Assembly Considerations for Linear Technology μModule™ LGA Packages

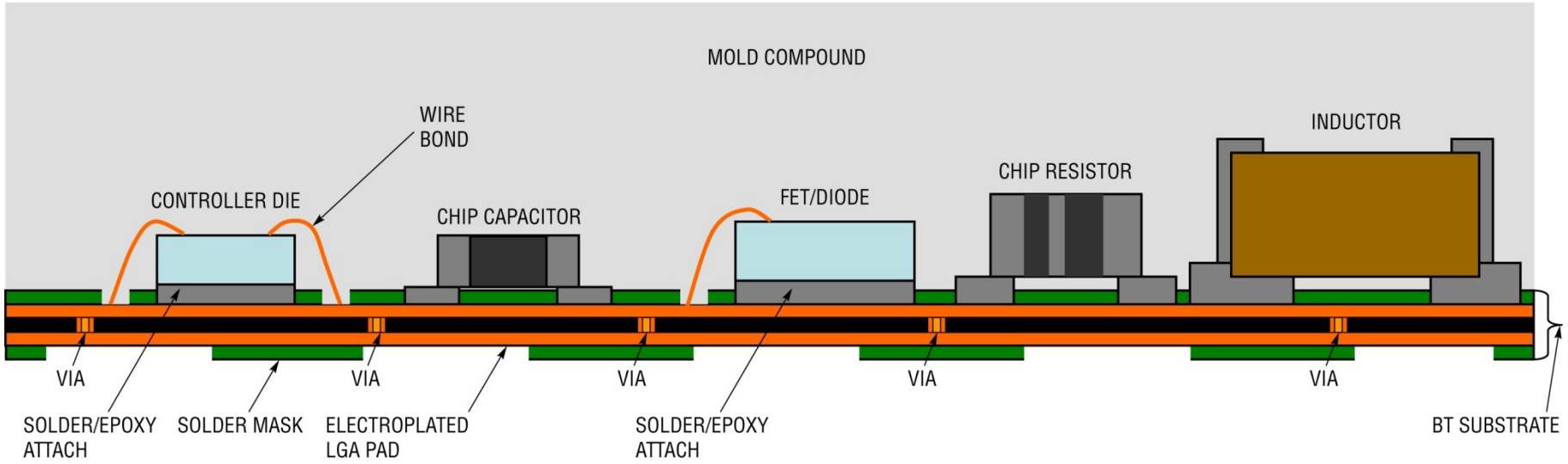
September 2010

Outline

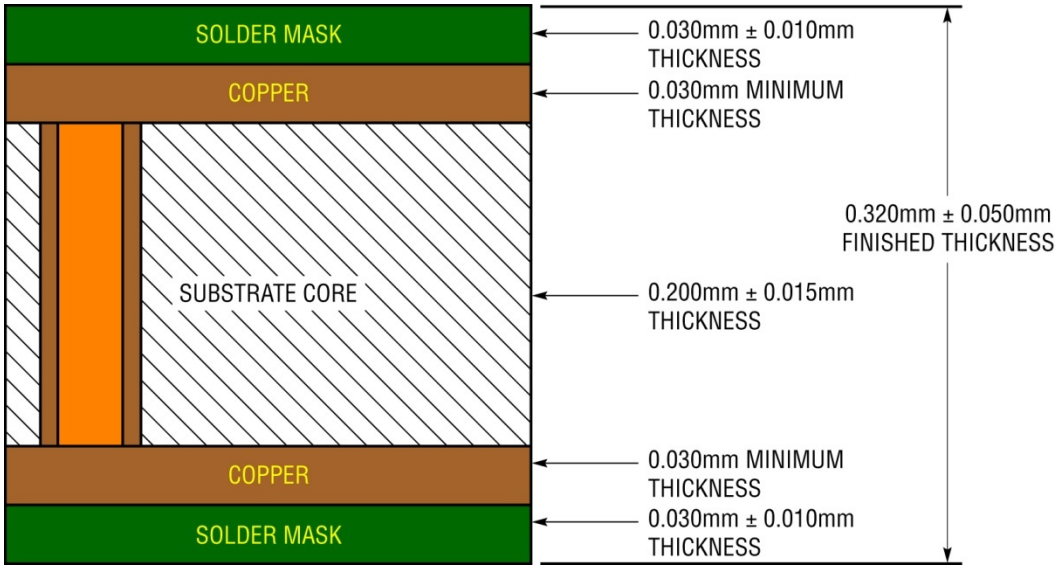
- Package Construction
- PCB Design Guidelines
- Moisture Sensitivity, Pack, Ship & Bake
- Board Assembly Process
  - Screen Print
    - Stencil Design
    - Solder Paste, Key Process Parameters
  - Placement
  - Reflow Profile
  - Cleaning
  - Removal and Rework
- FAQs

µModule™ LGA Package Construction

(Not To Scale)



Substrate Construction



All dimensions in mm

LTM4600 HIGH PERFORMANCE SUBSTRATE

Ni/Au Plating

SOLDER MASK = Taiyo ink PSR 4000

CORE = Mitsubishi Gas Chemical CCL-HL-832

Ni= 3 um minimum (5 um nominal)

Au = 0.3 to 0.8 um (0.5um nominal)

PCB Design Guidelines

- **µModule LGA pad**
  - All pads are solder mask defined (SMD)
  - 0.63 mm opening
  - 2 devices have larger pad openings
    - LTM4604: 0.889 mm (35 mils)
    - LTM4608 : 0.762 mm (30 mils)
  - Both devices have equivalent packages with standard pad size (LTM4604A, LTM4608A)
  - Recommend using LTM4604A and LTM4608A over LTM4604 and LTM4608 respectively for new designs

- **SMD vs NSMD pads on PCB**
  - NSMD pads preferred for signal pins
  - SMD OK to use

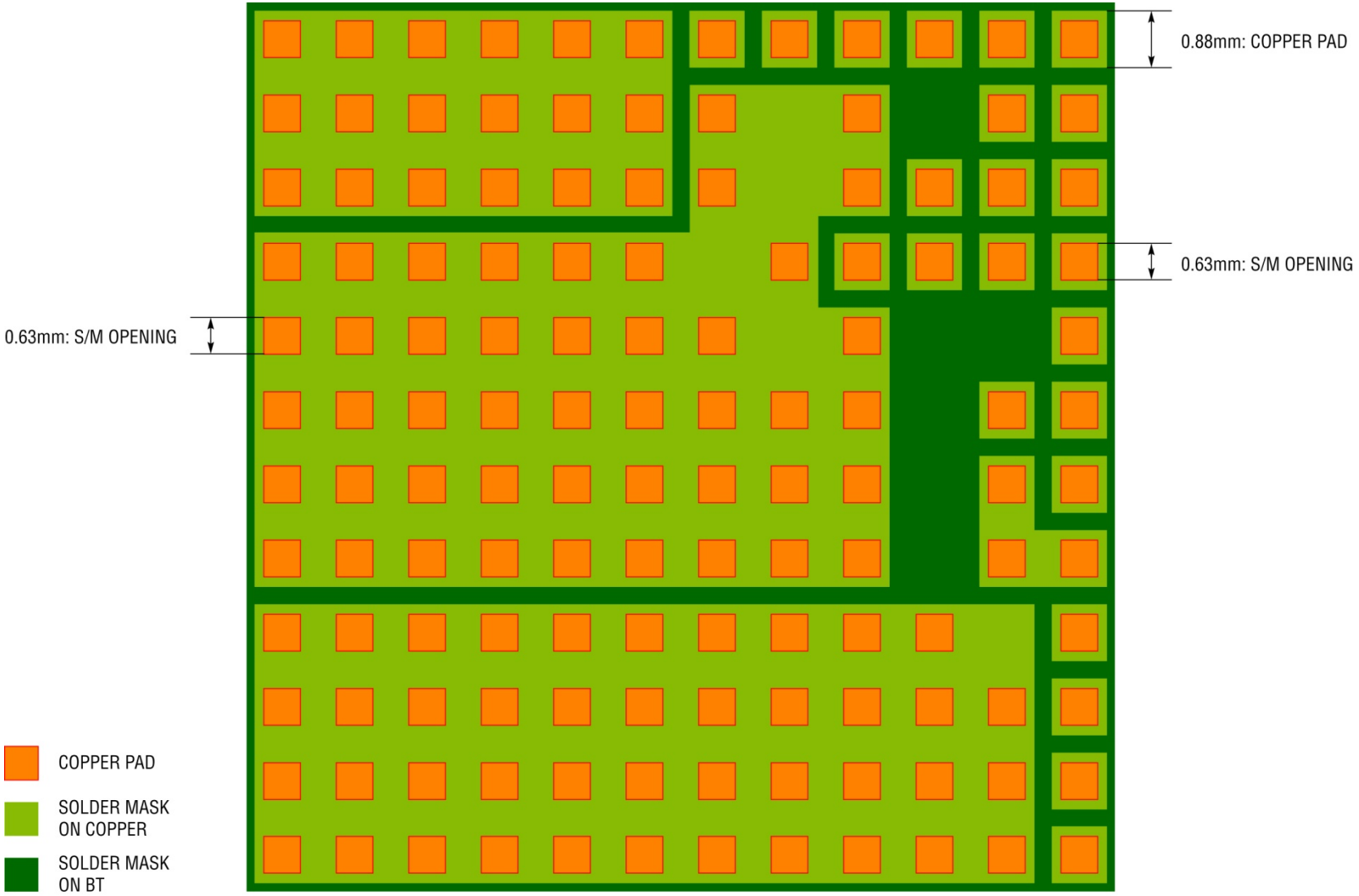
- **PCB Pad Layout (SMD Pads)**
  - Recommend using planes with SMD pads (same size as Package Pad opening 0.63 mm)
  - For signal pins (SMD)
    - Metal Pad Size 0.88 mm
    - Solder mask opening 0.63 mm
  - Vias between pads (on the planes) on top layer
  - No issue with reliability (all solder joint reliability data on µModule devices using SMD pads on package and PCB)

- **Non solder mask defined (NSMD) pads**
  - Recommended Pad Layout
    - 0.63 mm pad size
    - 0.83 mm minimum solder mask opening on NSMD pads
  - If some pads are NSMD and some are SMD (on planes), ensure that the SMD pad opening is 0.63 and NOT 0.83 mm – Refer to Figures in the next few pages

- **Pad finish on PCB**
  - OSP, ENIG recommended
  - Immersion Ag
    - Check for any dendritic growth with moisture
  - Immersion Sn
    - Oxidation issues

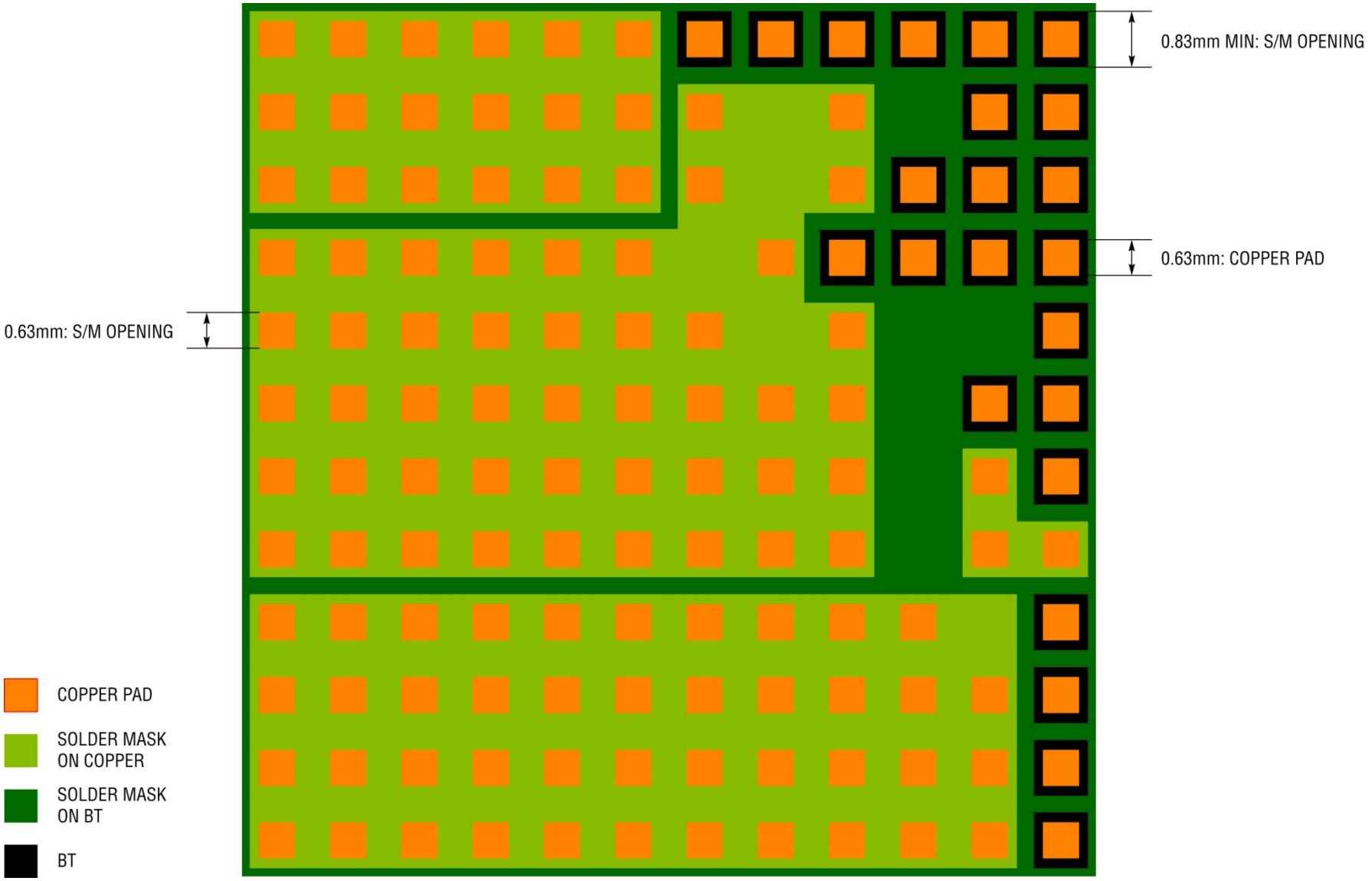
Solder Mask Defined Pads

Recommended PCB Pad Layout

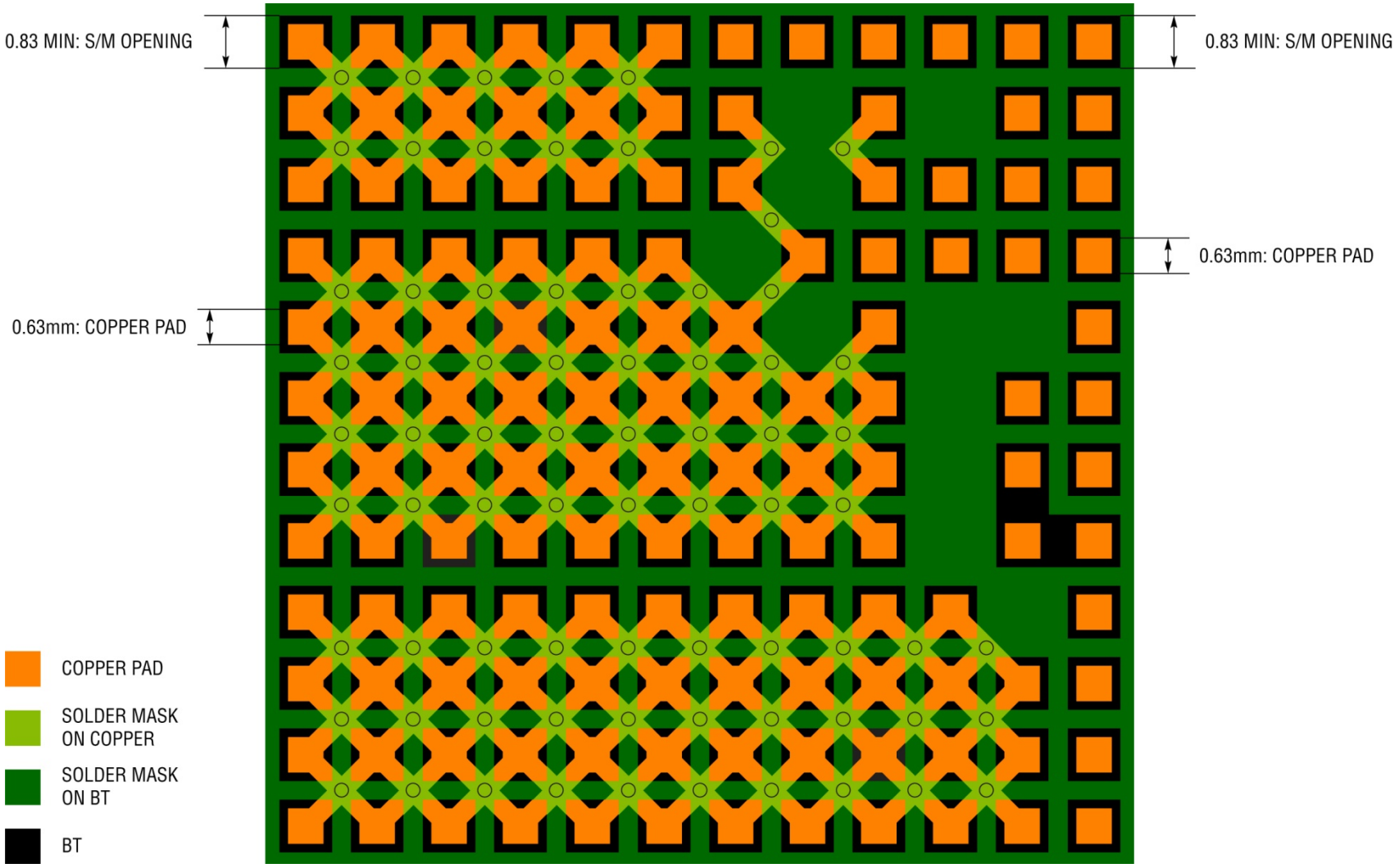


(Also see PCB Plane Separation Slide)

Mixed Pads (SMD and NSMD)



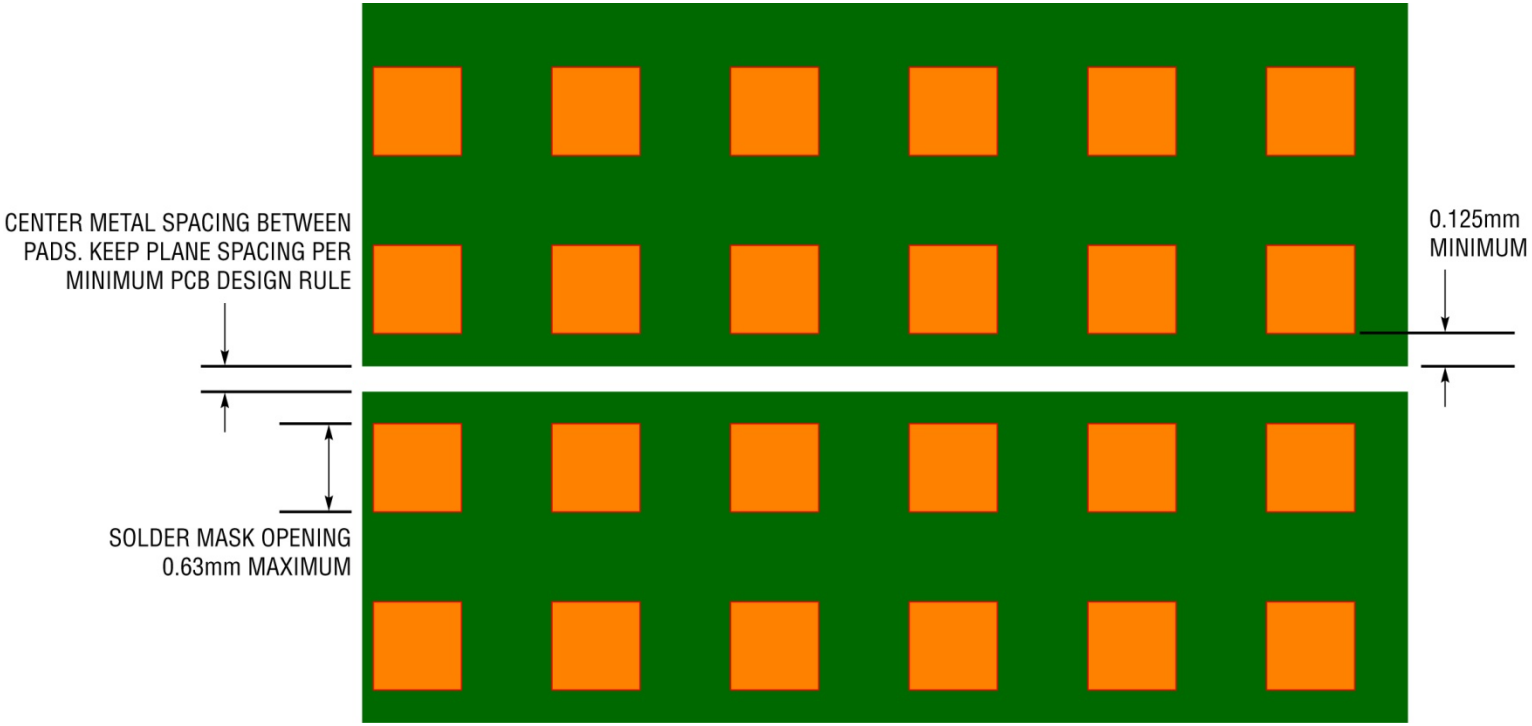
Thermal Relief: NSMD



Thermal relief is used by some customers on the power/ground planes. Customers need to ensure that the resulting thermal relief topology does not cause solder balling within the vias.

PCB Plane Separation

Maximum solder mask opening for plane separation needs to be controlled; Stencil opening in this area can be reduced to 0.6 to ensure no bridging;
Critical area – under Inductor and
plane separation



Moisture Sensitivity, Pack, Ship & Bake

- μModule products meet MSL level 3 of the JEDEC specification J-STD-020D.1 March 2008
- LTC ships all μModule devices in trays (or samples in sealed tubes) with desiccant and moisture level indicator
- Check the packing integrity (may need to check the source of shipment for repack procedures) if
  - Parts received in partial trays (other than from LTC)
  - Tape & Reel (**No Tape & reel to be used**)
  - Tubes (other than from LTC)
- If any of the above packing methods are encountered, moisture indicator shows pink color, or punctured seal of the bag is observed, bake the packages per the following conditions:
  - 125°C for 48 hours or 150°C for 24 hours
- Follow J-STD-033 “Handling, Packing, Shipping, and Use of Moisture/Reflow Sensitive Surface Mount Devices”

Solderability Test

- LTC μModule 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-B102D)

Stencil Design Recommendation

- Stainless steel laser cut stencils recommended
- Recommended stencil thickness 0.125 mm (5 mils) to 0.15 mm (6 mils)
  - Recommend slightly smaller stencil aperture than the pad opening (especially for SMD pads)
    - Stencil Opening 0.605 to 0.62 mm
    - To prevent paste from contacting solder mask
  - Solder volume ~0.05 mm^3 / Pad
  - 4 mil thick stencil not recommended (due to low stand off)
  - Corner radius of 0.06 mm on the aperture recommended
- Stencil area ratio (W/ 4*t) > 0.66 (not an issue for this aperture size)

  where W = Pad width, t = Stencil thickness

Screen Print

- Solder Paste
  - Low voiding paste
  - Type III or IV
  - Paste types used at Linear 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
- Key Process Parameters
  - Paste Rheology, Blade pressure, Paste floor life
- Check print definition, cleaning frequency
  - Stencil clogging can show as partial solder joints, not well defined joints

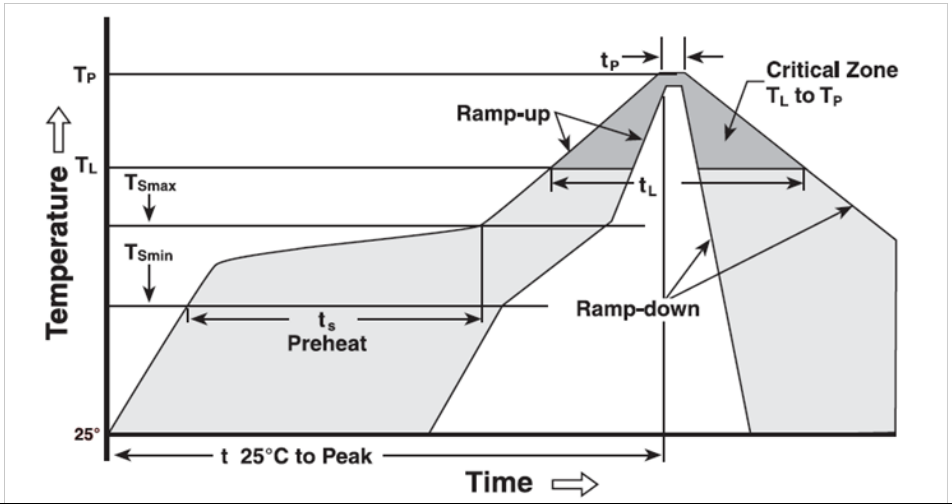
Placement

- 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
  - This ensures good solderability and less voiding

Reflow Profiles

- Both Air and N2 systems are OK (depends on solder paste)
- Recommend using a 9 zone or greater oven
- Profile with all components (fully populated board) and thermocouples under the μModule devices.
- Check that the solder paste vendor recommended profile conforms to LTC recommendations
  - If the LTC recommended profile cannot be met, adhere to the paste vendor profile except peak reflow temperature; Peak reflow temperature must be < 245°C
- If increased peak package body temperature is required (245°C < T_p < 260°C), decrease the floor life to 8 hours after bag opening or bake

Reflow Profile Guideline



Profile Feature		Lead-Free Solder	(Sn-Pb Eutectic Solder)
Pre-heat	Temperature Min (T_{Smin})	150°C	100°C
	Temperature Max (T_{Smax})	200°C	150°C
	Time (t_{Smin} to t_{Smax})	60-120 seconds	60-120 seconds
Reflow	Liquidus Temperature (T_L)	217°C	183°C
	Time (t_L)	30-90 seconds	30-90 seconds
Peak Package Body Temperature (T_p)		245°C	225°C
Time within 5°C of peak temp(T_p)		30 seconds	20 seconds
Average Ramp up Rate (T_{Smax} to T_p)		3°C/second max	3°C/second max
Ramp Down Rate		6°C/second max	6°C/second max
Time 25°C of peak temp(T_p)		8 minutes max.	6 minutes max
Do not exceed		245°C	220°C

Cleaning

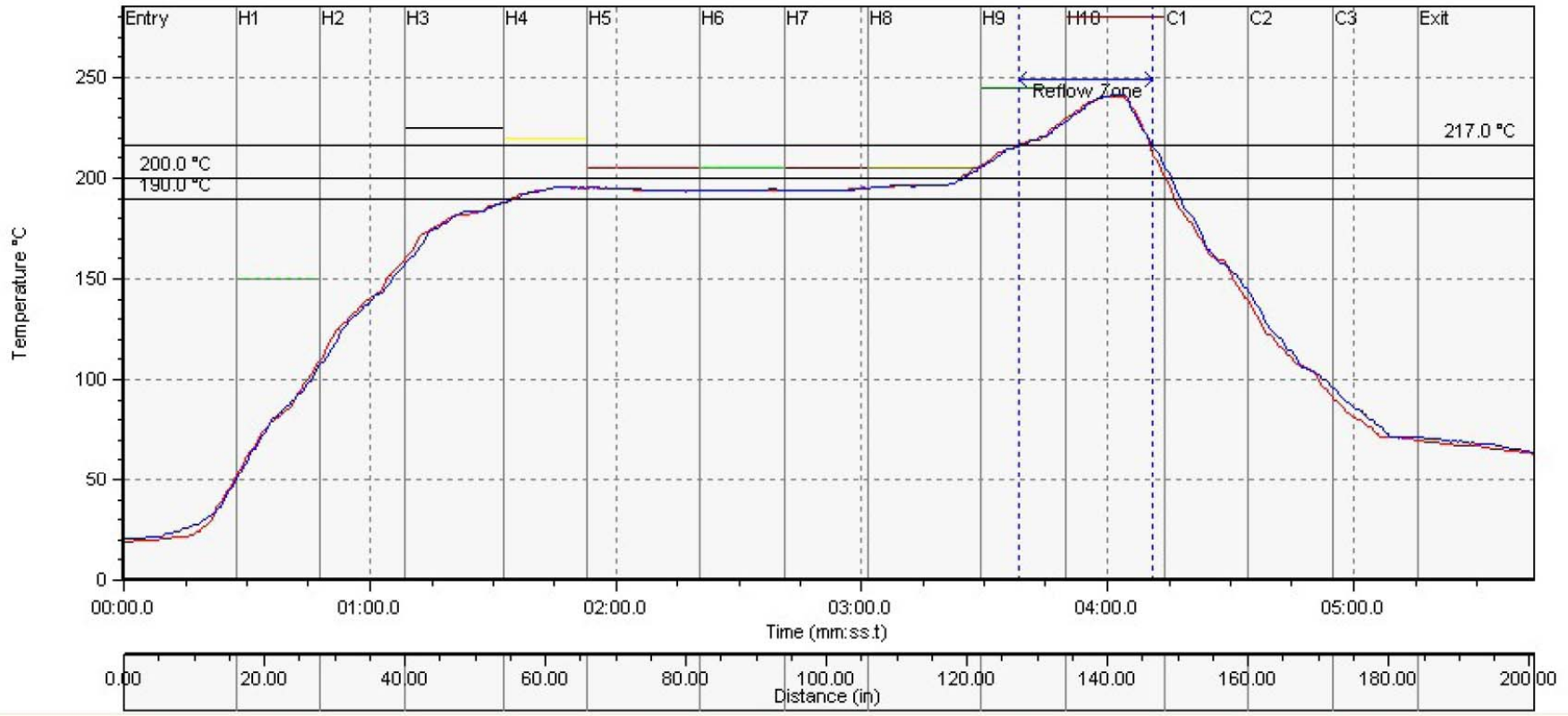
- No clean paste recommended
- Water soluble paste has been used successfully with the μModule packages
  - Use a saponifier or ultrasonic cleaning with water
  - Use DI water spray system to further clean under the LGA

Solder Joint Voiding

- No IPC standard for LGA pad voiding criteria
- LTC has tested devices with solder joint voids up to 30% and no reliability issues seen
- LTC recommends using a 25% maximum void criteria for solder joints
- Correct Z height adjustment ensuring contact of paste and LGA pad and soak profiles minimize solder joint voiding
- If the recommended LTC or paste vendor profile results in >25% voiding, then use a soak profile during reflow (Profile with Thermocouple underneath the LGA)
  - For Pb free paste, ramp to 180°C to 200°C and stay at 200°C to 210°C for 90 to 110 sec
  - For Sn/Pb paste ramp to 150°C and stay for 90 to 110 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

LTM 4601 Soak Profile Example (Long Soak)

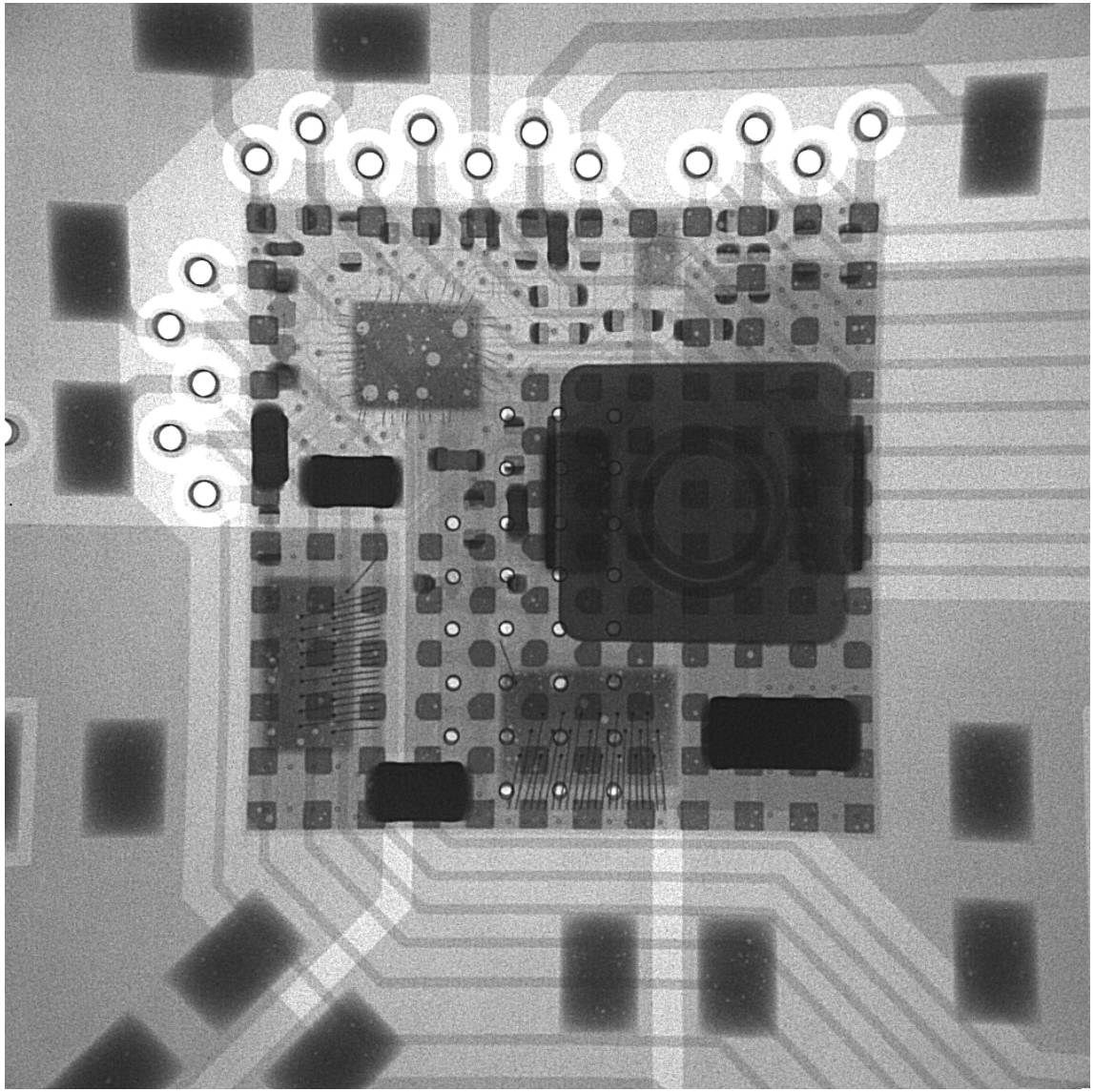
Solder Paste: AIM



Reflow Results

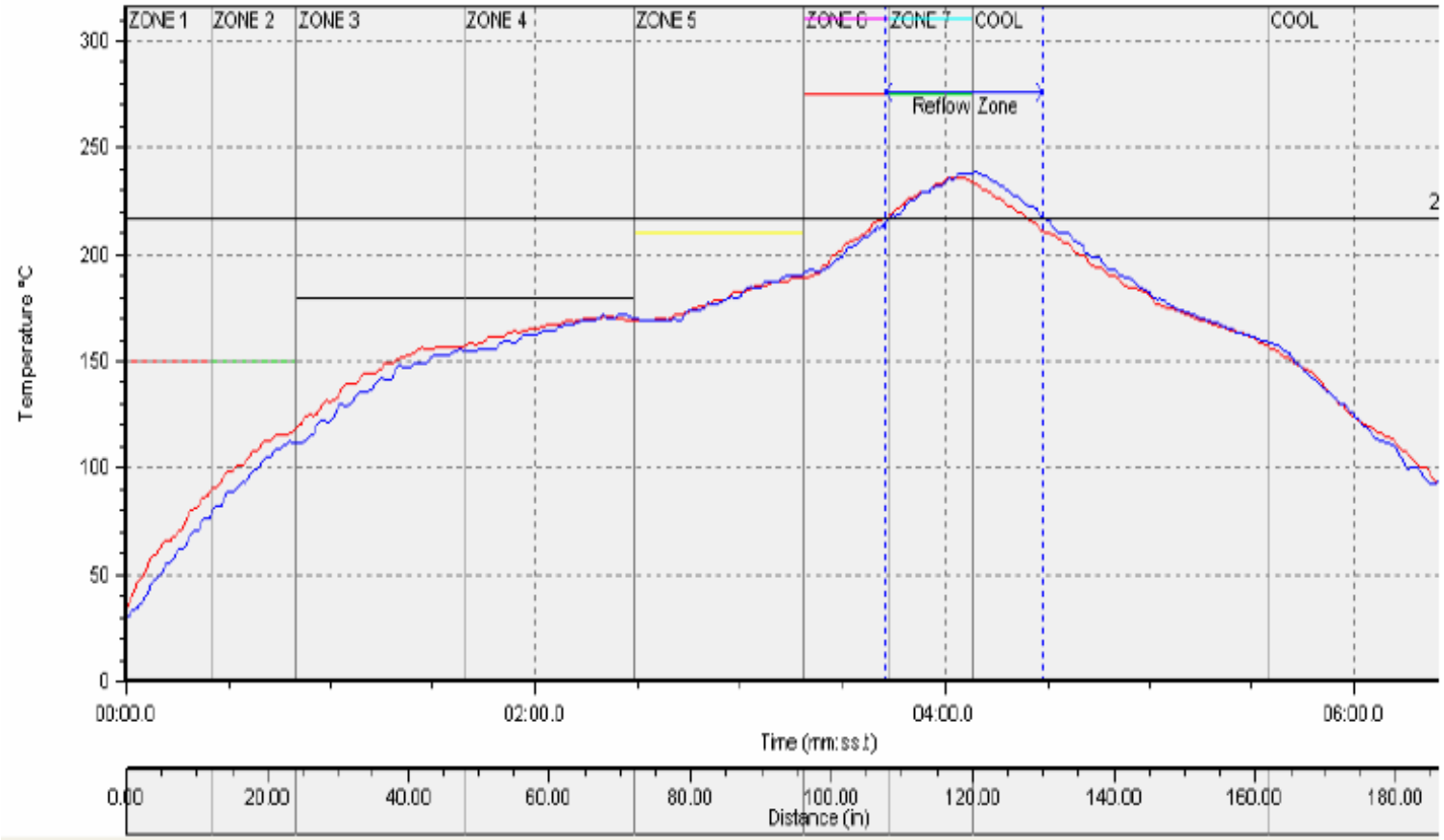
Probe	Positive Slope (°C/sec)	Positive Slope Time (mm:ss.t)	Rise Time (189.0 - 200.0°C) (mm:ss.t)	Time Above Liquidus (217.0°C) (mm:ss.t)	Peak Temperature (°C)	Delta T (°C)	Negative Slope (°C/sec)
#1 (°C)	3.57	00:26.0	01:51.0	00:32.0	241.5	0.5	-4.52
#3 (°C)	3.51	00:28.0	01:50.5	00:32.5	242.0		-3.98

LTM 4601 Void monitor
(Long Soak)



LGA joints show minimal voids

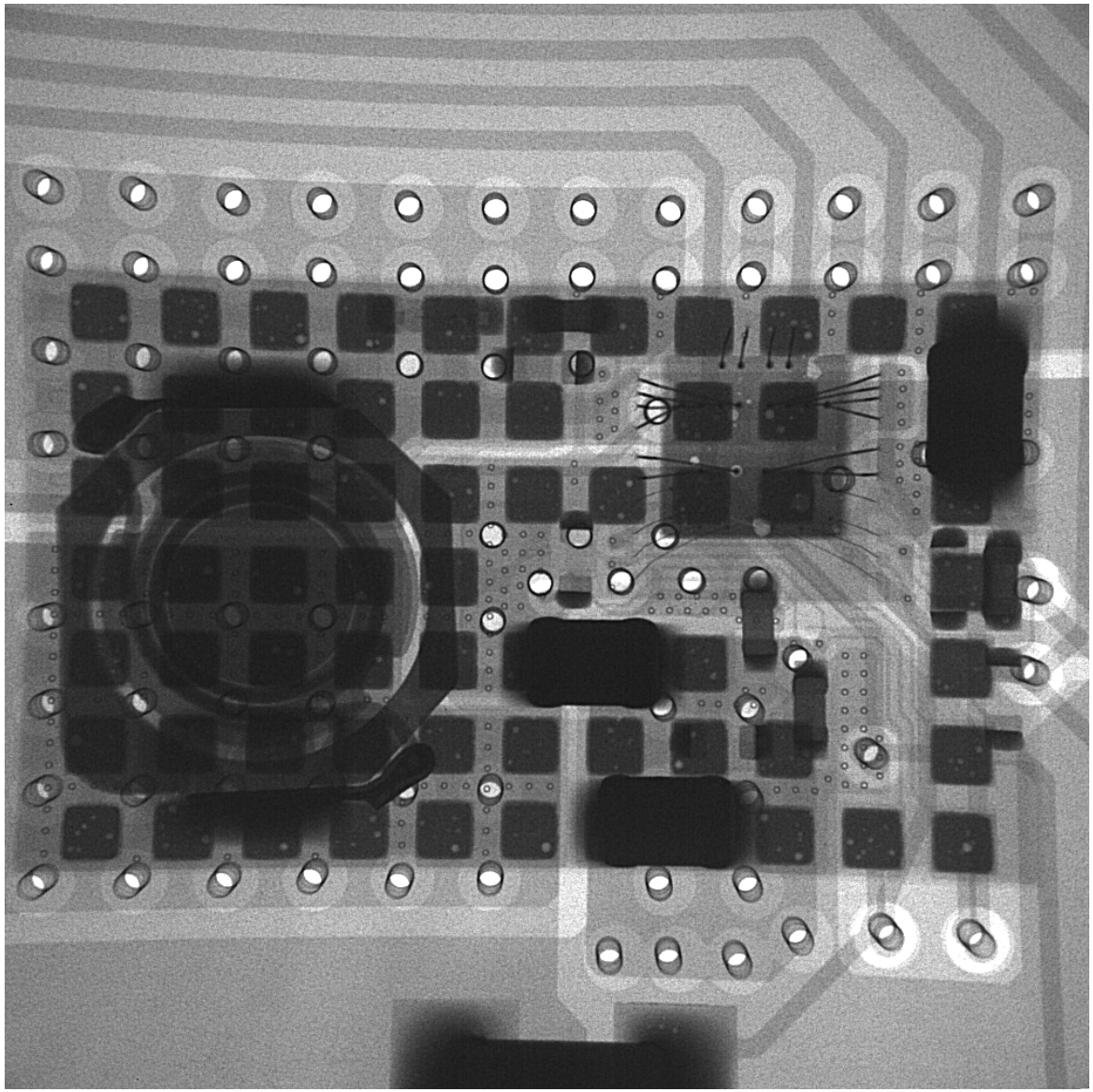
LTM 4604 Profile Example
(Short Soak)



Reflow Results

Probe	Positive Slope (°C/sec)	Positive Slope Time (mm:ss.t)	Rise Time (150.0 - 170.0°C) (mm:ss.t)	Time Above Liquidus (217.0°C) (mm:ss.t)	Peak Temperature (°C)	Delta T (°C)	Negative Slope (°C/sec)
#1 (°C)	2.88	00:05.0	00:55.0	00:42.0	236.0	2.5	-1.82
#3 (°C)	2.44	00:07.5	00:49.5	00:46.0	238.5		-1.64

LTM 4604 Void Monitor
(Short Soak)



LGA joints show minimal voids

Rework
Component (μModule device) Removal

- Determine the failure mode from the board and at what operation the defect occurred
  - After assembly
    - 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

- Remove the component from the board within 168 hrs of the moisture barrier bag opening prior to assembly or after baking the PC board assembly for 24 hours at 125°C
  - If this step is not followed, there is a possibility of delamination of the mold compound from the substrate (solder mask)
    - If the part is heated above 245 deg C, the internal solder in the module will melt and spread through the delaminated areas
    - If the part was baked properly, the solder does not spread and is held within the pad

Rework
Component (μModule device) Removal

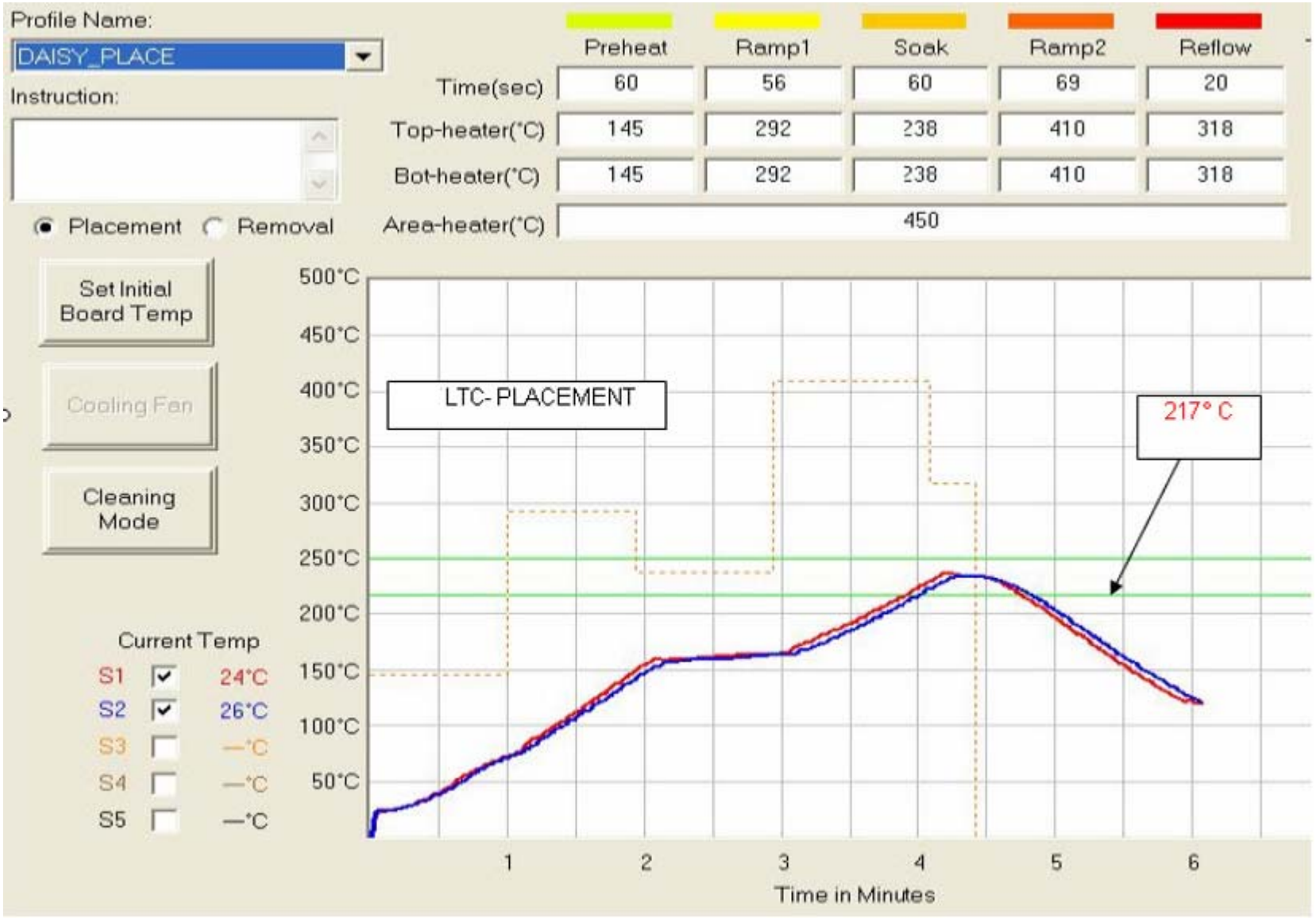
- Use a BGA rework station capable of profiling top and bottom of Module
  - Handheld heat guns or IR-only rework stations should not be used
  - Use appropriate heat shielding of sensitive components in proximity to the μmodules
  - The profile is done with a thermocouple on top of the part and another at the bottom of the part (close to the solder joints)
    - Maximum temperature for top of package = 245°C
    - 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
  - 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 Attachment

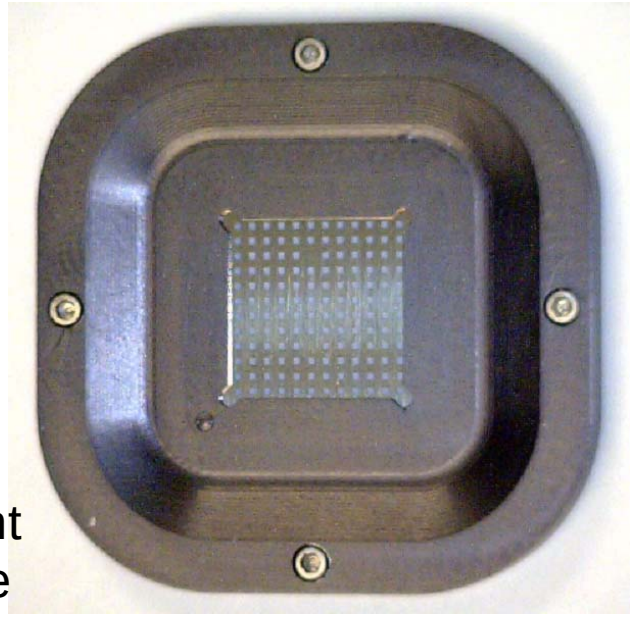
- Cleaning and prep of PCB lands
  - Solder wick or solder iron can used
  - Ensure PCB pads are not damaged during the cleaning process (excess heat or excess mechanical scraping can damage the pads)
- Screening of solder paste
  - Paste printing can be done on the component instead of PCB (no clearance issues)
  - Use a micro-stencil; Position the part onto the stencil frame; Hold the part in position
  - Print Type 3 or 4 no clean paste onto the new component on the pad side of the component
  - Ensure no clogging of the stencil; Clean stencil after each print
- Placement and reflow of new component
  - Removed component should not be reused
  - Use split vision system (align the printed pads on the component with the PCB land pattern)
  - Reflow profile to ensure adequate soaking time as well as time above liquidus
- Inspection of solder joints using X-ray

Rework Profile (Pb free)

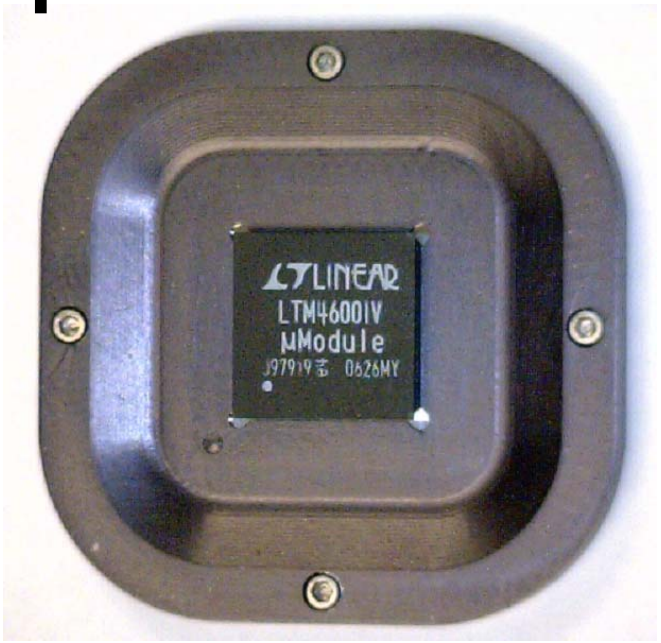
– Removal and Attachment



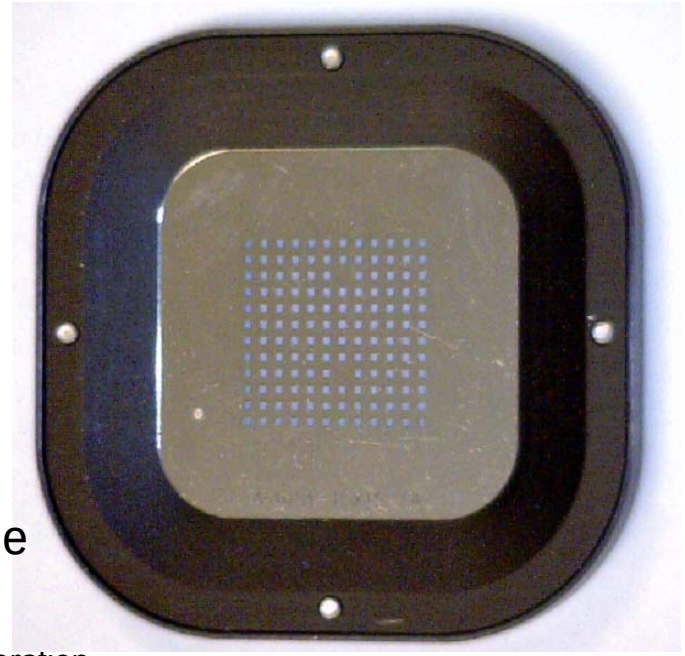
Micro Stencil, Component Placement



Component
(Nest) side

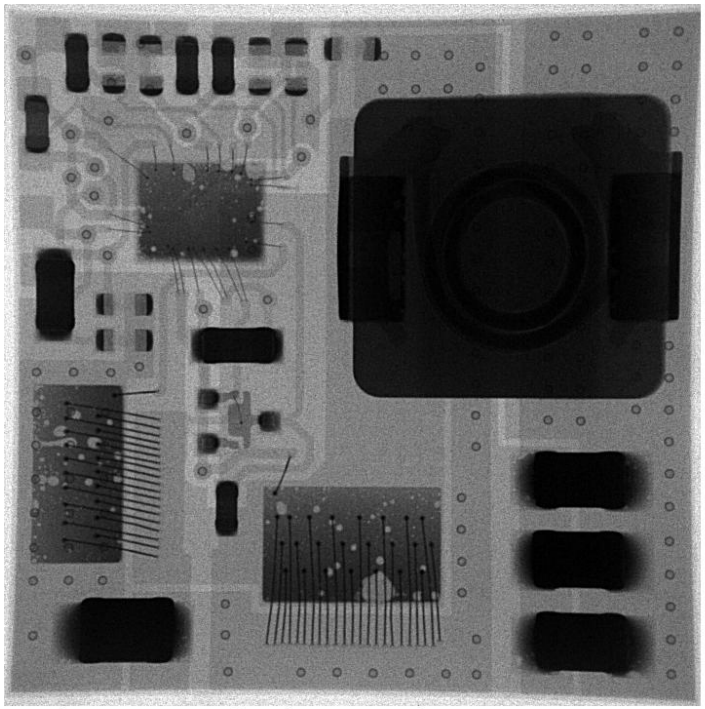


Place component in the nest
Hold component with one hand
Flip the fixture
Apply paste on print side



Print side

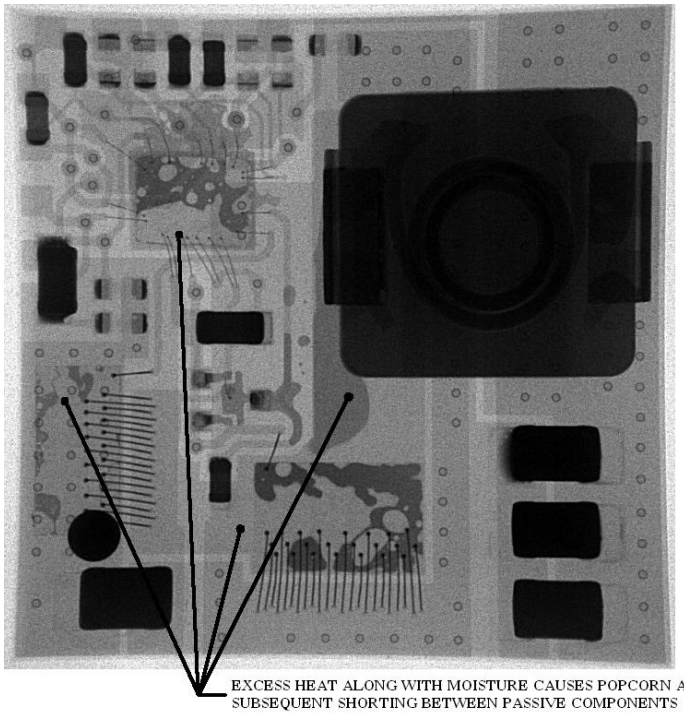
X-ray inside the μModule Device



Micro Module X-Ray following a correct reflow and rework procedure

NO SHORTS

- Part met Level 3 floor life
- Reflow peak temperature within the spec
- Part was baked and removed



SOLDER SHORTS INSIDE MODULE

- Part did not meet Level 3 floor life - delamination
- Reflow peak temperature out of spec – solder melted & spread due to #1
- Part was not baked prior to removal

FAQs

1. What is the stencil opening, thickness?
   - See Stencil design page
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; Finefocus X-ray can also be used (need to have good training 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?
   - Yes, provided the total exposure time (out of bag to 2nd reflow) is less than 168 hrs
6. Parts are shorting (Vin to Gnd or Vout to Gnd)
   - Check for solder joint shorts (check the schematic with the pin configuration)
   - Check inside the module to see if any solder spreading has occurred as shown in the prior slide

FAQs

6. How to prevent shorting inside the module?
   - Check the floor life of the parts (From out of sealed bag to reflow); If over 168 hours, parts need to be baked for 48 hrs at 125°C
   - Was the bag not sealed or moisture indicator showing pink color? – Bake parts for 48 hrs at 125°C or for 24 hrs at 150°C
   - Was the reflow peak temperature greater than the peak temp for the module size (refer to the Table on slide titled " μModule MSL Rating")? 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 to be baked for 24 hrs at 125 deg C
     - If heat gun was used, temperature may be excessive
     - Profile the rework station and remove component
7. How and where to get mini-stencil?
   – Photo stencil, Colorado Springs, CO
     - Phone 719-535-8544
     - Contact : Heather Marshall hmarshall@photostencil.com.
   – The alignment block (2 pieces that hold the above stencils) comes from
     - 2Spec Engineering (alignment blocks), San Jose,CA
     - Phone 408-227-3200
     - Contact : Ty Mingione Ty@2Spec.com
8. Can the removed module be reused?
   - We do not recommend the reuse of the module after removal. Use a fresh part to replace