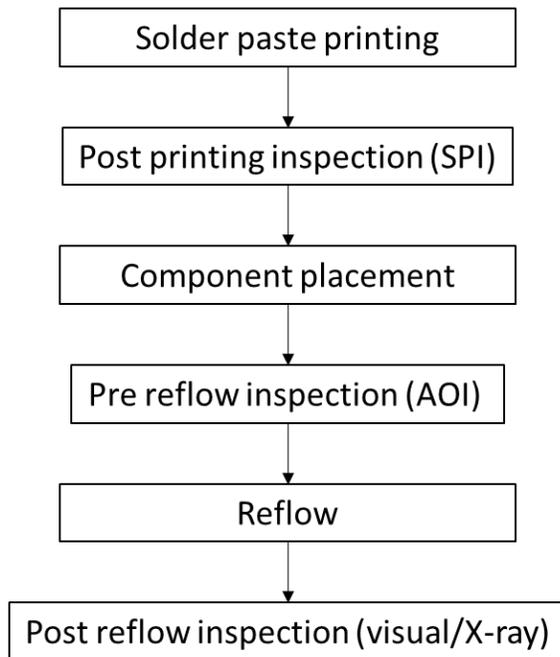


Figure 11 – SMT process flow



7. Solder Paste Printing

Due to mCube LGA Package apply the fine pitch and small terminal geometry, we need to pay special attention to the solder paste printing process.

Recommendations:

- Recommend to apply the automatic stencil printer to print out the solder paste onto the PCB
- Recommend to optimize the blade angle and speed to ensure the even solder line thickness.
- Recommend to have solder paste inspection (SPI) system for mCube LGA package assembly process to measure below data before parts placement. A uniform and repeatable solder deposit is most critical to reflow yield.
 - Solder paste height
 - PCB pad coverage
 - Registration accuracy to PCB land pad
- The bottom side of the stencil need to be cleaned regularly. Auto stencil cleaning frequency should be defined to avoid smearing.
- The solder joint thickness for LGA terminal pads typically to be 50um~75um.
- Recommend to have a stand along stencil cleaner to clean the small holes on stencil thoroughly every shift.



8. Component Placement

The LGA package is small in terminal pad size. For better accuracy control, mCube recommended to apply automated fine pitch placement machines with vision alignment.

Recommendations:

- Recommend to have an auto optical inspection (AOI) system for mCube LGA package SMT process to ensure no any solder paste relative issue before reflow process.
- Standard pick and placement machine with ± 0.05 mm accuracy. The accuracy of the pick and placement machine dominate the package placement and rotation alignment. Less than 50% off the pad center parts will automatically self-align during the reflow process.
- Z height distance control methods are recommended over the force control.
- Plastic pick up nozzles are recommended.



9. Solder Paste Reflow Guideline

The actual soldering reflow profile need to be calibrated considering several factors, including oven type, solder paste type, component density across the PCB, temperature difference across PCB, PCB layout, material properties, and etc. Therefore, it is not possible to define a specific solder profile for the sensor only. The actual optimized solder reflow profile and controlled ramp and cool down setting need to be defined by the end user base on above considerations.

Solder Assembly Process Recommendations

- Expose the sensor to only one single reflow.
 - So, if the sensor is mounted to a two-sided printed circuit board, incorporate the sensor in the second pass.
- Use a self-cleaning solder paste.
 - If a self-cleaning solder paste is not used, clean the flux from the board after soldering to eliminate the possibility of leakage between PCB pads.
- Do not define a specific solder profile for the sensor only.
 - Define the PCB soldering profile based on the thermal mass of the entire assembly board.
 - Use a time and temperature profile that is based on the PCB design and manufacturing process.
- Recommend to apply more than 10 zones reflow oven (temperature uniformity $\pm 5^{\circ}\text{C}$). More zones are easier adjustment of various profile
- Recommend to apply forced gas convention reflow oven with nitrogen supply during SMT reflow process. It can provide a less solder oxidation compared to air reflow.
- The non-destructive vision inspection and x-ray inspection are recommended to verify the soldering quality after reflow process.

Figure 12, Table 1 and Table 2 are provided as a recommended soldering profile based on JEDEC J-STD-020D.1 Standard.



PCB Design, Device Handling and Assembly Guidelines

Profile Feature	Pb-Free Assembly
Preheat/Soak Temperature Min (T_{smin}) Temperature Max (T_{smax}) Time (t_s) from (T_{smin} to T_{smax})	150 °C 200 °C 60 – 120 seconds
Ramp-up rate (T_L to T_p)	3 °C/second max.
Liquidous temperature (T_L) Time (t_l) maintained above T_L	217 °C 60 – 150 seconds
Peak package body temperature (T_p)	For users T_p must not exceed the Classification temperature listed in Pb-Free Process Table in Table 2 For suppliers T_p must equal or exceed the Classification temperature listed in Pb-Free Process Table in Table 2
Time (t_p)' within 5°C of the specified classification temperature (T_c), see Figure 10	30* seconds
Ramp-down rate (T_p to T_L)	6°C/second max.
Time 25°C to peak temperature	8 minutes max.
*Tolerance for peak profile temperature (T_p) is defined as a supplier minimum and a user maximum	

Table 1 – Classification reflow profiles

Pb-Free Process - Classification Temperatures (T_c)

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

Table 2 – Pb-Free process – Classification temperature (T_c)



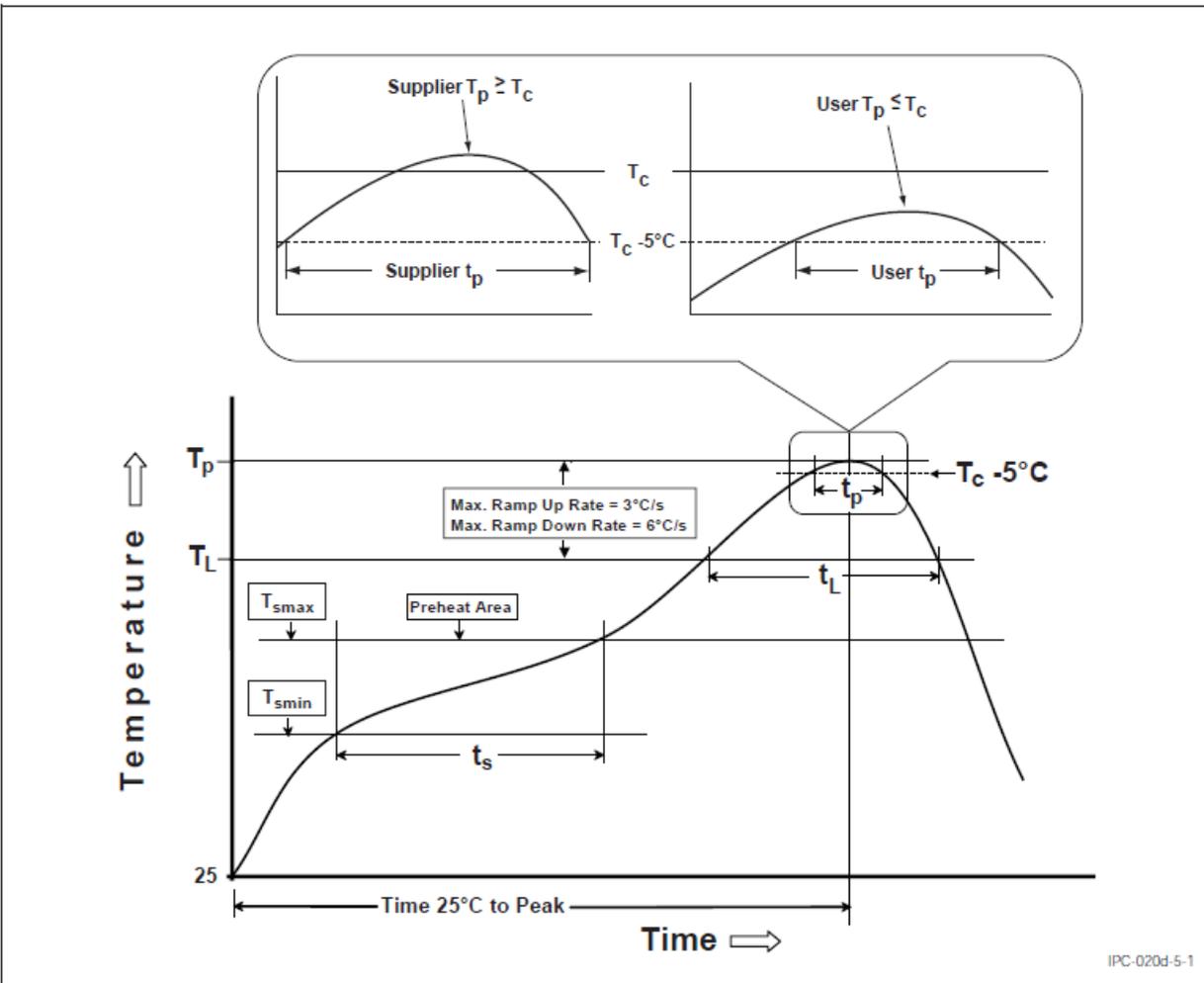


Figure 12 – Classification reflow profile (Not to scale)

