ASE GROUP

Surface Mount Requirements for QFN Package

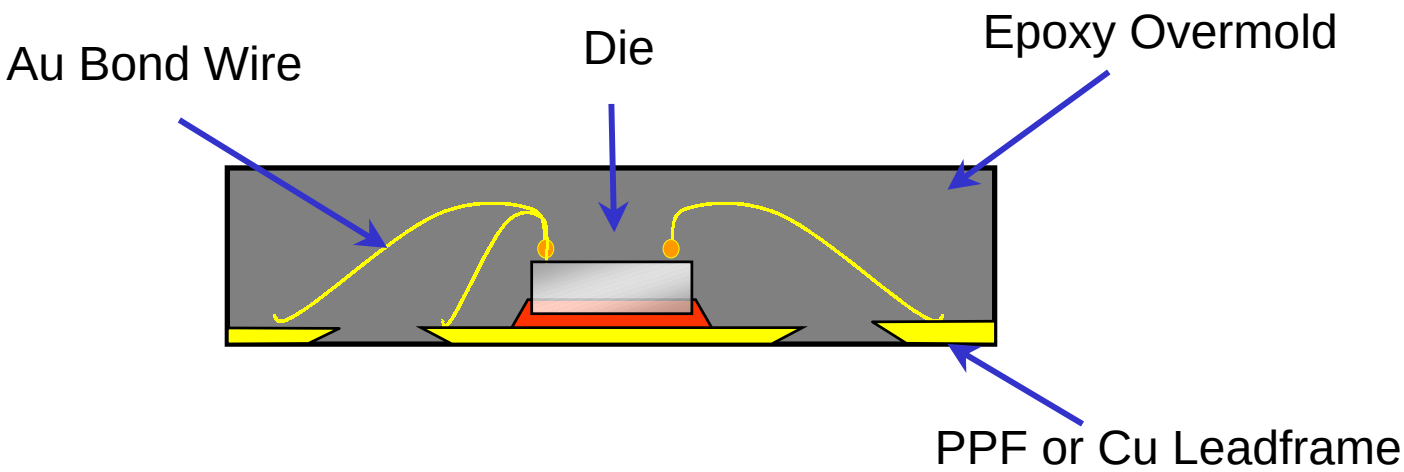
ASE GROUP

Contents

- Introduction
- Motherboard recommendations
  - Terminal pad types
  - Terminal pad size
  - Thermal pad land design / Thermal via design
- SMT assembly considerations
  - Stencil
  - Solder paste / Printing parameters
  - QFN placement
  - Reflow oven / profile
  - Inspections

Introduction (1)

The Quad Flat-No lead package (QFN) is a near CSP with a copper leadframe. Wire bond is typically used to connect the IC to the leadframe. Electrical, thermal and mechanical connections to the motherboard are made through the exposed lands and die pad on the bottom of the package.



Note

- This document is just a guide line to help the user in developing the proper motherboard design and surface mount process.
- Actual experience as well as development effort is needed to optimize the process as per user's surface mount practices and requirements.

Introduction (2)

ASE GROUP'S QFN product offerings cover a range of lead pitches, including 0.8 mm, 0.65 mm, and 0.5 mm. Further QFN products will likely have a lead pitch as small as 0.4 mm.

Total thickness	0.8m / 1.0mm
Mold cap	0.54mm / 0.7mm
Leadframe thickness	0.2mm / 0.2mm

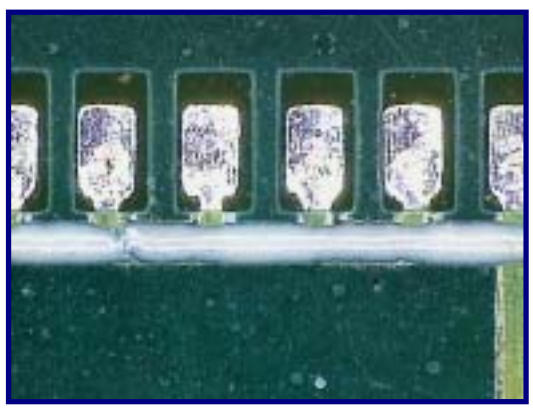
Package size (mm^2) / Lead Pitch (mm)	3x3	4x4	5x5	6x6	7x7	8x8	9x9	10x10	12x12
0.4						64		84	
0.5	12/16	20/24	28/32	36/40	44/48	52/56	64	68	84
0.65	8	16	20	28	32	40	44		
0.8	4	12	16	20	28	32			

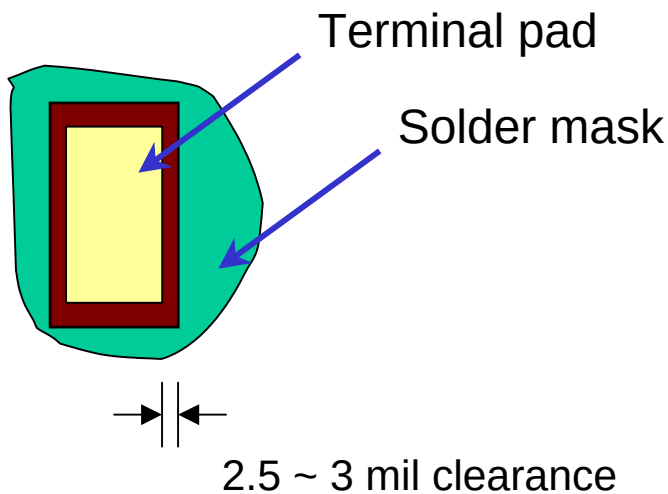
Note : **Blue :** ASECL Standard, **Black :** Customer design **Red :** Under development

Motherboard Recommendations

Motherboard Recommendations (1)

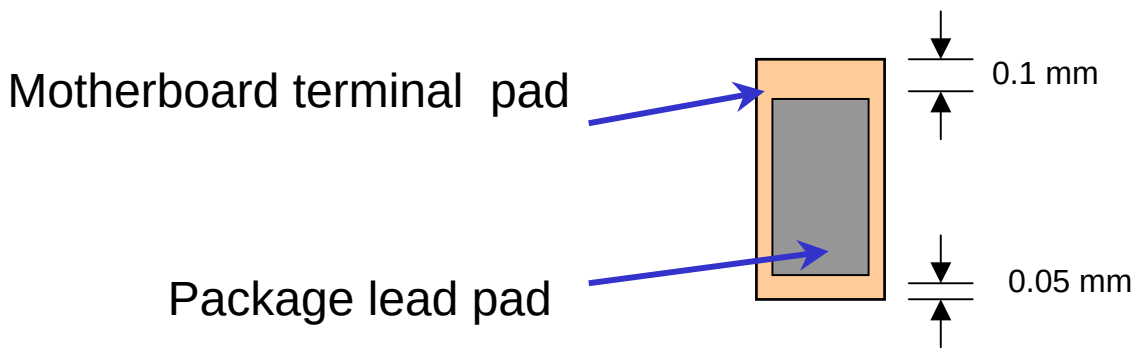
- **Terminal Pad types**
  - There are two basic types of land pad for the BGA package: Non-Solder Mask Defined (NSMD) and Solder Mask Defined (SMD). Both types are acceptable for use with the QFN package.
  - NSMD pads have a solder mask opening which is larger than the metal pad.
  - NSMD is recommended because the copper etch process has tighter control than the solder masking process. The reliability of solder joint was also improved.
  - The clearance around the copper pad and solder mask is 2.5 ~ 3 mil.





Motherboard Recommendations (2)

- **Terminal Pad Size**
  - An ideal solder joint is one that has equal cross-sectional areas at the terminal pad and the PCB interfaces.
  - Smaller pad better for thermal fatigue reliability
  - Larger pad better for bend or drop performance
  - The pad pattern dimension is most likely to be larger than the nominal lead dimension. We recommended using the nominal dimensions and extend the pad at least 0.1 mm on the outside and 0.05 mm on the inside.



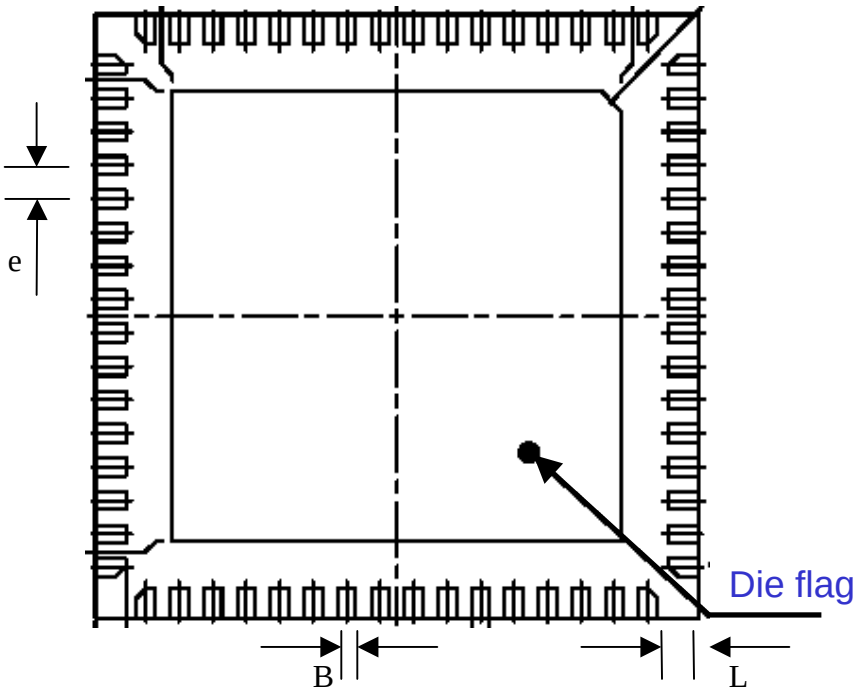
Motherboard Recommendations (3)

- **Thermal Pad Land Design**
  - The size of the thermal land should at least match the exposed die flag size. But it is necessary to avoid solder bridgeing between thermal pad and the perimeter pads. We recommend the clearence between thermal pad and perimeter pads is 0.15 mm.
- **Thermal Via Design**
  - In order to take full advantage of QFN thermal performance, thermal vias are needed to provide a thermal path from top to inner/ bottom layers of the motherboard to remove the heat.
  - Via size(in diameter): 0.3 ~ 0.33mm
  - Via pitch : 1.0 ~ 1.2 mm
  - # of thermal vias : depend on the application

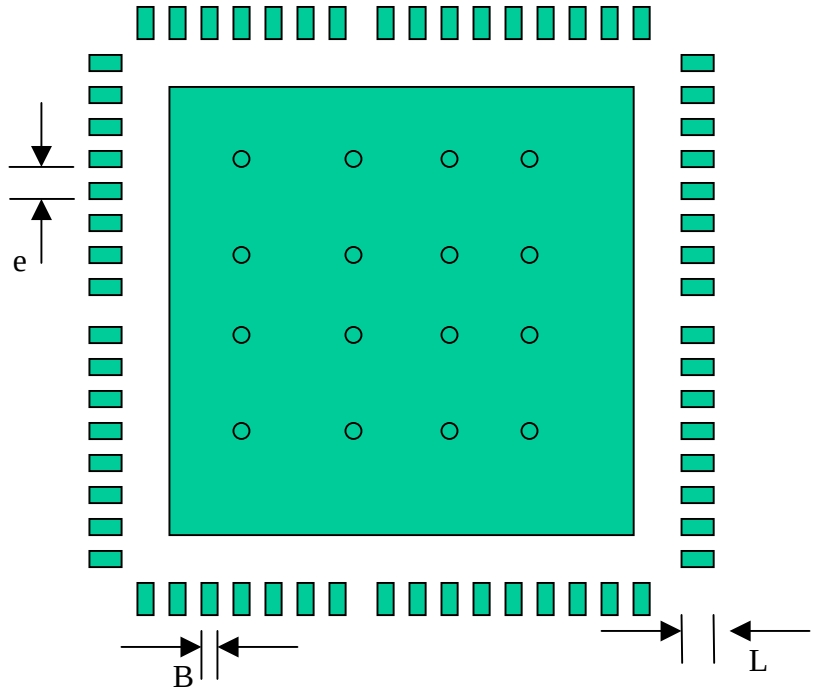
Motherboard Recommendations (4)

Example of QFN 9x9 64L motherboard design

QFN Package Outline



PCB Outline



# of leads: 64

Package size: 9 x 9 mm

e - QFN Terminal Pitch	0.5 mm
B - QFN Terminal Width	0.25 mm
L - OFN Terminal Length	0.45 mm

Die flag : 6.67 x 6.67 mm

PCB pad dimension

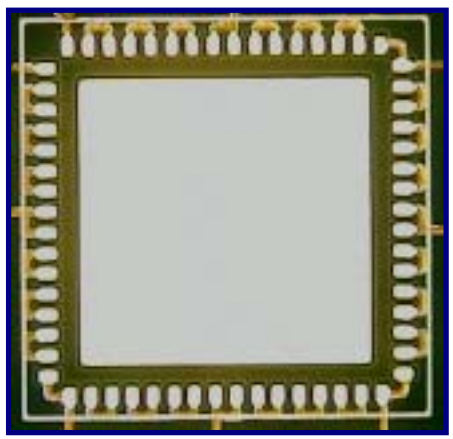
e - QFN Terminal Pitch	0.5 mm
B - QFN Terminal Width	0.27 mm
L - OFN Terminal Length	0.60 mm

Thermal pad : 7 x 7 mm

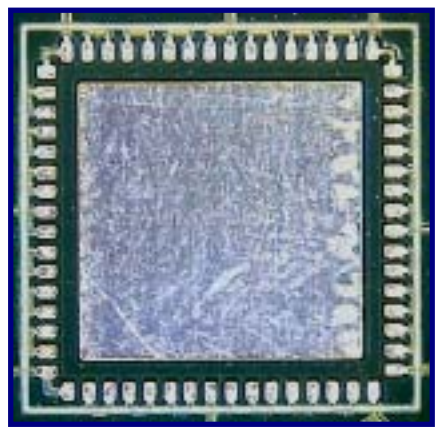
Thermal via : 4x4 array, 1.2 mm pitch
0.3 mm dia.

Motherboard Recommendations (5)

- **In order to achieve high assembly yields,use of a surface finish that is planar and has good solderability performance is important.**
- **OSP or NiAu surface finish is recommended.**
- **Gold thickness : 0.05 μm ~ 0.127 μm**



Ni-Au

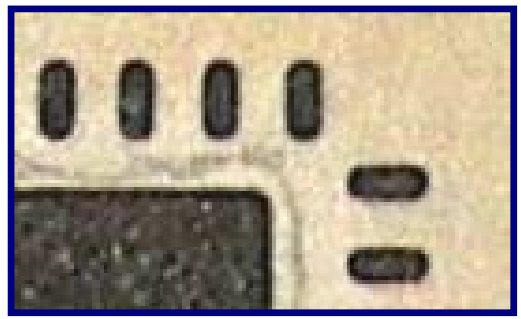


HASL

SMT Assembly Considerations

SMT Assembly Considerations (1)

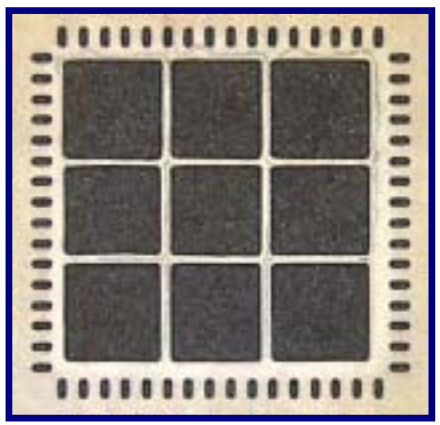
- **Stencil**
  - Stencil Type & Thickness
    - Laser cutting
    - stainless steel
    - Thickness
      - 0.5 mm Pitch ： thickness < 0.15 mm



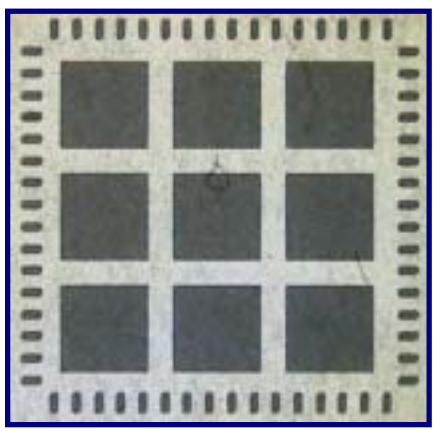
  - Aperture size and Shape for terminal pad
    - Aspect ratio (width/thickness) > 1.5
    - Aperture shape
      - The stencil aperture is typically designed to match the pad size on the PCB.
      - Oval-shaped opening should be used to get the optimum paste release.
      - Rounded corners to minimize clogging
      - Positive taper walls (5^{o} tapering) with bottom opening larger than the top

SMT Assembly Considerations (2)

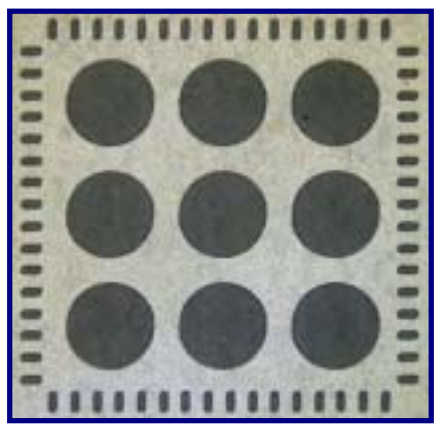
- **Stencil**
  - Aperture design for thermal pad
    - The small multiple openings should be used in steady of one big opening.
    - 60 ~ 85% solder paste coverage
    - Rounded corners to minimize clogging
    - Positive taper with bottom opening larger than the top



Don't recommend
Coverage 91%



Recommend
Coverage 77%

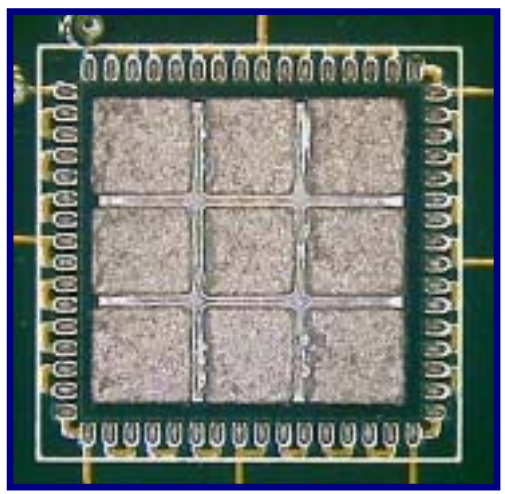


Recommend
Coverage 65%

SMT Assembly Considerations (3)

- **Solder Paste**
  - Alloys: 63Sn/37Pb or 62Sn/36Pb/2Ag
    Sn/Ag/Cu family for lead-free application
  - No Clean flux
  - Particle size: Type III (25 ~ 45 μm) or Type IV (25 ~ 38 μm)*Type IV is preferred to improved printing performance*.
- **Printing Parameters**
  - Metal squeegee with 45^{o} ~ 60^{o} printing angle
  - Speed : start with 20 mm/sec,increasing with experience
  - Printing pressure : 10 N/mm^{2}
  - Snap-off distance : 0 mm

For the pitch ≤ 0.5 mm, 3D post-printing inspection is highly recommended.

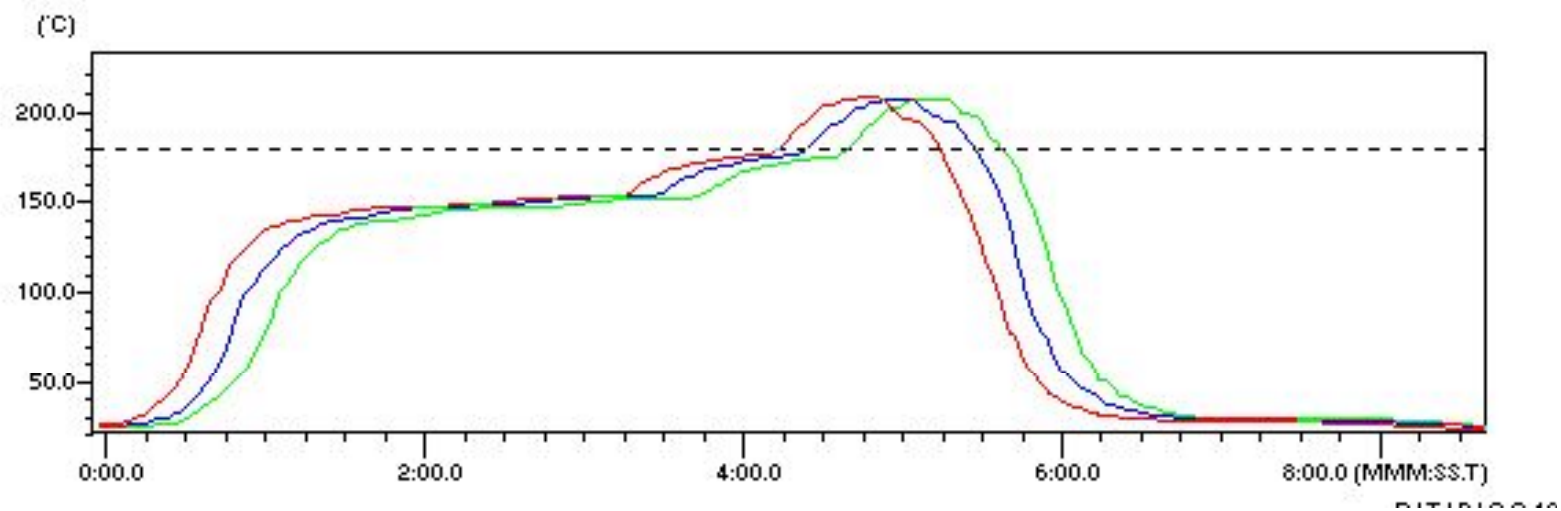


SMT Assembly Considerations (4)

- **QFN Placement**
  - Using standard pick and place machine with ± 0.05 mm accuracy
  - Mounting with slower speed and higher force
  - Place the package 1 ~ 2 mils into the paste
  - QFNs has excellent self-alignment during solder reflow if a minimum of 75% of the lead diameter intersect with the pad.
- **Reflow Oven**
  - Forced convection reflow with nitrogen
  - Temperature uniformity ≤ ± 5°C
- **Profile**
  - Follow datasheet of solder paste
  - Other concerns: other parts,board density,board thickness….
  - For the PPF leadframe, the palladium will be dissolved into the solder and the joint is formed with the underlying nickel layer. Higher reflow temperature is required to foam a good solder joint.

SMT Assembly Considerations (4)

Recommended Reflow Profile for 63/37 Solder Paste or Cu lead frame



This profile is designed for use with Sn63 or Sn62 and can serve as a general guideline in establishing a reflow profile.

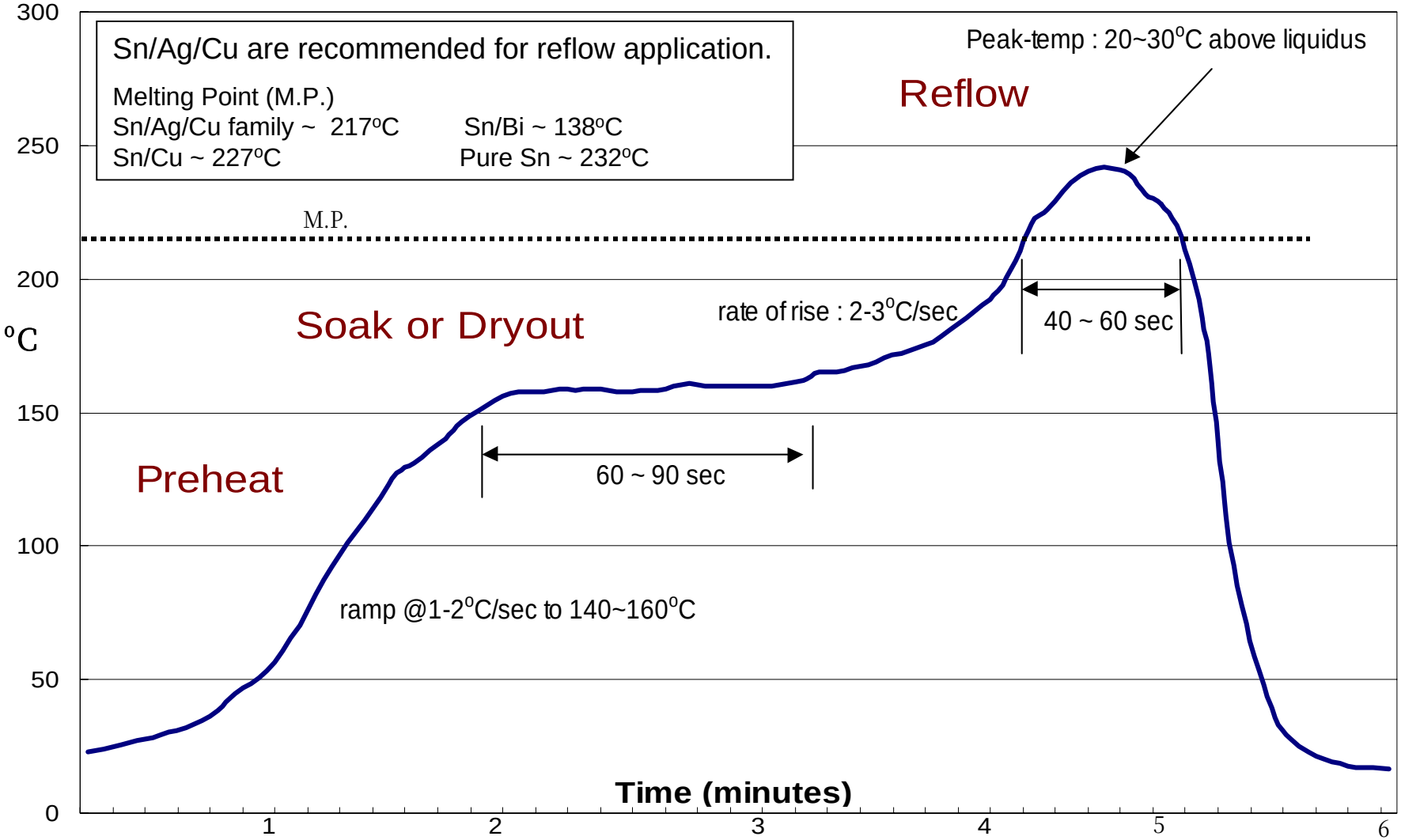
Reflow Profile:

- Heating-up @1~3°C/sec to 140°C
- Preheat @ 140-150°C for 120 ~ 160 sec
- Ramp @ 2~3 °C/sec to peak temperature (220 ~ 225 °C), Temperature over 183°C for 45~ 75 sec
- Cooling down to room temperature @4~2°C/sec to avoid undesired intermatalic compound layer.

ASE GROUP

SMT Assembly Considerations (5)

Recommended Reflow Profile for Lead-free Solder Paste or PPF lead frame



SMT Assembly Considerations (6)

- **Inspections**
  - Using transmission X-ray to sample monitoring of the solder joint.
  - Side-view inspector is also recommended.