# Assembly Considerations for µModule® BGA and LGA Packages

June 2024


AHEAD OF WHAT'S POSSIBLE™

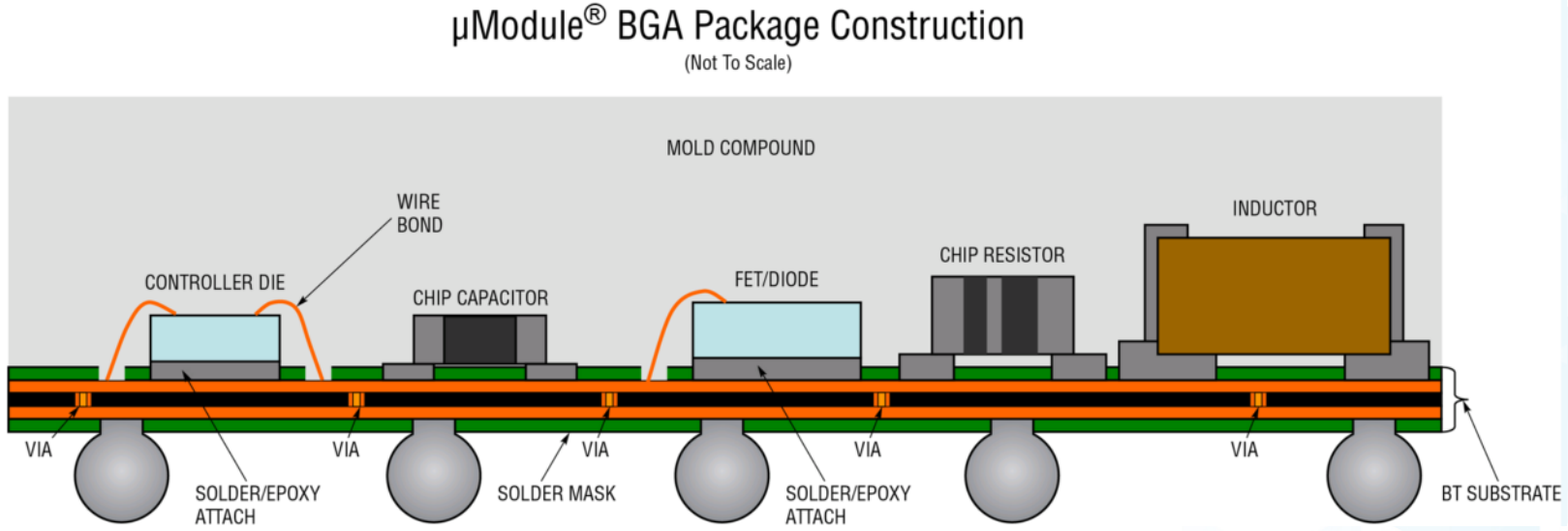
# Outline

- Package Construction for BGA and LGA Package
- PCB Design Guidelines for BGA and LGA Package
  - BGA PCB Pad Layout
  - LGA PCB Pad Layout
- PCB Design Guidelines – Round Pads for BGA & LGA Package
- PCB Design Guidelines – Square Pads for LGA Package
- Moisture Sensitivity Considerations for BGA and LGA Package
- Solderability Test for LGA Package
- Board Assembly Process for BGA and LGA Package
  - Stencil Design
  - Screen Print
  - LGA µModule® Placement Additional Guidelines
  - Reflow Profile
  - Solder Joint Void
  - Cleaning
- Second Side Assembly Guideline for BGA and LGA Package
- Rework for BGA and LGA Package
- BGA Package: Dual Component on Package Handling
- Maximum Mechanical Load for Heatsink Attach
- FAQs

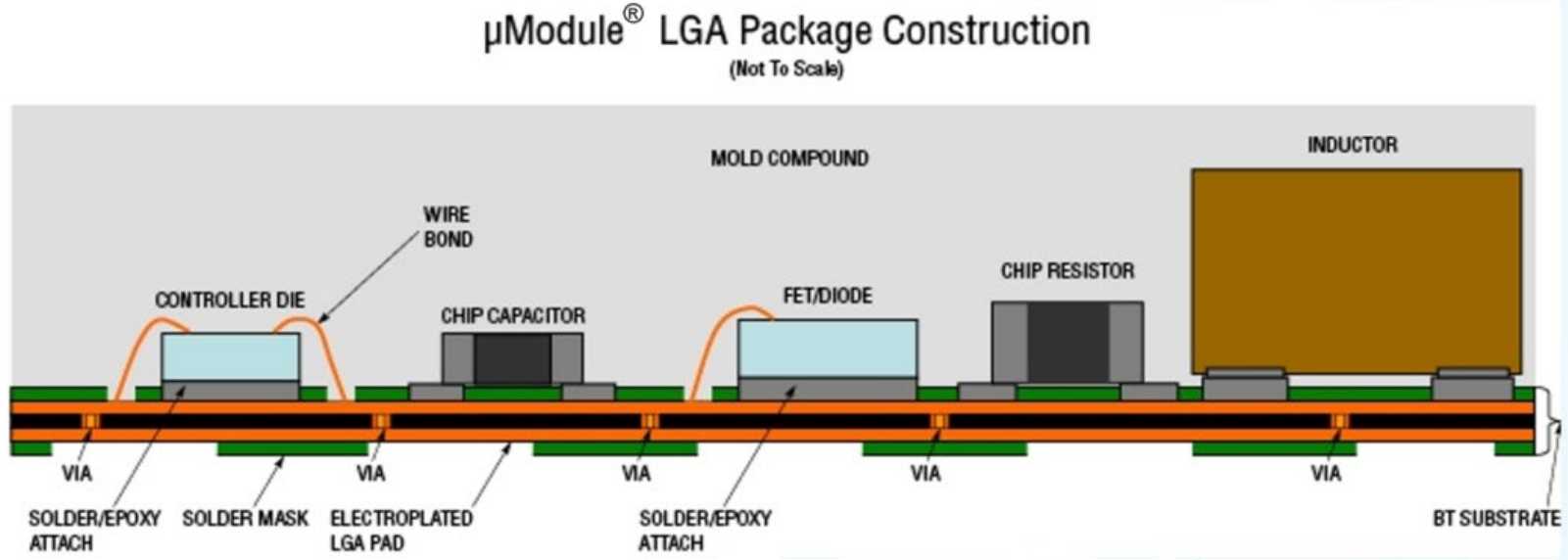
# Package Construction for BGA and LGA Package

# BGA and LGA Package Construction

1.)

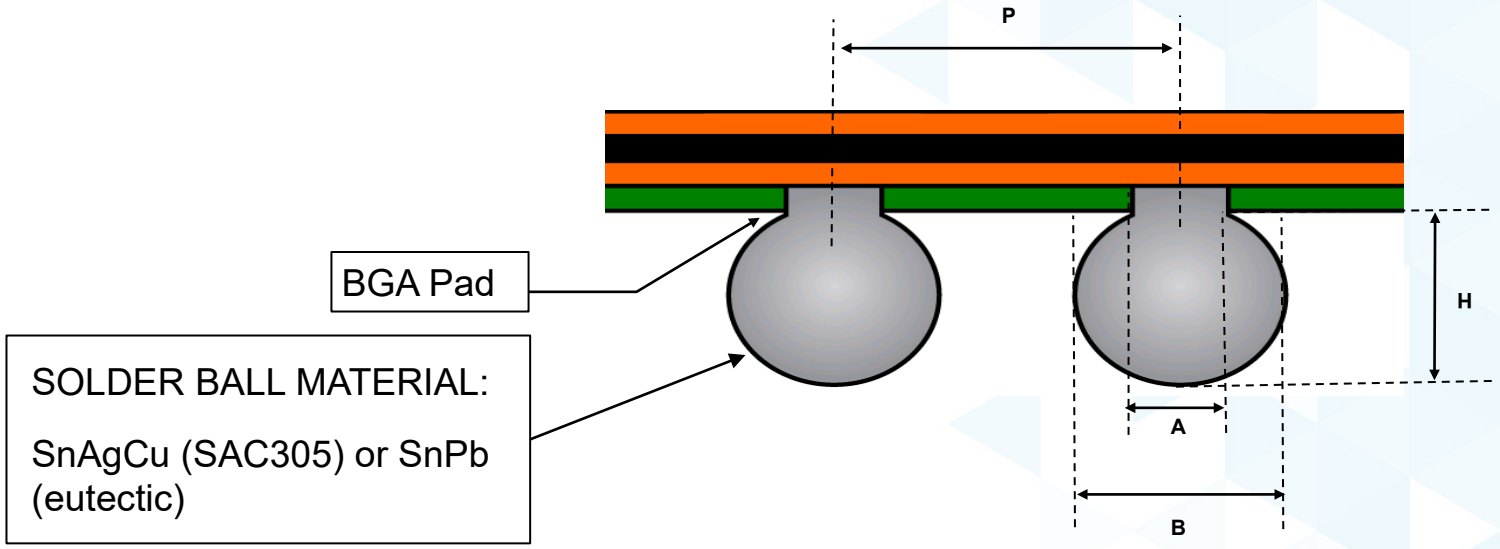



2.)




| Table 1 : LGA Pad Finishing | |
|---|---|
| **Electroplating** | **ENEPIG** |
| Ni : 3 – 20 µm | Ni : 3 – 8 µm |
| Au : 0.3 – 0.8 µm | Pd : 0.05 – 0.1 µm |
| | Au : 0.1 – 0.5 µm |

# BGA Pad and Ball Geometry




| Table 2 : Package Solder Ball and Pad Dimensions (mm) | | | |
|---|---|---|---|
| BGA Pad Pitch, (P) | Pad Opening, (A) | Solder Ball Diameter, (B) | Solder Ball Height, (H) |
| 0.65 | 0.35 | 0.40 | 0.30 |
| 0.80 | 0.40 | 0.50 | 0.40 |
| 1.00 | 0.50 | 0.60 | 0.50 |
| 1.27 | 0.63 | 0.75 | 0.60 |

# PCB Design Guidelines for BGA and LGA Package

# PCB Design Guidelines

- µModule® BGA/LGA pad design
  - All µModule® BGA packages have solder mask defined (SMD) pads

| Table 3 : BGA PCB PAD LAYOUT | | | | | |
|---|---|---|---|---|---|
| **ROUND PADS** | **Pitch (mm), (P)** | **SMD PADS** | | **NSMD PADS** | |
| | | **Metal Pad Size (mm), (B)** | **Solder Mask Opening (mm), (A)** | **Metal Pad Size (mm), (B1)** | **Solder Mask Opening (mm), (A1)** |
| **All devices** | 0.65 | ≥ 0.45 | 0.35 | 0.35 | ≥ 0.45 |
| | 0.80 | ≥ 0.50 | 0.40 | 0.40 | ≥ 0.50 |
| | 1.00 | ≥ 0.60 | 0.50 | 0.50 | ≥ 0.60 |
| | 1.27 | ≥ 0.73 | 0.63 | 0.63 | ≥ 0.73 |

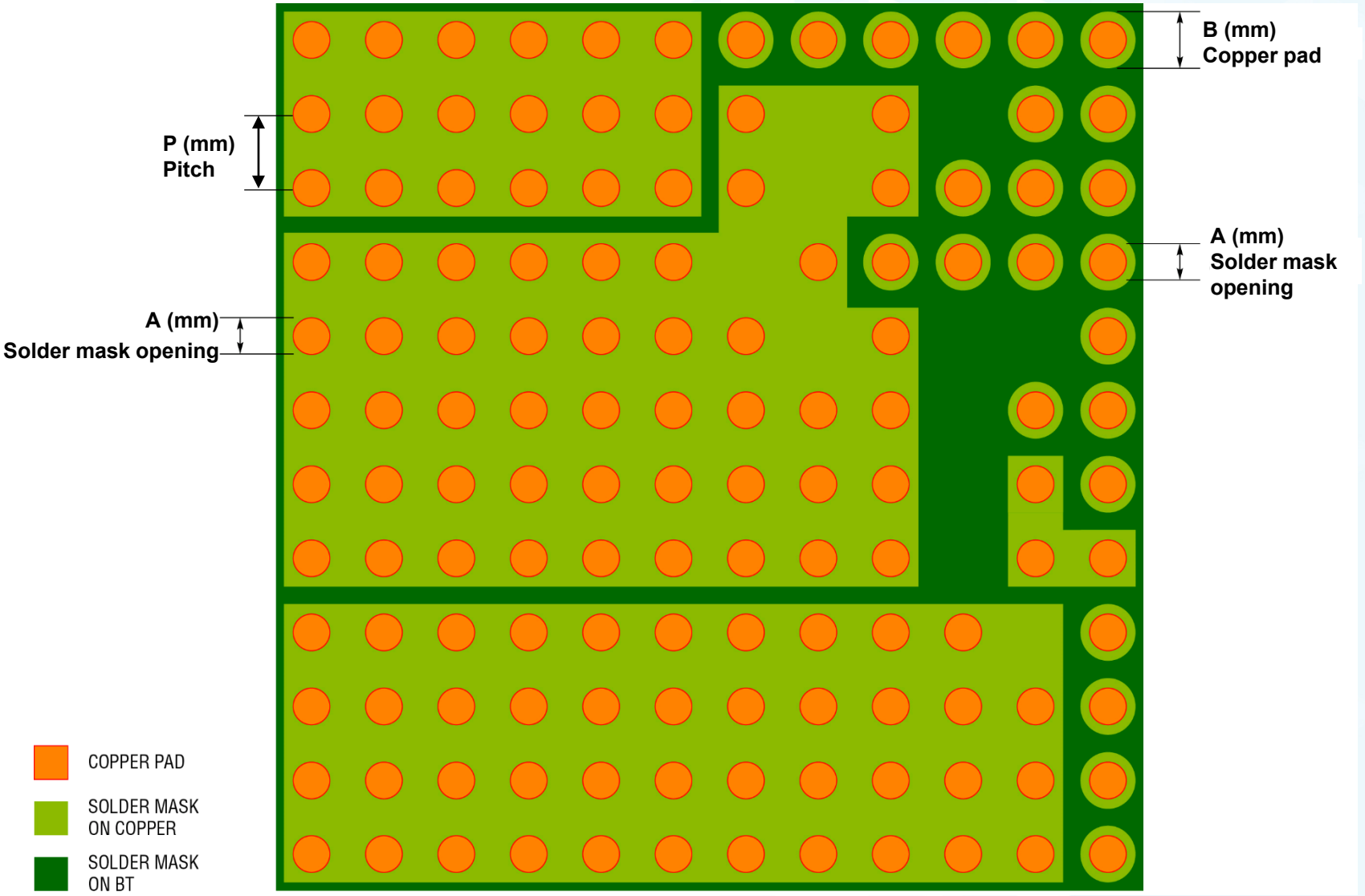
| Table 4 : LGA PCB PAD LAYOUT | | | | | |
|---|---|---|---|---|---|
| **SQUARE PADS** | **Pitch (mm), (P)** | **SMD PADS (Recommend use same size as Package Pad Opening)** | | **NSMD PADS** | |
| | | **Metal Pad Size (mm), (B)** | **Solder Mask Opening (mm), (A)** | **Metal Pad Size (mm), (B1)** | **Solder Mask Opening (mm), (A1)** |
| **All devices** (except LTM4604 & LTM4608) | 1.27 | ≥ 0.73 | 0.630 | 0.630 | ≥ 0.73 |
| LTM4608 | 1.27 | 0.862 | 0.762 | 0.762 | 0.862 |
| LTM4604 | 1.27 | 0.989 | 0.889 | 0.889 | 0.989 |
| **ROUND PADS** | | | | | |
| **All devices** | 0.65 | ≥ 0.45 | 0.35 | 0.35 | ≥ 0.45 |
| | 0.80 | ≥ 0.50 | 0.40 | 0.40 | ≥ 0.50 |
| | 1.00 | ≥ 0.60 | 0.50 | 0.50 | ≥ 0.60 |
| | 1.27 | ≥ 0.73 | 0.63 | 0.63 | ≥ 0.73 |

**Note: If there are mixed SMD & NSMD pads, ensure that the finish pad sizes are as designed**

# PCB Design Guidelines – Round Pads for BGA and LGA Package

# BGA & LGA Solder Mask Defined Round Pads

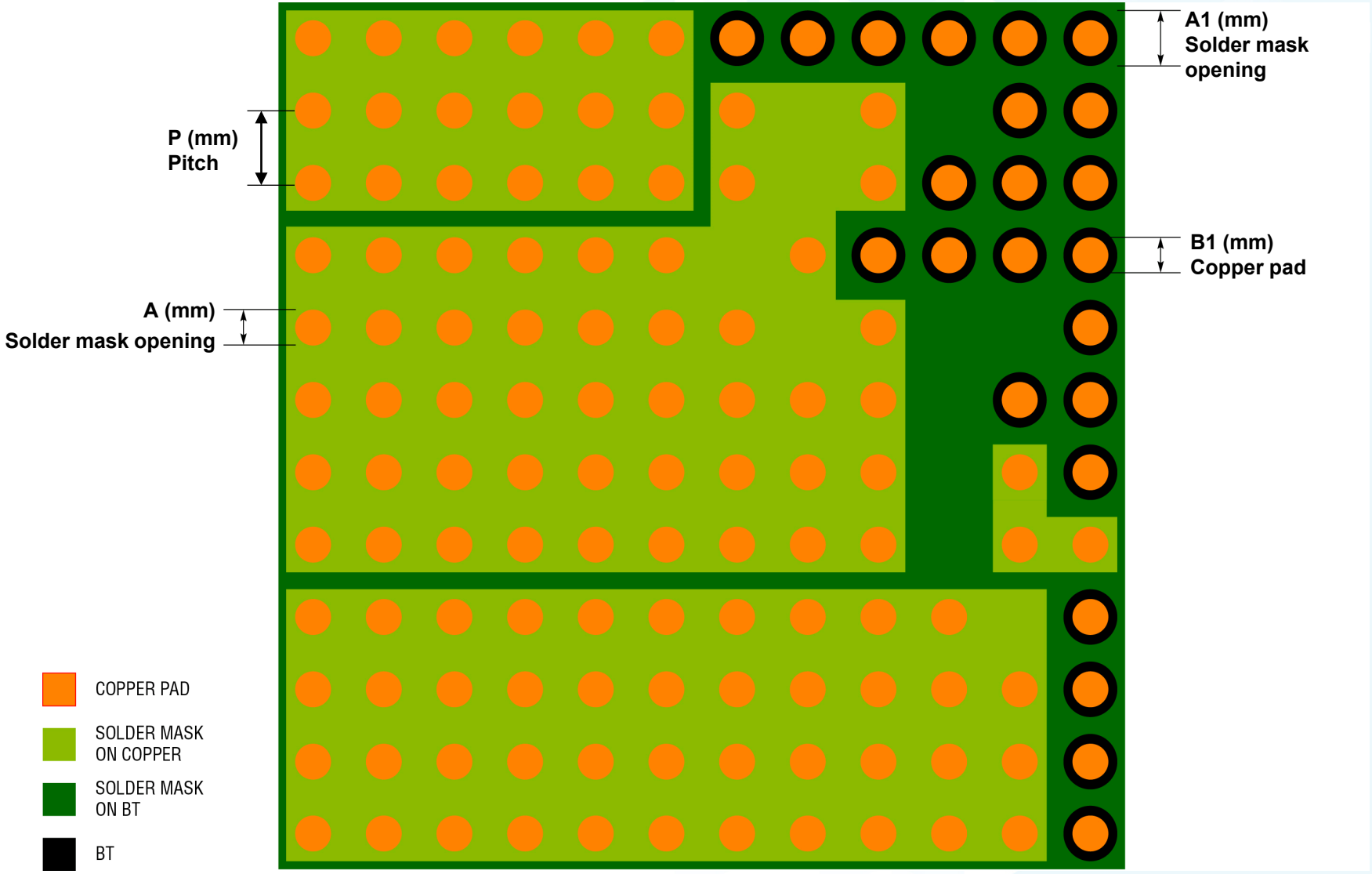
## Recommended SMD Pad Layout




**Refer to Table 3 & 4**

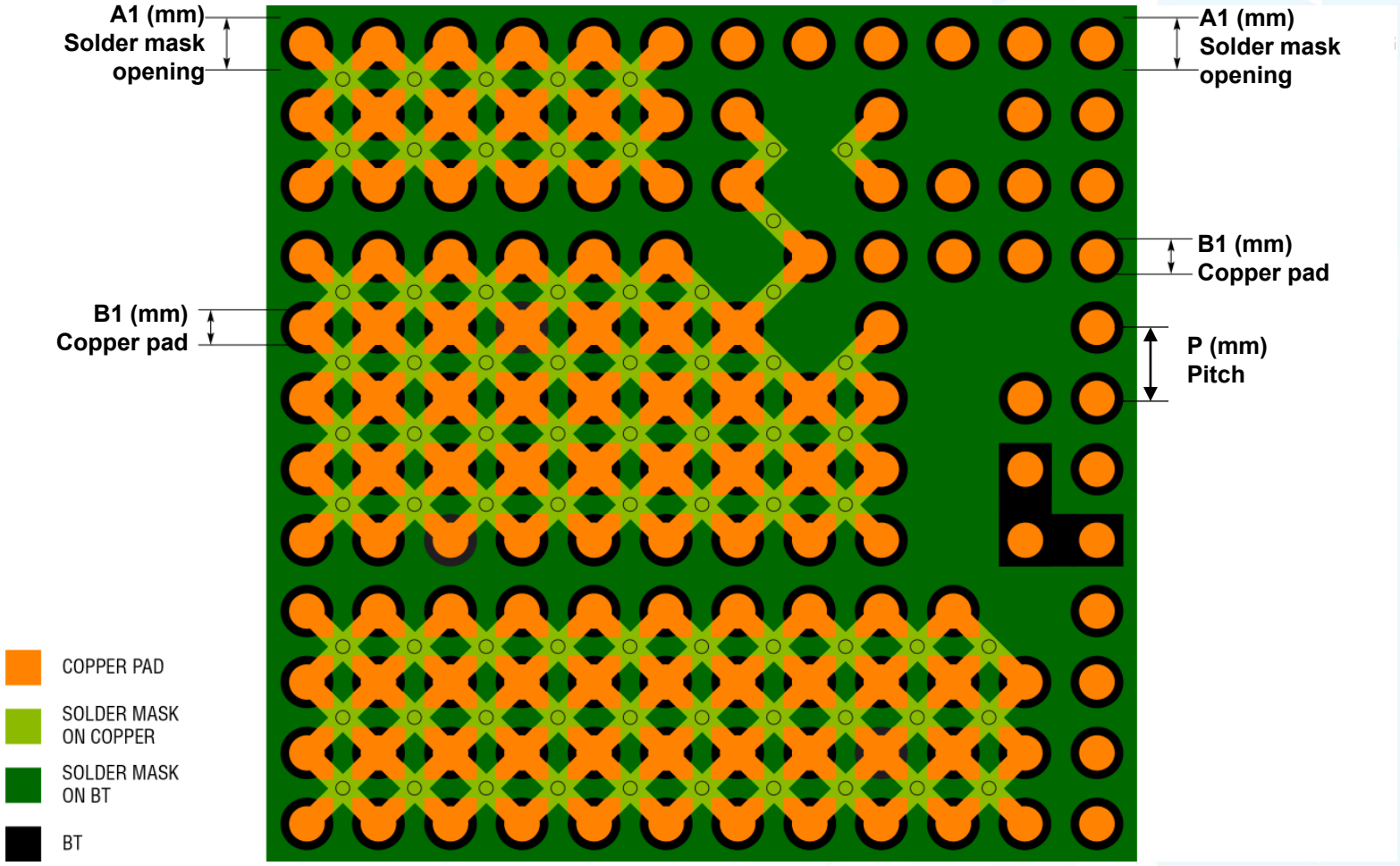
# BGA & LGA Mixed Round Pads

## Recommended SMD Plane and NSMD Signal Pad Layout



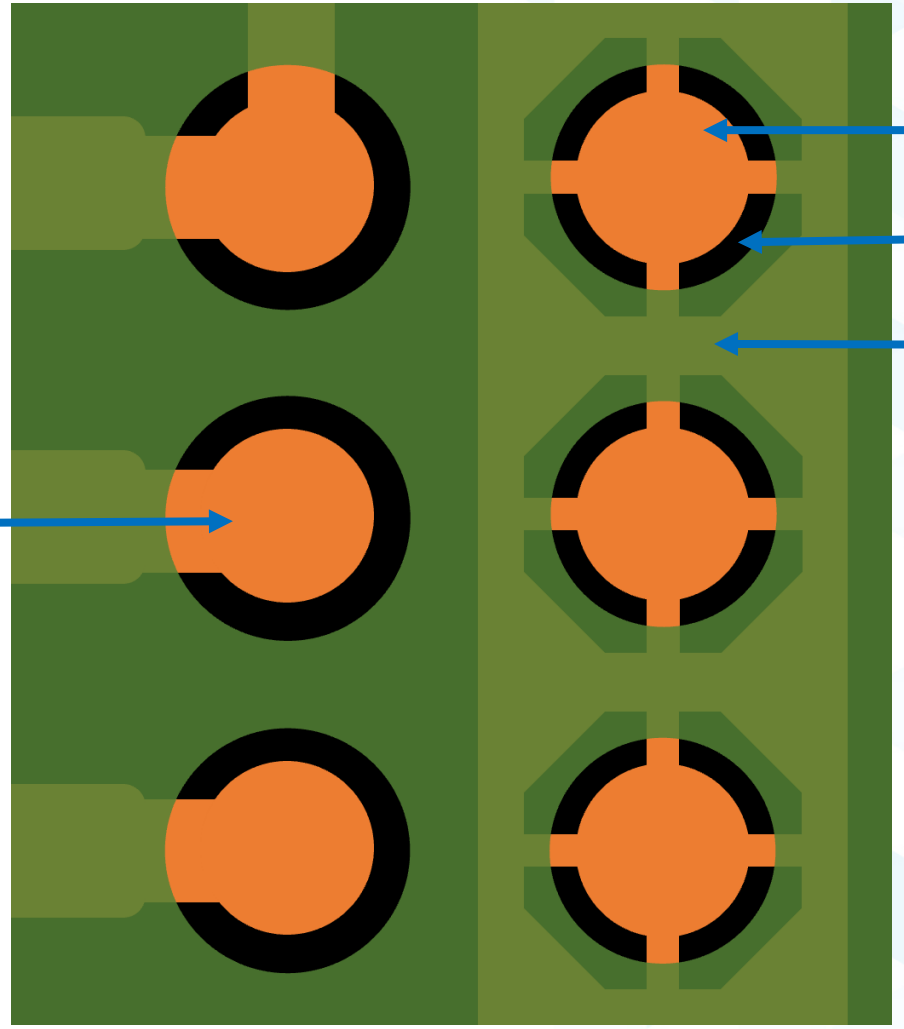

**Refer to Table 3 & 4**

# BGA & LGA Non-Solder Mask Defined Round Pads
## (Interstitial vias- thermal relief)
### Recommended NSMD with Thermal Relief PCB Pad Layout



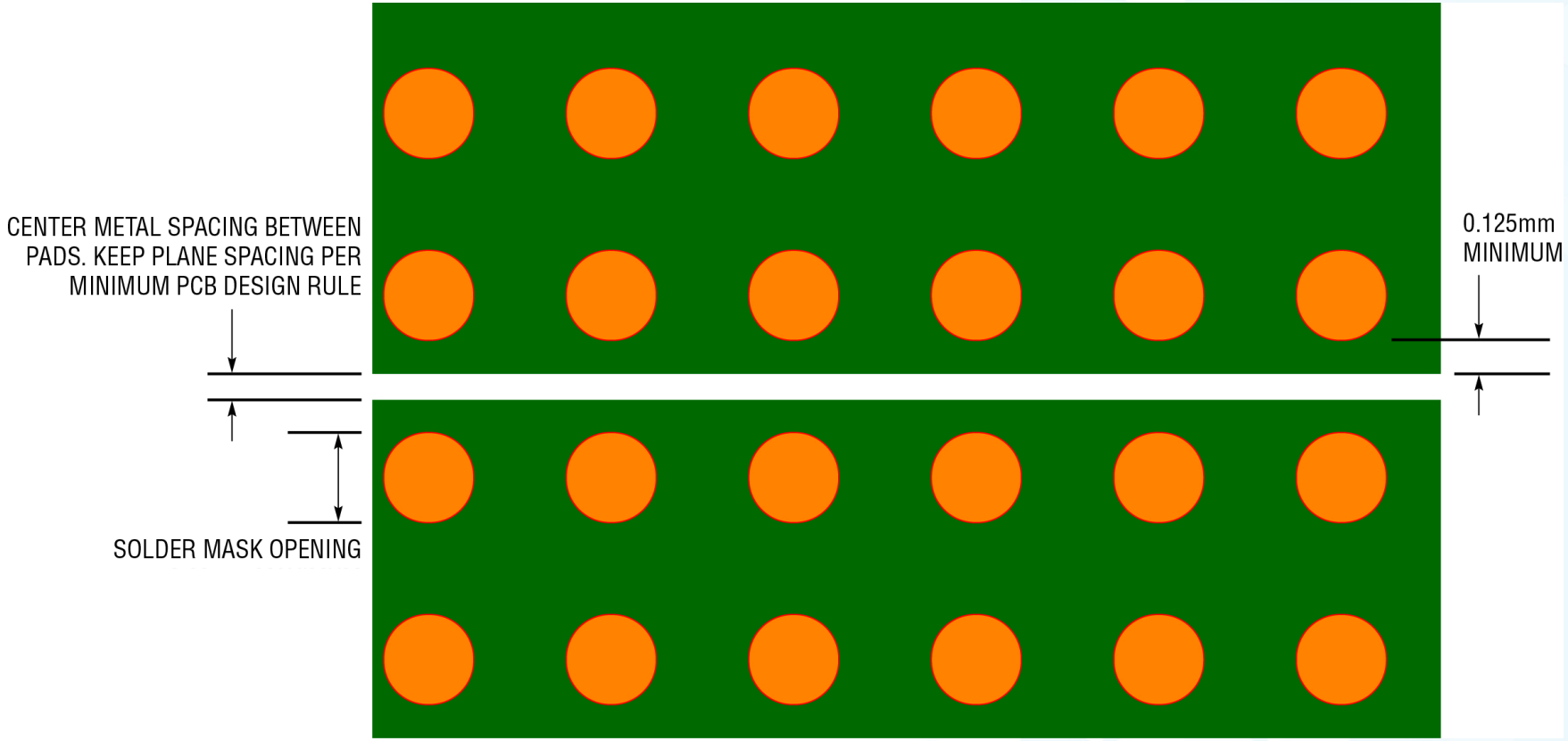

**Refer to Table 3 & 4**

# BGA & LGA Isolator Products Pad Design Guideline
# LTM28XX, LTM9100



# BGA & LGA PCB Plane Separation Guideline

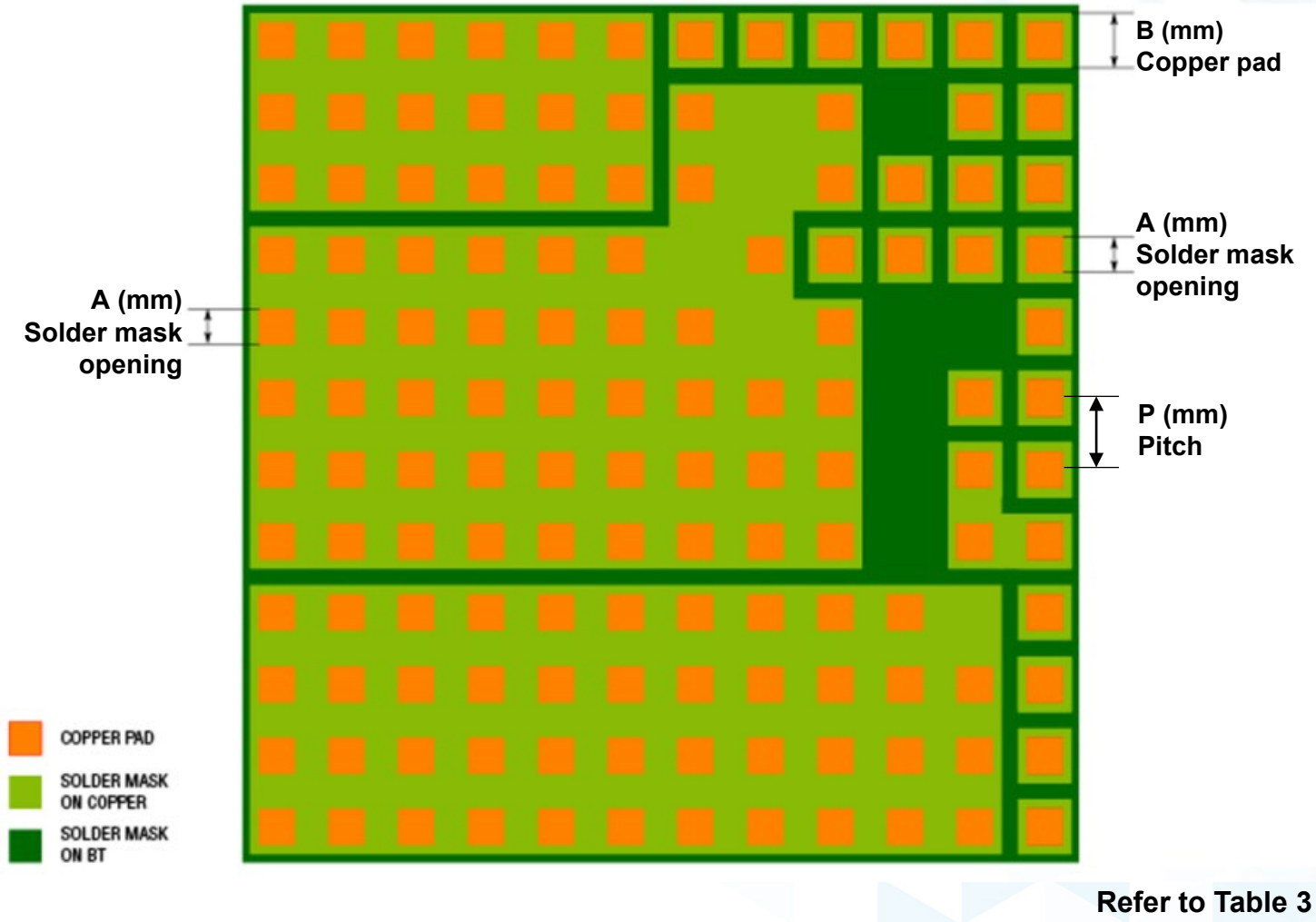
Maximum solder mask opening for plane separation needs to be controlled (≥0.8mm pitch)



# PCB Design Guidelines – Square Pads for LGA Package

# LGA Solder Mask Defined Square Pads

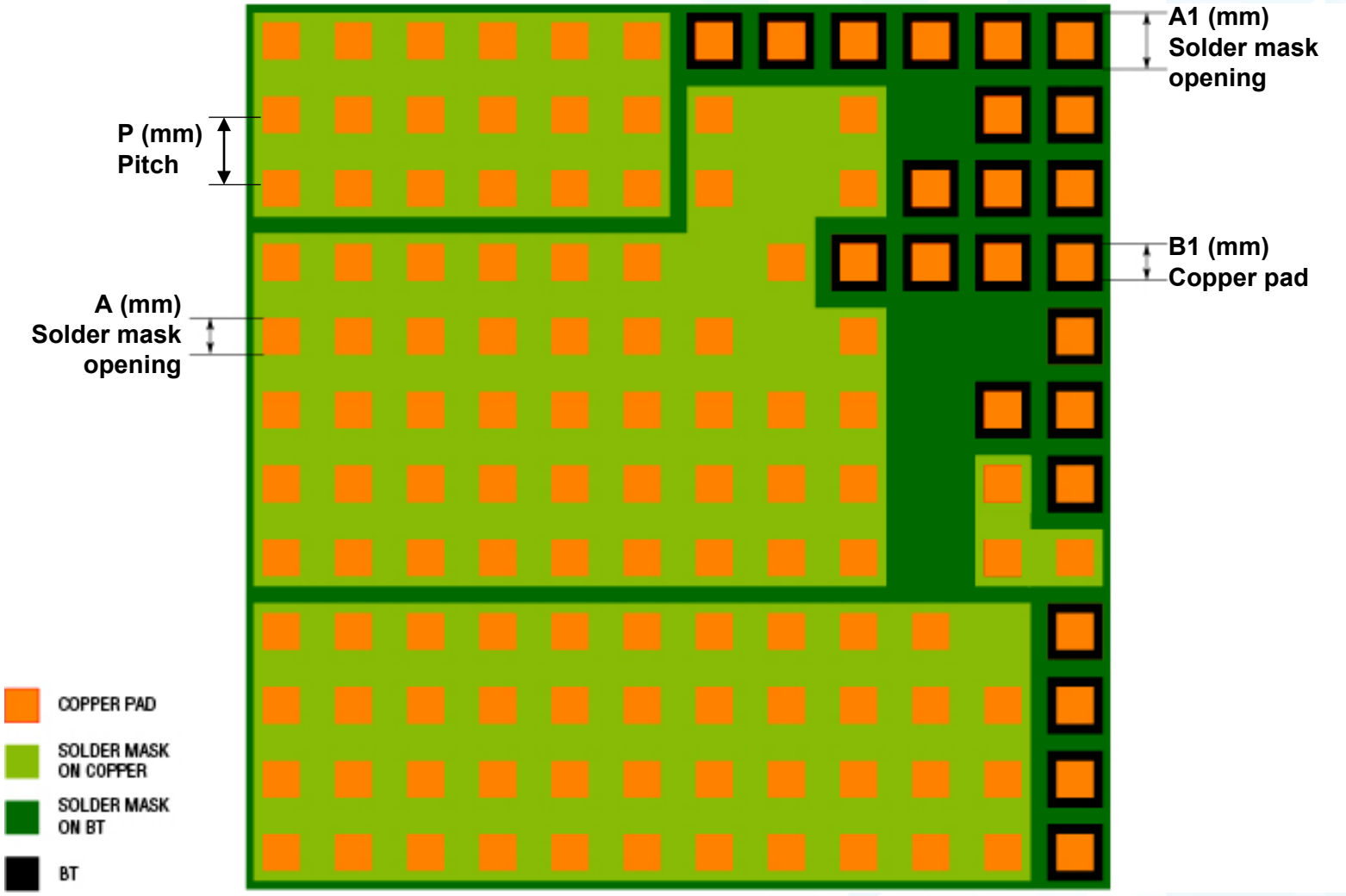
## Recommended SMD Pad Layout




**Refer to Table 3 & 4**

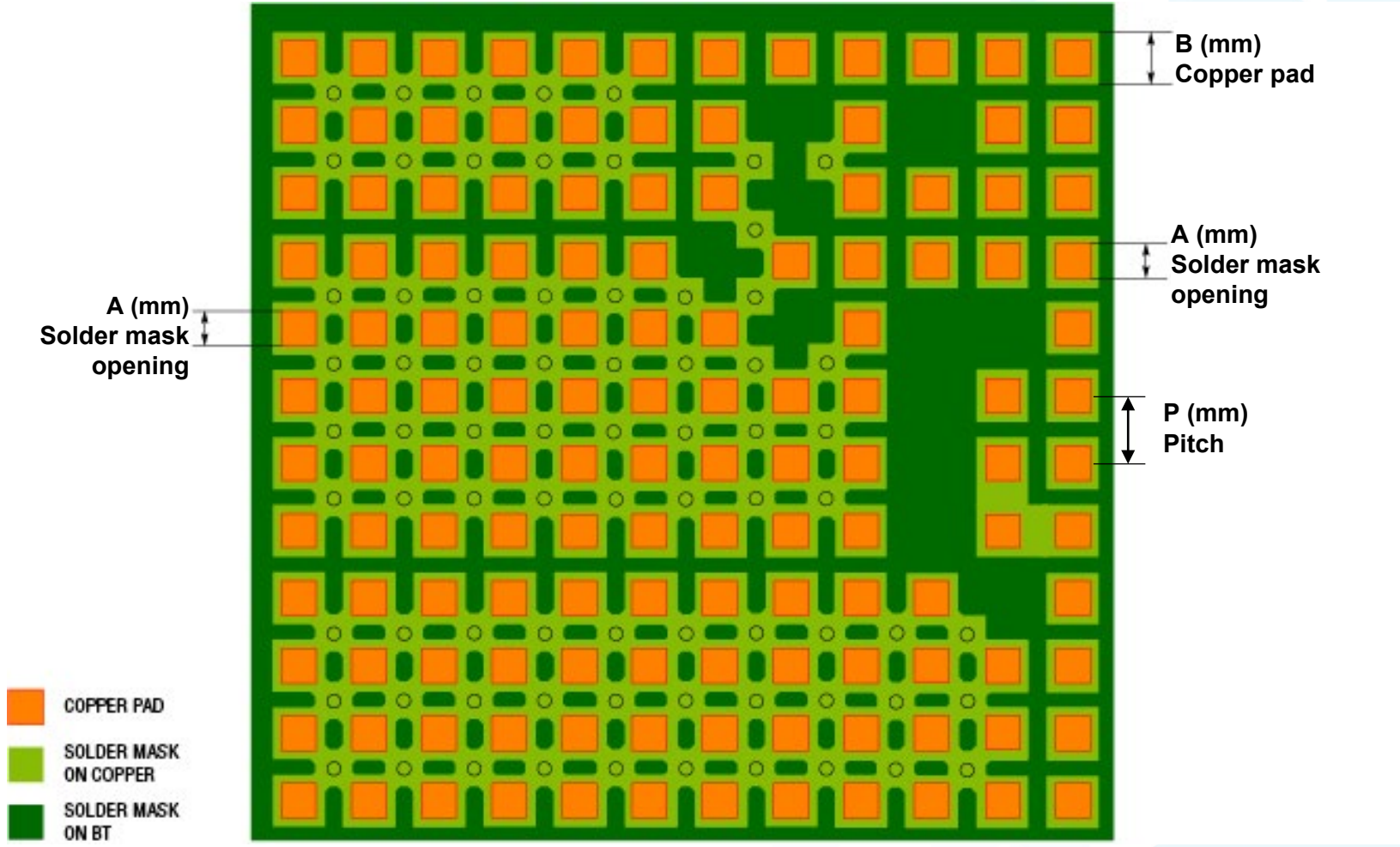
# LGA Mixed Square Pads

## Recommended SMD Plane and NSMD Signal Pad Layout



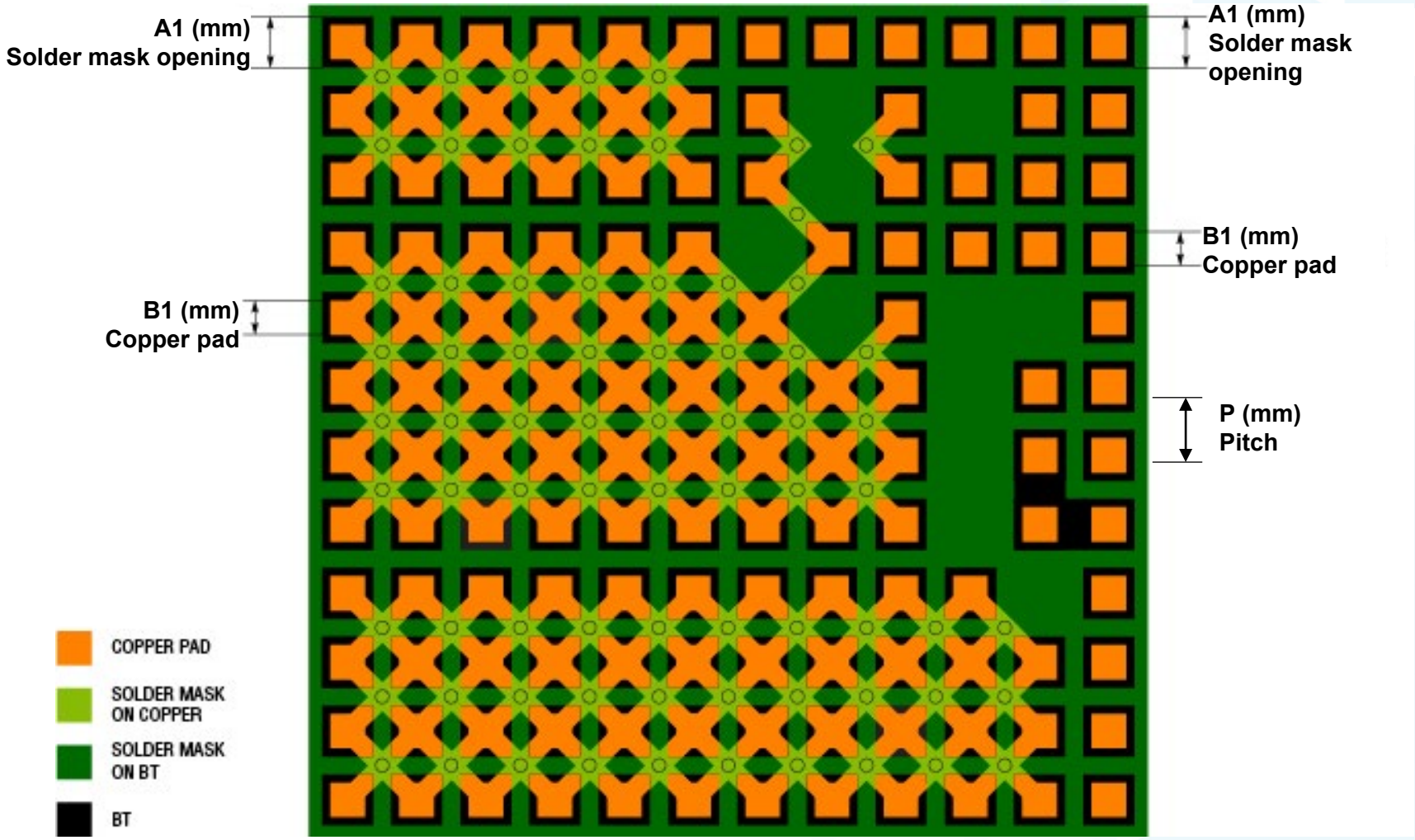

**Refer to Table 3 & 4**

# LGA Solder Mask Defined Square Pads
## (Interstitial vias-thermal relief)
### Recommended SMD with Thermal Relief PCB Pad Layout




**Refer to Table 3 & 4**

# LGA Non-Solder Mask Defined Square Pads
## (Interstitial vias- thermal relief)
### Recommended NSMD with Thermal Relief PCB Pad Layout




Refer to Table 3 & 4

# Moisture Sensitivity Considerations for BGA and LGA Package

# BGA & LGA Moisture Sensitivity, Bake, Pack, & Ship

- µModule® products meet MSL 3 or 4 of the IPC/JEDEC specification J-STD-020
  - Floor life (time outside moisture barrier bag) must be tracked
  - Floor life for MSL 3 = 168 hours
  - Floor life for MSL 4 = 72 hours
- ADI ships all µModule® devices in bake-able trays with desiccant and moisture level indicator inside vacuum sealed moisture barrier bag
- Check the packing integrity (may need to check the source of shipment for repack procedures) if parts received in partial trays (other than from ADI)
- If moisture indicator shows pink color, or punctured seal of the bag is observed, bake the packages at 125°C for 48 hours minimum
- Any balance of a used lot or partially used tray should be vacuum sealed with desiccants and Humidity Indicator Card (HIC) in moisture barrier bag with clear indication of accumulated floor life.
  - When vacuum seal is not available, bake packages at 125°C for 48 hours minimum before reflow process.
  - If the µModule® product is already mounted on a board and is expected to be subjected to a second reflow, the floor life time still needs to be considered. Bake the board at 125C for 48 hours minimum prior to the second reflow if the accumulated floor life time of the µModule® product exceeds its MSL rating.

# Solderability Test for LGA Package

# LGA Solderability Test

- No solderability test on ADI μModule® BGA products
- ADI μModule® LGA products cannot be checked for solderability using the solder dip method
- Solder paste needs to be screened onto the LGA pads and the part taken through reflow furnace (Surface mount Process simulation test per JEDEC spec “Solderability” JESD22-B102)

# Board Assembly Process for BGA and LGA Package

# BGA Stencil Design

- Stainless steel laser cut stencils recommended
- Recommend slightly smaller stencil aperture than the pad opening (especially for SMD pads)

| Table 5 : Recommended Stencil Dimension | | | |
|---|---|---|---|
| **BGA Pad Pitch, (mm)** | **Pad Opening, (mm)** | **Stencil Opening, (mm)** | **Stencil Thickness, (mils)** |
| 0.65 | 0.35 | 0.33 | 4 |
| 0.80 | 0.40 | 0.38 | 4 |
| 1.00 | 0.50 | 0.48 | 4 |
| 1.27 | 0.63 | 0.60 | 4 |

# LGA Stencil Design

- Stainless steel laser cut stencils recommended
- Recommended stencil design parameter refer to table below
  - Corner radius of 0.06 mm on the aperture recommended

| Table 6 : Recommended Stencil Dimension | | | |
|---|---|---|---|
| **LGA Pad Pitch, (mm)** | **Pad Opening, (mm)** | **Stencil Opening, (mm)** | **Stencil Thickness, (mils)** |
| **SQUARE PADS** | | | |
| 1.27 | 0.63 | 0.60 | 5 |
| 1.27 (LTM4604) | 0.89 | 0.84 | 5 |
| 1.27 (LTM4608) | 0.76 | 0.72 | 5 |
| **ROUND PADS** | | | |
| 0.65 | 0.35 | 0.33 | 4 |
| 0.80 | 0.40 | 0.38 | 4 |
| 1.00 | 0.50 | 0.48 | 5 |
| 1.27 | 0.63 | 0.60 | 5 |

# BGA and LGA Screen Print

- Solder Paste
  - Low voiding paste
  - Type III or IV
  - Paste types used at ADI or by our customers include, but are not limited to,
    - Sn/Pb : Kester 531, AIM WS483, Alpha OM-5300
    - Pb free (SAC305) : No Clean Kester 907, AIM NC254 , Indium SAC - 5.1AT, 5.8LS, Alpha OM-325, OM-338T
    - Pb free(SAC305) : Water Soluble Kester 520A, AIM SAC-WS353, Indium 3.2
- Check print definition, cleaning frequency
  - Stencil clogging can show as partial solder joints, not well defined joints

# LGA µModule® Placement Additional Guidelines

- Typical placement systems used for any BGA package are acceptable
- The LGA part needs to be pushed into the solder paste to achieve good contact of solder paste onto the LGA pads (pads are solder mask defined)
  - Adjustments to the setup on placement systems are done using force as a variable or Z height from the PCB as reference
  - Need to find the correct setting so that paste is not squeezed out of the pad, but at the same time ensure the contact with the LGA pad
    - Can be checked with X-ray before and after screen printing (before reflow)
  - This ensures good solderability and less voiding

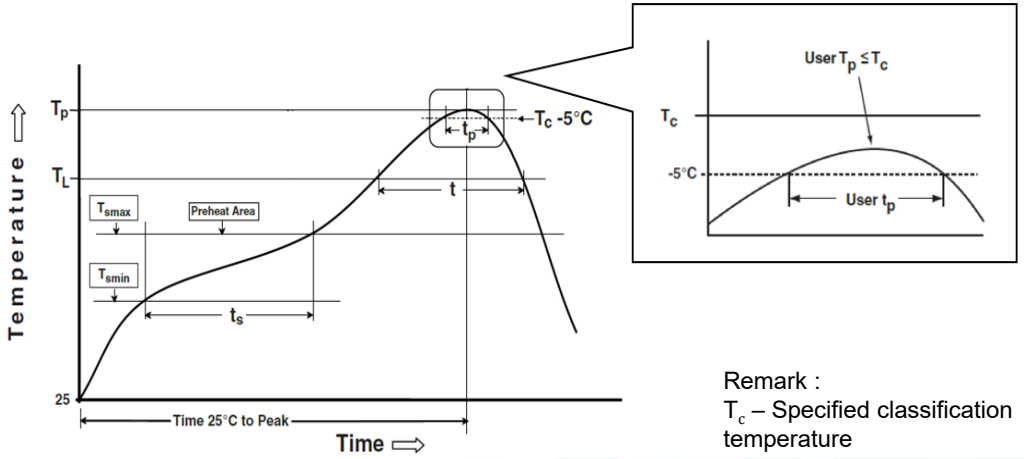
# BGA and LGA Reflow Profiles

- Recommend using a 9 zone or greater oven with oxygen level control
- Profile with all components (fully populated board) and thermocouples under and on top of the µModule® devices.
- µModule® devices are not allowed to be reflowed more than 3 times (including rework).
- Check that the solder paste vendor recommended profile conforms to ADI recommendations
  - If the ADI recommended profile cannot be met, adhere to the paste vendor profile except peak reflow temperature; Peak reflow temperatures must not exceed the temperatures listed in Table 7
- All µModule® devices are designed to withstand Pb-Free peak reflow temperature

**Table 7: Pb- Free Process - Package Peak Reflow Temperature**

| PACKAGE THICKNESS | VOLUME < 350 mm³ | VOLUME 350 mm³ - 2000 mm³ | VOLUME ≥ 2000 mm³ |
|---|---|---|---|
| < 1.6mm | 260 +0/-5 °C | 260 +0/-5 °C | 260 +0/-5°C |
| 1.6mm - 2.5mm | 260 +0/-5 °C | 250 +0/-5 °C | 245 +0/-5 °C |
| ≥ 2.5mm | 250 +0/-5 °C | 245 +0/-5 °C | 245 +0/-5 °C |

# BGA and LGA Reflow Profiles (cont'd)




Remark :
$T_c$ – Specified classification temperature

| Table 8 : Reflow Profile | | | |
|---|---|---|---|
| **Profile Feature** | | **Lead-Free Solder** | **Leaded Solder** |
| **Pre-heat** | **Min Soak Temperature Min ($T_{Smin}$)** | **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 (t)** | **30-90 seconds** | **30-90 seconds** |
| **Peak Package Body Temperature ($T_p$)** | | **See Table 7** | |
| **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 (a lower rate is recommended)** | **2.5°C/second max ( a lower rate is recommended)** |
| **Time 25°C to peak temp** | | **8 minutes max.** | **8 minutes max.** |

# BGA and LGA Solder Joint Void

- ADI/IPC recommends using a 25% maximum void criteria for solder joints
- If the recommended ADI or paste vendor profile results in >25% voiding, then use a longer soak profile during reflow (Profile with Thermocouple underneath the BGA)
  - For Pb free paste, keep the soak time at 150°C to 200°C for 90 to 120 sec
  - For Sn/Pb paste, keep the soak time at 100°C to 150°C for 90 to 120 sec
- If the above soak times are outside the range of the paste vendor recommended profile, keep the soak time to the maximum allowed per the paste vendor

# BGA and LGA Cleaning

- No clean paste recommended
- Water soluble paste has been used successfully with the µModule® packages
  - Use a saponifier and/or DI water spray system is recommended

# Second Side Assembly Guideline for BGA and LGA Package

# Second Side Reflow Guidelines

- Analog Devices Inc. recommends that the suitability of all LGA and BGA µModule® devices be evaluated for second side reflow with consideration towards the PCB assembly equipment and process.
- Component Weight / Total Pad Area ≤ 0.0465 gr/mm$^2$ can be used as a guidance to determine component candidacy for second side reflow
- Second side reflow is not recommended for the following µModule® devices
  - Isolated µModule® devices:
    - LTM2881, LTM2882, LTM2883, LTM2884, LTM2885, LTM2886, LTM2887, LTM2889, LTM2894, LTM9100
  - Exposed Heatsink µModule® devices:
    - Not limited to the following devices: LTM4620, LTM4620A, LTM4633, LTM4634, LTM4637, LTM4650, LTM4650-1A, LTM4650-1B, LTM4650-2, LTM4650A, LTM4650A-1, LTM4652
  - All Component on Package (CoP) devices:
    - Not limited to the following devices: LTM4626, LTM4636, LTM4638, LTM4656, LTM4657, LTM4660, LTM4662, LTM4664, LTM4678, LTM4680, LTM4681, LTM4683, LTM4700, LTM4702, LTM4710-1, LTM4712
  - All µModule® devices greater than 4mm thickness.

# Rework for BGA and LGA Package

# Pre-Rework Process Guideline (μModule® device)

- Determine the failure mode from the board and at what operation the defect(s) occurred
  - Is it opens or shorts?
    - Opens : Check solder joint quality,  partial joint, no joint, cold solder
    - Shorts : 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

- Bake **PCB board assembly** for 48 hours at 125°C before removing μModule® packages.
  - If this step is not followed, there is a possibility of delamination of the mold compound from the substrate resulting in solder spread within the module.
  - This is critical to prevent damage to the devices adjacent to the rework component.

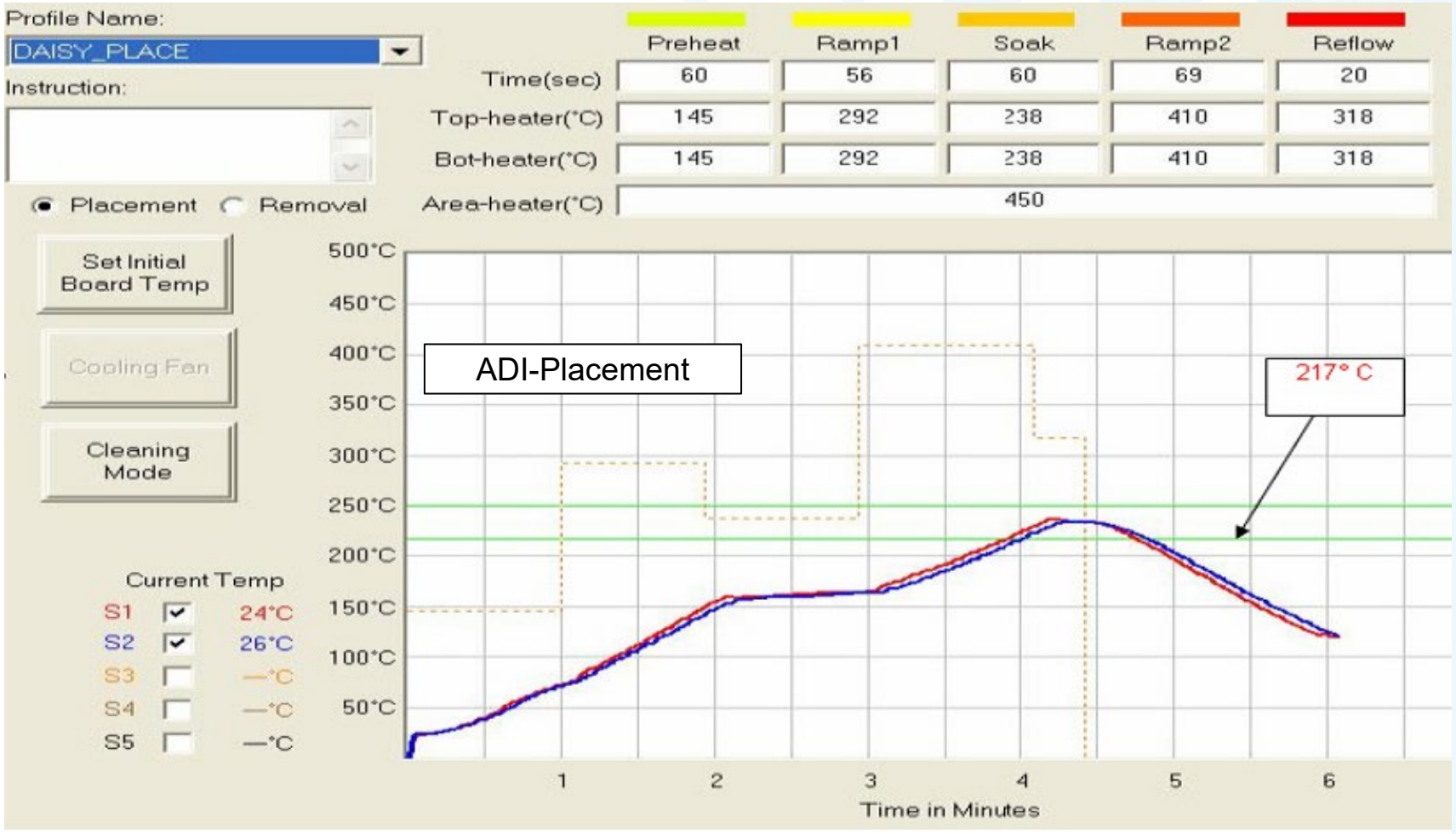
# Rework Component Removal (µModule® device)

- Use a BGA & SMD rework station capable of profiling the top and bottom of module
  - Handheld heat guns or IR-only rework stations should not be used
  - Use appropriate heat shielding of sensitive components in proximity to the µModule® package
  - The profile is done with a thermocouple on top of the part and another at the bottom of the part (close to the solder joints)
    - Maximum temperature for top of package = Refer to **Table 7**
    - Maximum bottom temperature (at solder joint) = 230°C to 245°C
      - Keep the bottom temperature as low as possible and increased time to melt the solder for package removal
      - Note: maximum top/bottom temperatures for Sn/Pb BGA can be set to 220°C
  - Ensure that the solder has reached above the liquidus temperature
  - If the solder is not completely molten, the PCB pads may be lifted during removal

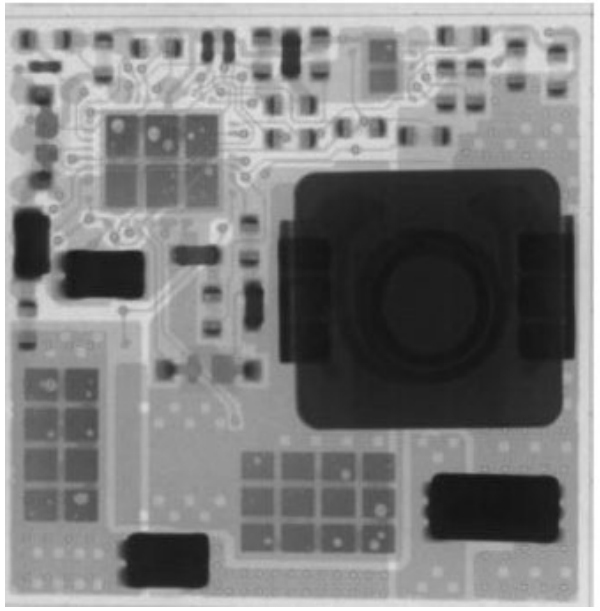
# Rework Component Reflow (μModule® device)

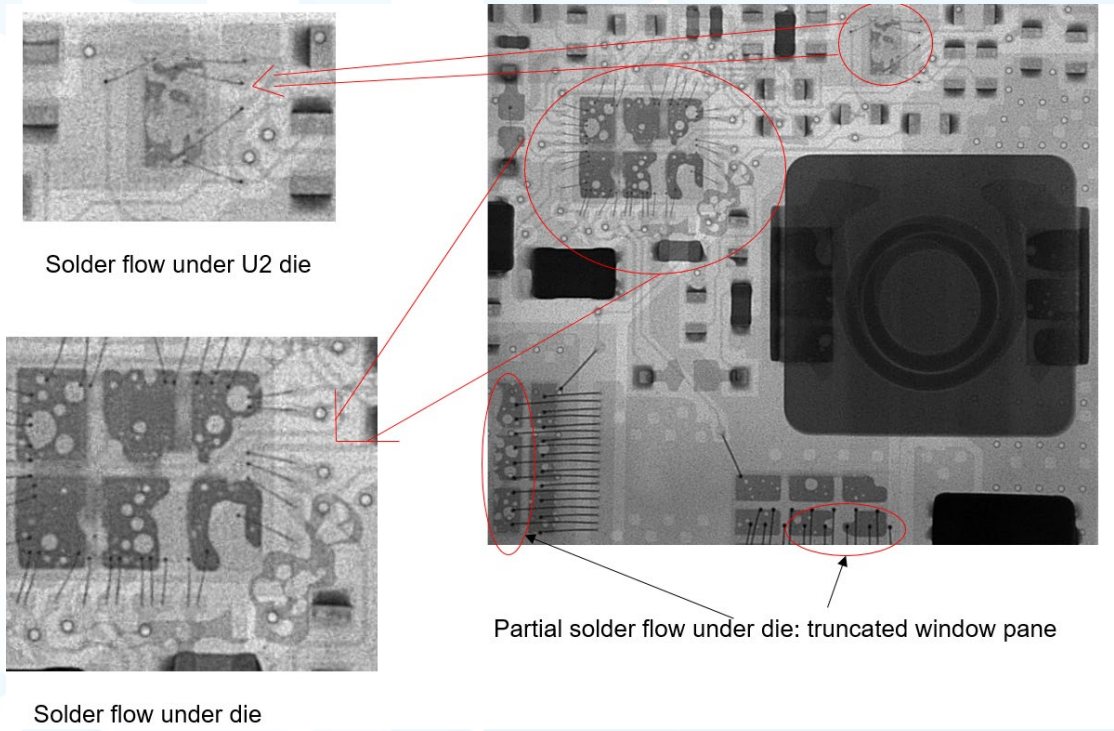
- Placement and reflow of new component
  - Removed component should not be reused
  - Reflow profile to ensure adequate soaking time as well as time above liquidus (refer to **Table 8**)
  - Profile with thermocouples on all modules which are connected in parallel
- Inspect solder joints using X-ray

# Rework Profile Example



# Example of good and bad µModule® Device after PCB assembly and rework






NO SHORTS

- Part met MSL floor life
- Reflow peak temperature within the spec
- Part was baked prior to removal

SOLDER SHORTS INSIDE MODULE

- Part exceeded MSL floor life – delamination
- Reflow peak temperature out of spec – solder melted & spread
- Part was not baked prior to removal

# BGA Package: Dual Component on Package Handling

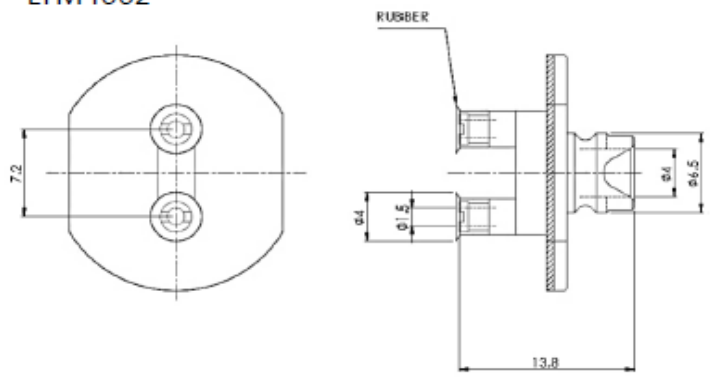
# SMT Nozzle, Pick n Place for Packages with Dual Component(s) on Top

## Type 1: Inductor Pick-up (Dual inductor design)

Nozzle design layout.

**Panasonic Dual Tip Vacuum Pad Nozzle (Pitching 7.2mm)**

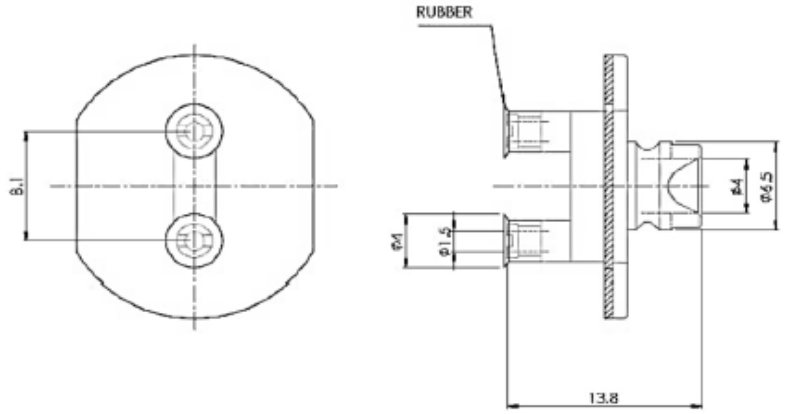
LTM4662



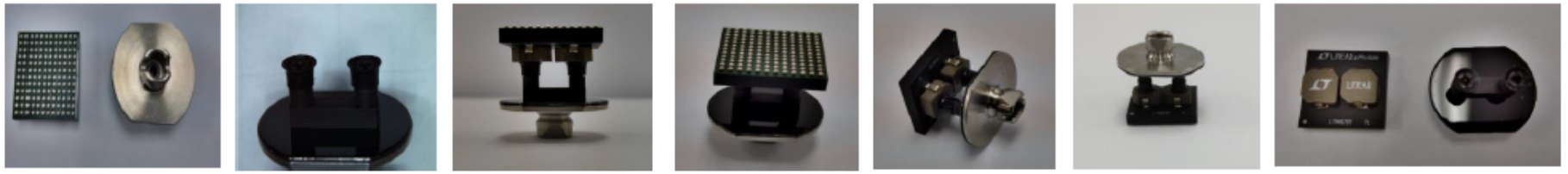

Bottom view Side view

**Panasonic Dual Tip Vacuum Pad Nozzle (Pitching 8.1mm)**

LTM4678

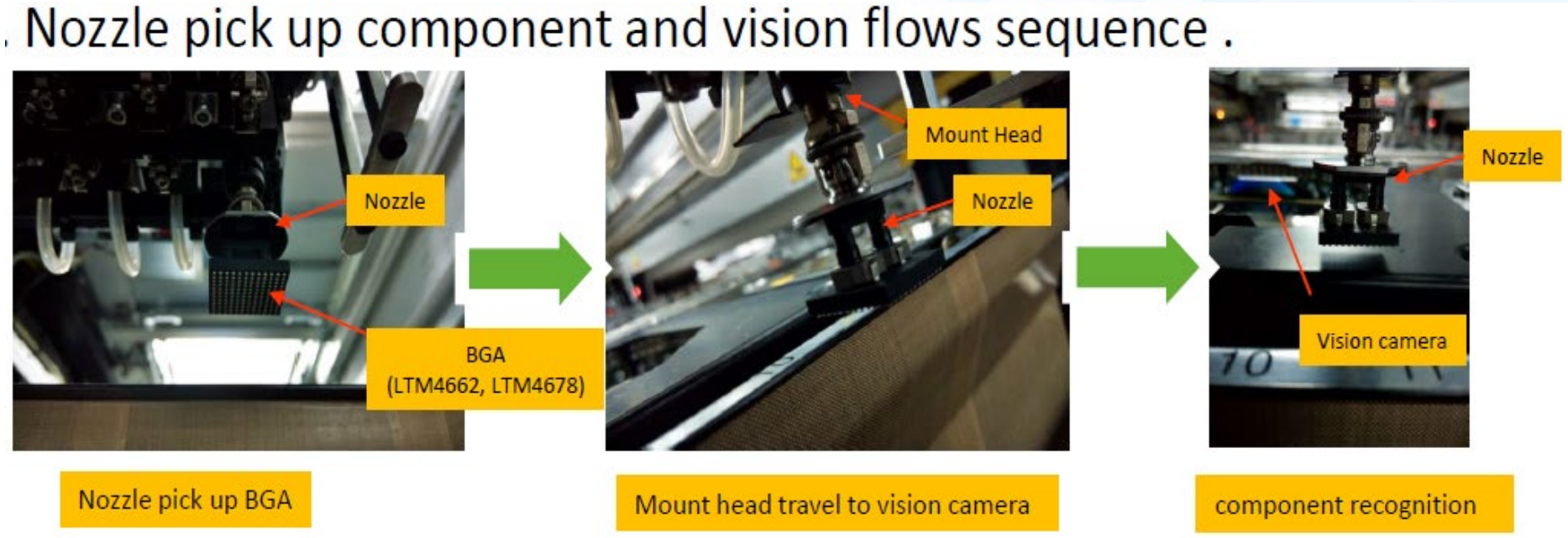



Bottom view Side view



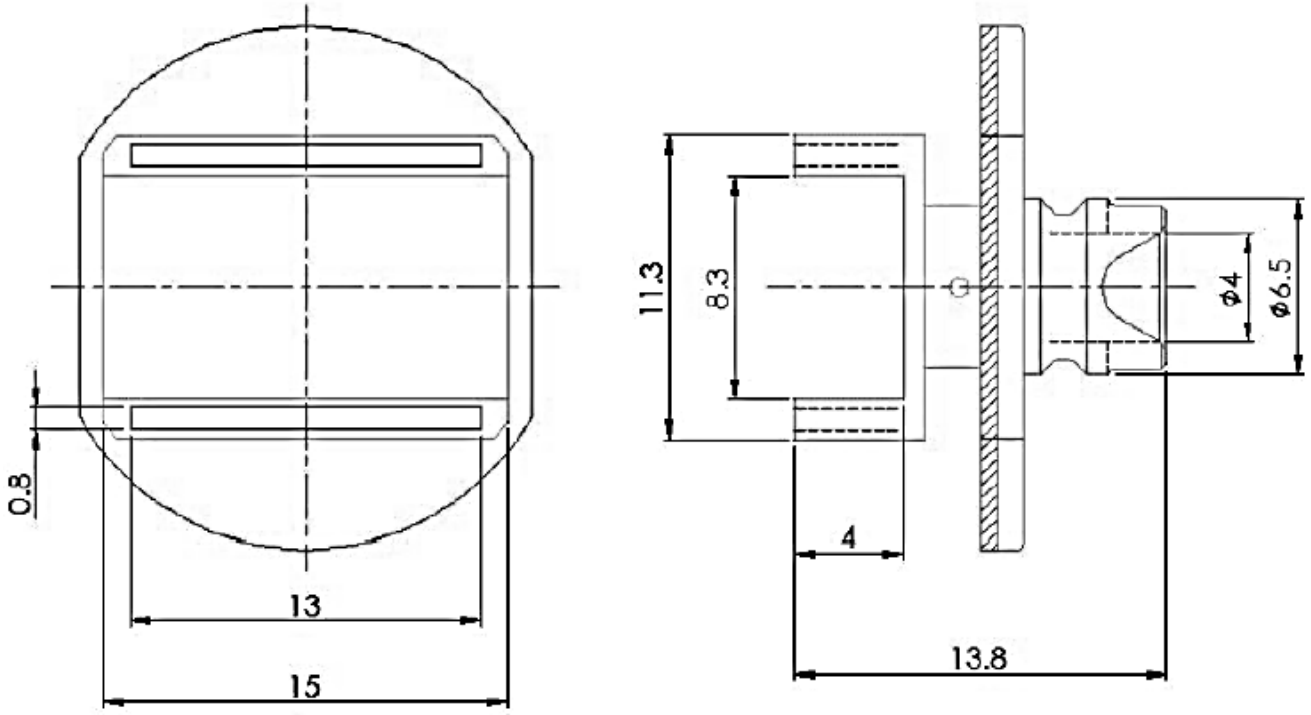
# SMT Nozzle, Pick n Place for Packages with Dual Component(s) on Top (cont'd)

## Type 1: Inductor Pick-up (Dual inductor design)



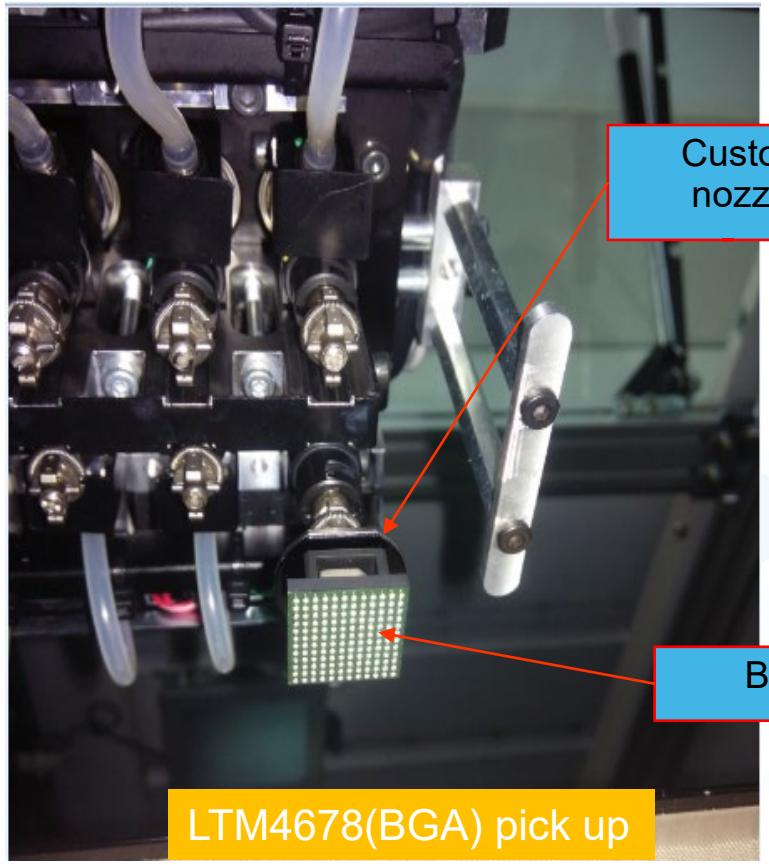
# SMT Nozzle, Pick n Place for Packages with Dual Component(s) on Top (cont'd)

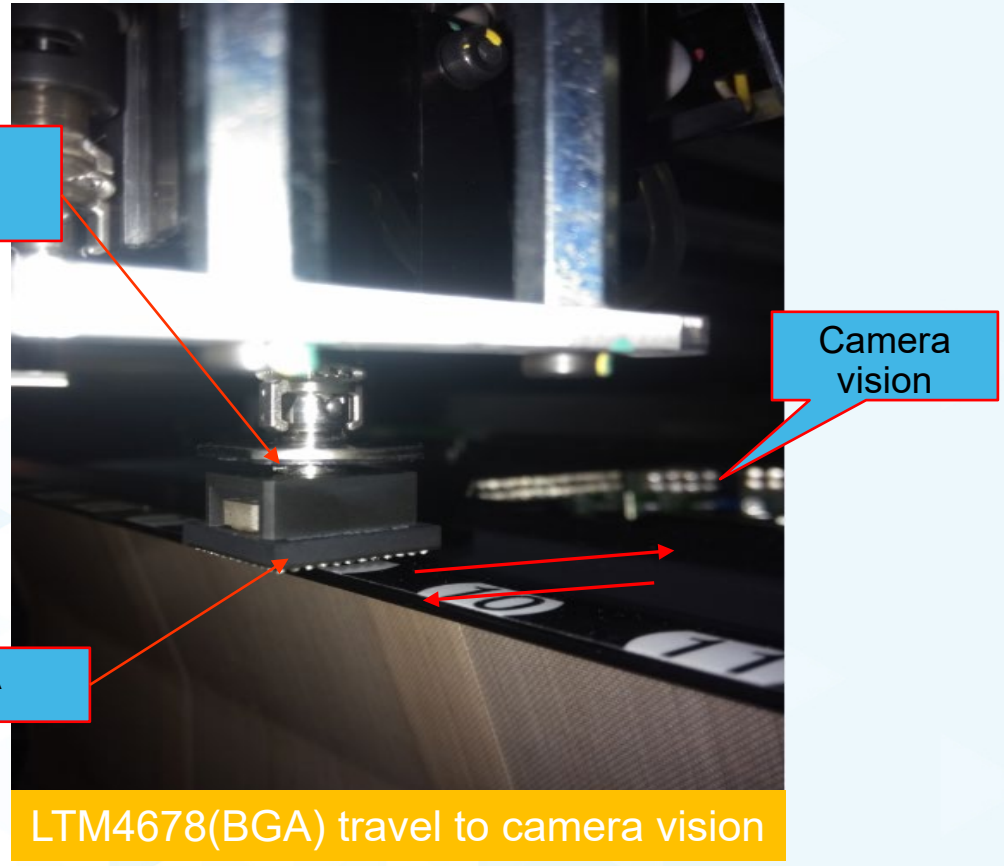
## Type 2: Package Body Pick-up



# SMT Nozzle, Pick n Place for Packages with Dual Component(s) on Top (cont'd)

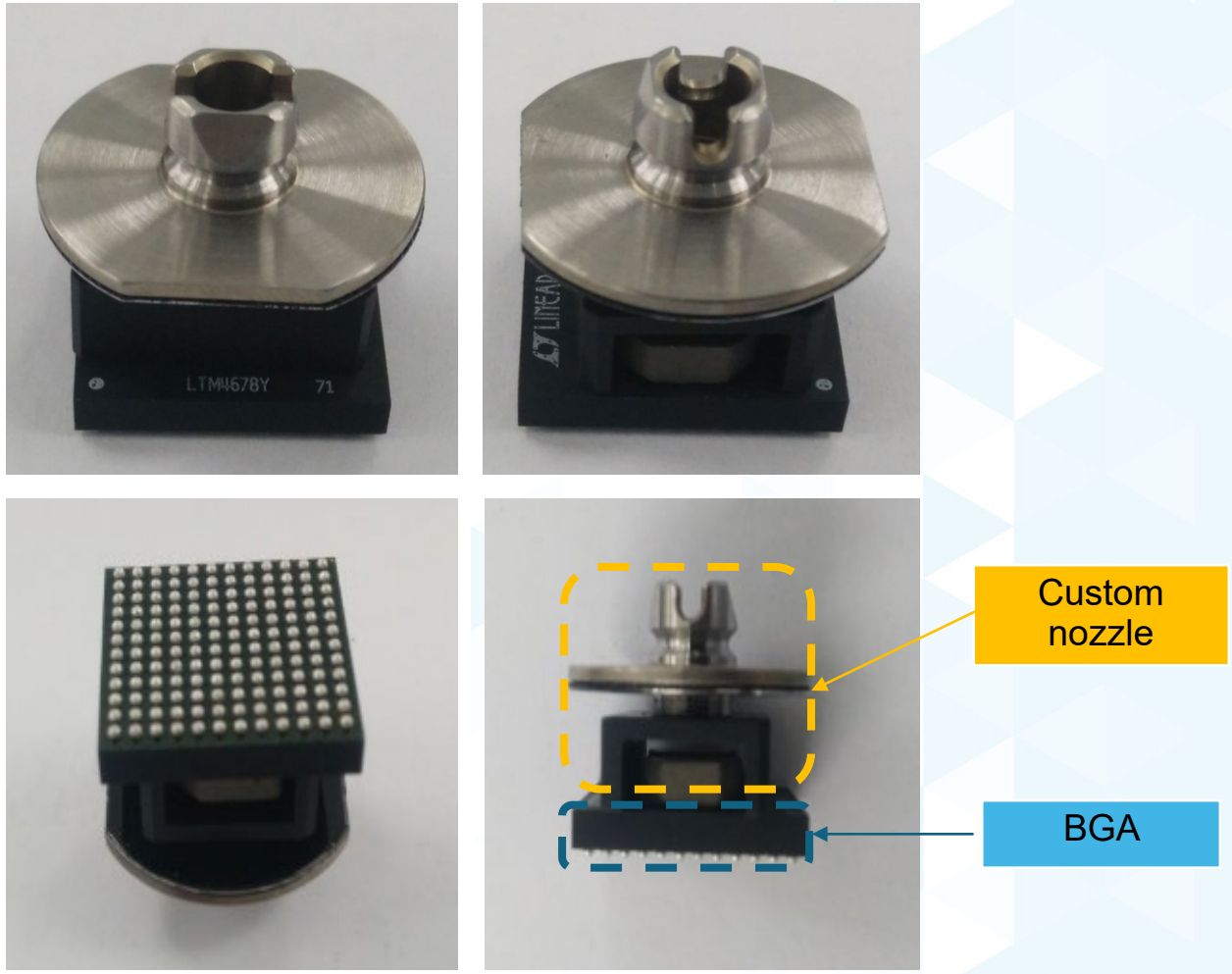
## Type 2: Package Body Pick-up



# SMT Nozzle, Pick n Place for Packages with Dual Component(s) on Top (cont'd)

## Type 2: Package Body Pick-up



# Maximum Mechanical Load for Heatsink Attach

# Maximum Mechanical Load for Heatsink Attach

- Mechanical load must be apply uniformly, perpendicular to the package top surface
- Mechanical load must be applied for a short period of time, for example heatsink attach
- PCB must be fully supported under the component area to prevent flexing of the PCB
- Maximum load 350 psi

# FAQ’s

# FAQs

1. What is the stencil opening and thickness?
   - See Stencil Design page (**Table 5 and 6**)
2. What type of paste to use?
   - Both no clean and water soluble are OK; Type III or IV
3. Can the PCB be cleaned effectively?
   - Yes; Both inline and rotary aqueous systems have been used to clean effectively
4. How to inspect for the solder joints?
   - 5DX is an effective method to check for solder joint shorts; YXLON/ Dage X-ray can also be used
   - Should be able to differentiate the PCB solder joints from the solder inside the µModule® device
5. Can the µModule® product be used on both sides of the PCB?
   - Except devices stated in “Second Side Reflow Guideline” slide
6. Why are parts shorting (Vin to Gnd or Vout to Gnd) ?
   - Check for solder joint shorts (check the schematic with the pin configuration)
   - Check inside the module (by X-ray) to see if any solder spreading has occurred

# FAQs

7. How to prevent shorting inside the module?
   - Check the floor life of the parts (From opening of sealed bag to reflow); If over 168 hours (MSL3) or over 72 hours (MSL4), parts need to be baked for min 48 hrs at 125°C
   - Was the bag not sealed or moisture indicator showing pink color? – Bake parts for min 48 hrs at 125°C
   - Was the reflow peak temperature greater than the peak temp for the module size (refer to **Table 7**)? If yes, redo profile to bring the peak temperature below the spec level for the package
   - Did the shorting happen after rework (removal)?
     - PCB must be baked for min 48 hours at 125°C before rework
     - Heat gun should not be used for removal
       - If heat gun was used, temperature may be excessive
     - Profile the rework station and remove component
8. Can the removed µModule® device be reused?
   - No, a fresh part must be used to replace the removed part.
9. What is the maximum amount of force that can be exerted on a module package?
   - Max 120 lbs of force based on package size 15 x 15 mm (full array 144 pin; 1.27mm pitch)
   - Refer to maximum mechanical load for heatsink attach slide for details

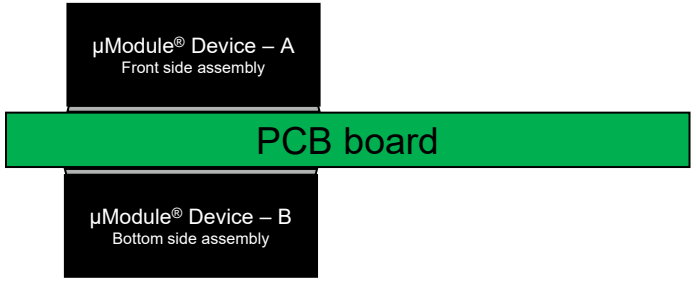
# FAQs

10. For 2 sided assembly, can the module on bottom side be placed right below the top side module?

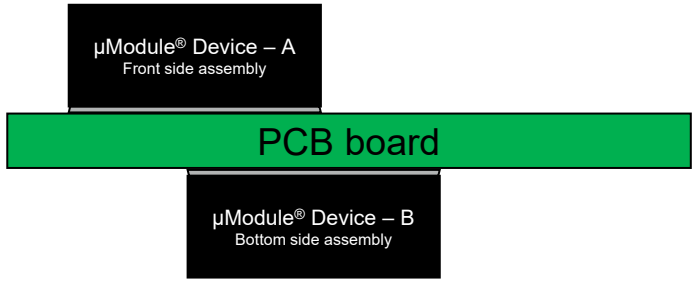
- No, do not attach a µModule® device under another µModule® device on the bottom side of PCB board. (Refer to diagrams below)

► Not recommended

Condition 1:




Condition 2:




► Recommended