SMT Board Assembly Process Recommendations

Intel® Manufacturing Enabling Guide

March 2016

Disclaimers

Table of Contents

Figures

Tables

Introduction

This chapter addresses the surface mount technology (SMT) board assembly process for reflow soldering SMT components to boards, as well as rework soldering for removing and replacing individual components on already-assembled boards.

The information in this document is for reference only. Manufacturing processes are unique, and may require unique solutions to ensure acceptable levels of quality, reliability, and manufacturing yield. Due to differences in equipment and materials, customer-specific process parameter development and validation is required.

Solder Paste Printing

Although there are a number of lead free (Pb-free) alloys, the most commonly used compositions contain tin, silver, and copper, commonly expressed as SAC, for SnAgCu. Within SAC solders, by far the most common usage is Sn/3Ag/0.5Cu, a near-eutectic which melts between 217°C and 220°C.

Stencil aperture sizes can be 1:1 with pad size, but certain parts may require reduced or increased stencil apertures to reduce solder ball defects. For example, in the BGA world, the thinning of substrates and overall package thickness has led to increased package warpage during the SMT process, and hence the need to adjust stencil openings. In some cases the package may tend to bend away from the PCB during SMT reflow leading to an open, and by increasing paste volume deposited on the PCB pads via the stencil aperture size, you can compensate for this gap and ensure a good joint between package and PCB during reflow. The reverse situation can also occur, where a package warps and compresses against the PCB, and a smaller stencil aperture is required to minimize bridging defects.

Using a metal squeegee reduces scavenging and provides more consistent printed paste volume.

Process parameters (such as squeegee speed, pressure, and separation speed) need to be optimized for the specific solder paste used.

Component Placement

Pick and place machines are used to place components on boards. Front-side lighting is often used for ball recognition on BGA's.

Reflow Soldering

In reflow soldering, the solder paste must be heated sufficiently above its melting point and become completely molten, in order to melt the balls of BGA components, causing them to collapse and form reliable joints. In the case of components with leads, the solder paste must wet the plating on component leads to form the desired heel and toe fillets.

Solder joint formation depends on temperature and time which are reflected in the reflow profile. In BGA's the balls on the component are the main contributor to the solder volume of the joint and the paste volume applied is critical to the formation of the joint.

There is no one best reflow profile for all board assemblies. Ideally, a reflow profile must be characterized for each board assembly using thermocouples at multiple locations on and around the device. The solder paste type, component and board thermal sensitivity must be considered in reflow profile development.

Pb-free Reflow Soldering

Pb-free reflow requires relatively high temperatures, due to the high melting range of typical Pb-free solders. Typical Pb-free solder such as SAC305 (Sn/3Ag/0.5Cu) have s an initial melting point of 217°C and a final melting point of 220°C.

Pb-free reflow soldering requires a narrow temperature range, in order to produce reliable joints, without damaging components.

Because of additional oxidation that occurs at higher temperatures, an inert reflow atmosphere (nitrogen) may be beneficial for Pb-free reflow soldering.

Of course, high temperatures drive the need for all Pb-free components to be rated to higher temperatures. Finally, these high temperatures, along with thinner packages, can cause greater warpage in PCB's, and in some cases, may require alternate PCB materials, support pallets, careful attention to specific paste formulations, and carrier fixtures during reflow.

Reflow Profile Development Considerations

Each customer should develop their own reflow profile and oven settings, appropriate to their materials, equipment, and products. As a starting point, this chapter contains considerations and recommendations for reflow solder parameters. Because some reflow parameters differ with solder paste formulation (even if they have the same metal composition), the profile envelope recommended by the solder paste manufacturer should be considered.

Reflow Profile Board Preparation

Reflow profile measurement is a vital part of setting up reflow the solder conditions. The measurements are typically carried out using thermocouples attached to a high temperature resistant recording device which travels through the reflow oven furnace with the PCB under test. Special care must be taken to ensure proper placement of thermocouples to accurately measure temperature at the desired locations.

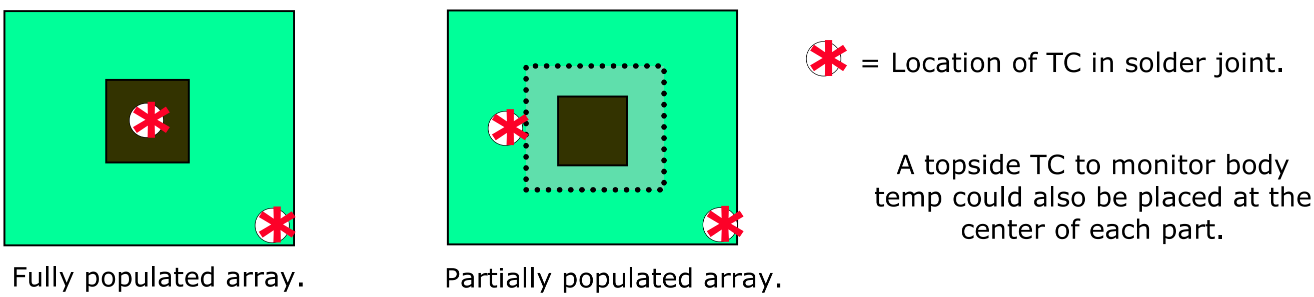
Unless stated otherwise, all temperatures in this chapter are measured at solder joints, rather than at components bodies, PCB surface, or air around components. This provides the best repeatability and accuracy.

Thermocouples (TC's) for solder joints should be placed in joints expected to be the hottest and coolest, so that the range of peak temperatures for all components on the board can be confirmed to be within specifications. The hottest joints on a board are typically on small passive components, so one of these should be monitored for peak temperature on the profile board. The coolest joints on a board are typically large BGA's and sockets. A TC should be used in a joint at one corner of the component, and in a joint at the center of the part, or as near to the center of the part as possible. Sockets with actuating mechanisms may require an additional TC at a joint near the mechanism, if its mass could make that area harder to heat.

In addition to solder joints, component body temperature, measured at top center or as close as possible, may also need to be monitored, to avoid exceeding the body temperature spec of the part.

Here are examples of TC locations for reflow profiling on BGA's or sockets, for both fully populated arrays and partially populated arrays (no balls in the center area of the part).

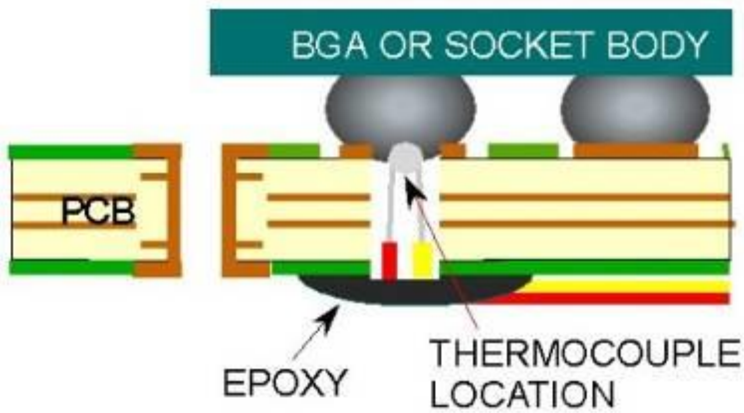
Figure 0-1 Reflow Profiling TC Location Example (BGA or Socket)



Here is a method for placing TC's to measure SMT joint temperatures:

- Before the component is soldered to the board, drill a small hole through the pad of the joint to be measured
- Insert the thermocouple from the bottom of the board
- Hold the thermocouple tip flush with the top surface of the board
- Apply epoxy from the bottom side of the board to keep the thermocouple in this position, where it will be in contact with the joint, but not interfere with paste printing
- Print solder paste, place components, and reflow the board

Figure 0-2 SMT Joint Temperature TC Placement Example



2.1.1.1 Minimum Solder Joint Peak Temperature

With SAC305 or SAC405 Pb-free solder pastes, the coolest joints on a board should generally reach at least 228°C, preferably 230°C. For BGA's, this applies to ball alloys SAC305 or SAC405. For SAC305/405, 228°C represents at least 11°C superheating above the initial melting point (217°C). 230°C represents 13°C superheating. Temperatures lower than these can result in solder joints that are not fully formed, or in reduced solder joint reliability.

2.1.1.2 Maximum Solder Joint Peak Temperature

250°C is recommended as the maximum temperature for all solder joints on the board, except for components with temperature ratings lower than 250°C. If maximum solder joint temperatures exceed 250°C, PCB damage such as delamination and warpage may result when standard FR4 (Tg =130°C) material is used. Higher Tg material is not necessarily more resistant to this damage, and must be tested for compatibility.

Components are typically rated as per J-STD-020C (or later), based on their package thickness and volume. Although Intel BGA's are generally rated at 260°C, other components, especially large ones, may be rated at 250°C or 245°C. This means that 250°C may not be usable as the max joint temperature; a lower temperature may be required. Since larger parts normally reach lower maximum temperatures during reflow than smaller parts due to the physics of heat transfer, keeping them below their ratings may not be difficult, as long as 250°C is used as the maximum for the joints of smaller parts.

2.1.1.3 Time Above (Initial) Melting Point

The length of time that joints spend above the min peak temp is also an important factor for solder joint reliability. Intel recommends that this time be measured from the time a joint goes above the initial melting point of the alloy (217°C for SAC305/405), until it goes below it during cooling.

Time Above Liquidus, or TAL, is often used to describe this time. But technically speaking, 217°C is the solidus temperature of SAC305, the

point at which the solder becomes fully solid during cooling. 220°C is the **liquidus** temperature, the point at which it becomes **fully liquid during heating**. Between these two temperatures, the solder is partially molten and partially solid.

Unfortunately, common usage in the lead free industry has often incorrectly used the term 'liquidus' to refer to the initial melting point, rather than the final melting point. This is probably a carry-over from SnPb soldering, where 'liquidus' was used correctly, since liquidus and solidus are both the same temperature (183°C). In order to avoid confusion, this document will avoid the use of "Liquidus" and "Time Above Liquidus (TAL)". Reflow time will be stated as Time Above 217°C (TA217), rather than TAL.

Intel recommends that TA217 of 40-90 seconds be used for SAC305 or 405 solder paste and solder balls. With large or massive boards, an exception may be required, allowing up to 120 seconds above 217°C.

Rising and Falling Ramp Rate

To avoid component damage, manufacturers often recommend that rate of temperature increase during heating (Rising Ramp Rate) be kept below 3°C / sec. This applies throughout the heating process, up to peak reflow.

During cooling, Intel recommends that a minimum ramp rate be used instead of a maximum rate. Solder joints with cooling rates of 2°C / sec are characterized by finer microstructure features. Literature studies indicate that this is better for long term reliability. Faster cooling rates also inhibit growth of intermetallic compounds in the bulk solder. The Falling Ramp Rate applies from peak temperature down to 205°C.

Reflow Equipment

Producing and controlling the necessary temperature range, while maintaining production speeds, typically relates to the number of heating zones in the oven. Assemblers with reflow equipment with greater temperature control (i.e. greater numbers of zones) will be better positioned to meet Pb-free process envelope requirements, particularly for larger, more complex boards.

Reflow Atmosphere

Reflow soldering in an inert atmosphere, such as nitrogen, reduces the amount of solder and pad oxidation that occurs during soldering. This can improve the quality and appearance of SMT joints, but can have a large impact on hole fill at wave solder and on contact resistance during test, especially with PCB's using OSP surface finish. Higher contact resistance on bed-of-nails test fixture, such as for Manufacturing Defects Analyzers or In-Circuit Testers, results in false failures, causing excessive debugging and retesting. However, other measures can often be taken to achieve similar results without using nitrogen at reflow. Examples include:

- Solder paste selection
- PCB surface finish selection
- Printing solder paste on test pads rather than leaving them exposed during reflow
- Test probe head style selection

Examples specific to wave soldered boards include:

- Wave flux selection
- Wave flux application method
- Wave flux volume applied
- Wave flux distribution and depth of penetration in holes
- Wave solder parameters, such as:
    - Preheat configuration
    - Preheat profile
    - Wave solder alloy selection

  - Wave solder pot temperature
  - Solder wave dynamics

Board Warpage

As mentioned in Section 1.2, because of the higher temperatures required for Pb-free assembly, boards may sag and warp more than during SnPb assembly. This is particularly noticeable on thin PCB's, such as those used in mobile applications. Although there is no common industry specification for the warpage of assembled boards (only for bare boards), board assemblers may prefer to reduce warpage.

- A picture-frame style pallet, with hold downs, can be used to support the board on all four sides
- Channel-type support rails can be attached to the leading and trailing edge of the board, after solder printing but prior to reflow
- Oven manufacturers offer various center support mechanisms, such as an adjustable center support wire or chain, but this imposes a placement stay-out zone on the design

Warpage amount varies with PCB size / thickness / laminate, number of reflow cycles, and warpage control method.

Double Sided SMT Board Assembly

Both primary and secondary side reflow profiles should meet the same target specification. Because the board assembly has greater thermal mass during second reflow, different oven settings may be required for first and second reflow, in order to meet the same target profile. This requires two profile boards to simulate actual board configuration (thermal mass) during each respective reflow.

Sample Reflow Parameters

Table 0-1 Sample Reflow Parameters

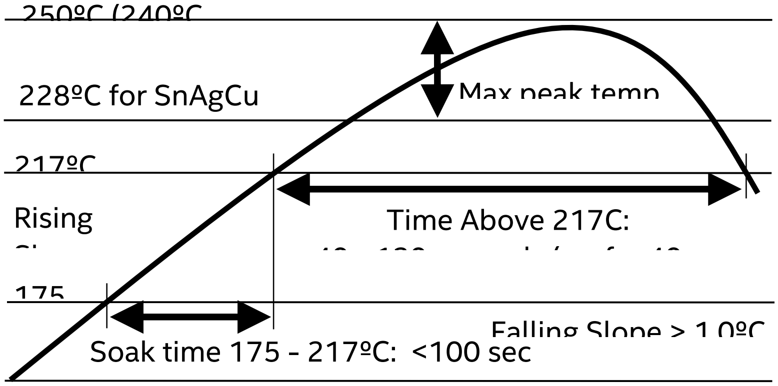
Solder paste	No-clean, flux class ROL0 per J-STD-004 Alloy Sn/3Ag/0.5Cu or Sn/4Ag/0.5Cu Metal content 89%

Soak	Paste dependent: Consult paste manufacturer.
Ramp rate - rising and falling	Rising: < 3°C per second, up to peak temp Falling Maximum 3° per second. Minimum 1°C per second, from peak temp down to 205°C
Time Above 220°C	Prefer 40-90 seconds for N_2 (0_2 < 3000PPM) reflow. 60-90 sec for Air reflow.
Minimum solder joint peak temp	228°C, if all BGA balls on board are SnAgCu 230°C, if any BGA balls on board are SnAg
Maximum solder joint peak temp	Never exceed 260° C
Maximum substrate body temp	Never exceed 260° C
Oven type	10 zone forced convection oven

- Sample process applies to all types of SMT components on the board, not just the BGA's
- All temperatures are measured with thermocouples inside solder joints, for better accuracy
- Max temp applies to the hottest joint on the board, typically a joint of a small passive device
- Reference process applies to all PCB's with nominal thickness 0.040" to 0.077" (1.02 to 1.96mm), and to PCB's 0.078" to 0.093" (2.0 to 2.36 mm) with large active devices on one side only
- Thick PCB's with thermally massive parts on both sides may require adjustments to Peak Temp and TA217°C

Below is a graphical representation of information in the table above.

Figure 0-3 Sample Reflow Parameters

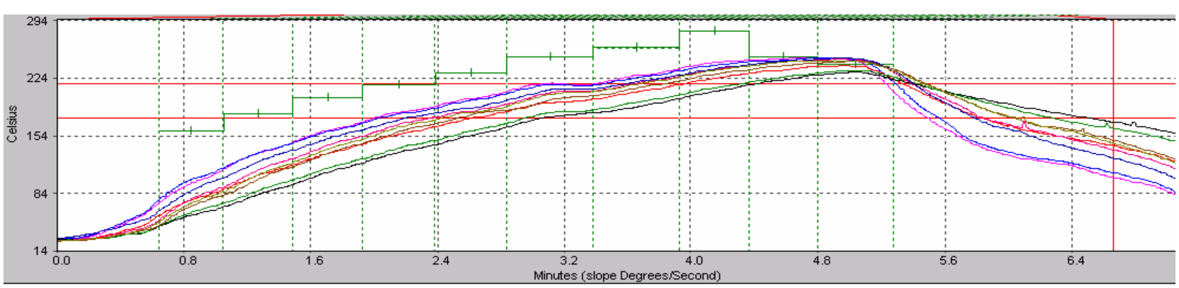


Soak time 175 - 217ºC: <100 sec
(Soak specs are paste dependent: May be different for different pastes.)

Sample Reflow Profiles

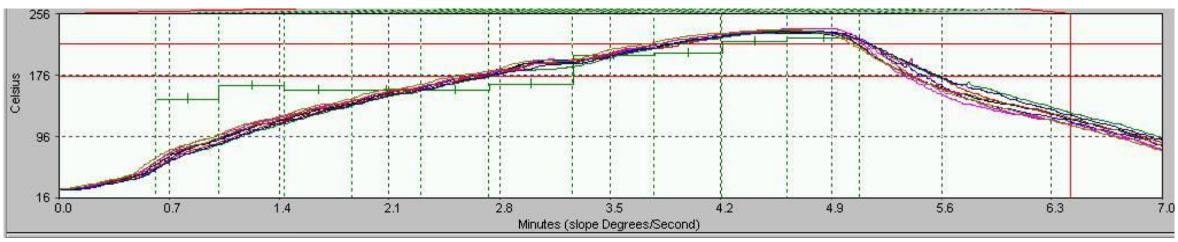
Here are sample reflow profiles for certain Pb-free desktop, mobile, and server boards. These are not meant to indicate reflow requirements, merely to illustrate typical reflow profiles.

Figure 0-4 Sample Desktop Reflow Profile



	Peak	Max Rising Slope	Max Falling Slope	Time Between 175/217	Total Time Above 217
1 U1F2	247.8	2.51	-3.17	76.36	117.28
2 0603 CAP	247.3	2.85	-2.96	85.17	109.76
3 SKT CORNER	238.7	2.08	-1.67	80.07	82.12
4 SKT CENTER	230.6	1.37	-0.84	81.29	57.94
5 SKT LEVER	232.9	1.53	-1.01	81.88	65.62
6 MCH CENTER	243.5	2.14	-1.54	78.74	97.92
7 MCH CORNER	246.5	2.34	-2.07	77.22	104.09
8 U4A1	247.3	2.31	-2.24	84.23	107.56
9 U5G1 ICH	245.0	2.07	-1.55	75.10	106.78
TC Range	17.2	1.48	2.33	10.06	59.34

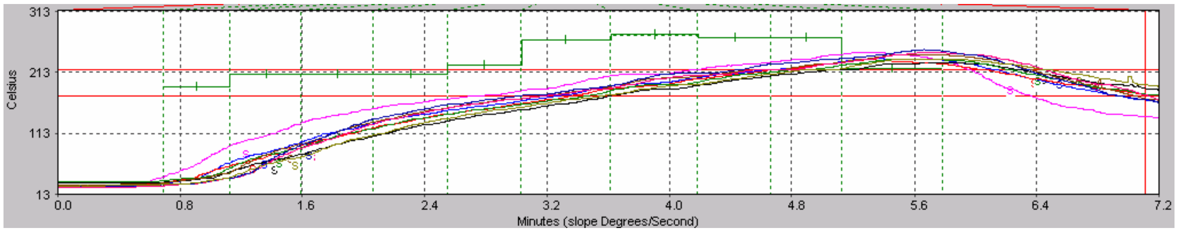
Figure 0-5 Sample Mobile Reflow Profile



	Peak	Max Rising Slope	Time Between 175/217	Total Time Above 217
1 MCH3	233.3	2.42	77.14	71.74
2 MCH3	233.6	2.24	76.31	74.15
3 CPU1	232.6	2.04	76.10	70.06
4 CPU1	231.2	2.35	76.55	71.18
5 MCH1	232.1	1.95	70.29	72.66
6 MCH1	233.7	2.26	70.34	74.69
7 ICH1	236.6	2.43	71.24	79.40
8 ICH1	233.1	2.09	70.05	74.36
9 EDEN	234.9	2.49	72.25	77.87

Figure 0-6 Sample Server Reflow Profile

	Peak	Max Rising Slope	Max Slope	Time Between 175/217	Total Time Above 217
1 Q8A2	246.3	1.67	1.67	81.60	109.04
2 J8K1 COR	235.9	1.46	1.46	80.75	89.76
3 J2K1 CNT	233.1	1.26	1.26	79.33	83.02
4 U5G2 CNT	229.0	1.11	1.11	86.70	73.78
5 U5G2 COR	233.6	1.34	1.34	85.55	80.43
6 U15	242.2	1.56	1.56	92.16	99.11
7 U3 CNT	245.5	1.72	1.72	83.43	112.21
8 C3A16	249.4	1.88	1.88	84.57	113.53
9 J100 CNT	234.6	1.29	1.29	78.18	96.43



Rework

Rework Profile Board Preparation

Because a rework profile is developed for a single component at a time, rather than the entire board, each component can have many thermocouples (TC's) on it, rather than the one or two locations used on reflow profile boards.

Thermocouples should initially be located at:

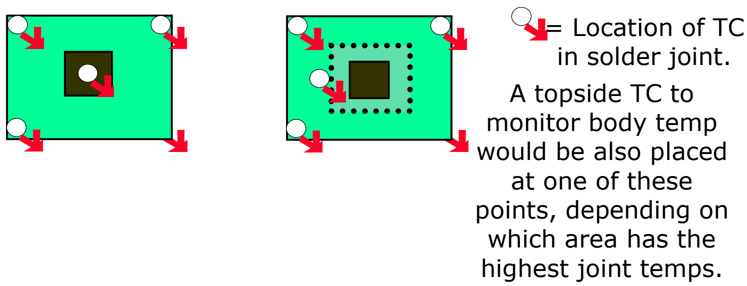
- Solder joints at all four corners of the hot air reworked component
- A solder joint at the center of the part, or as near to the center as possible, to represent the coolest joints on the component
- Sockets with actuating mechanisms may require an additional TC at a joint near the mechanism, if its mass could make that area harder to heat

After developing the initial profile, place an additional thermocouple at the topside location corresponding to the thermocouple with the hottest joint temperature.

- Because of the nature of hot air rework, and the variety of nozzle designs, there may be a significant temperature gradient across the part during rework
- Therefore, monitoring body temperature with a thermocouple only in the center may not represent what the rest of the body is exposed to
- Use this topside thermocouple to confirm that component body temp is not exceeding its max rating. Adjust profile if needed.

Here are examples of thermocouple locations for rework profiling on BGA's or sockets, for both fully populated arrays and partially populated arrays (no balls in the center area of the part).

Figure 0-7 Rework Profiling TC Location Example (BGA or Socket)



Fully populated array. Partially populated array.

Thermocouples to measure joint temperatures are installed through holes in the board, using the same method described earlier for reflow profile boards.

Pad Cleanup After Component Removal

While wicking solder off of pads:

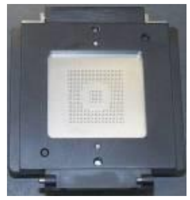
- Always clip off the used portion of the wick; it behaves as a heat sink
- Apply liquid flux to the wick, to minimize sticking of the wick to the pads
- Place the soldering iron on the solder wick off to the edge of the pads being soldered, to heat iron tip and wick prior to de-soldering
- Do not let the solder iron or wick freeze on pads, to prevent pad lift
- Do not lift the iron or wick up and down on the pads.
- Apply very light pressure, similar to writing with a pencil. Soldering is achieved by temperature difference, not by tip pressure
- Apply heat for 2 to 3 seconds after solder melts. Total contact time may be 6 to 7 seconds. Excess heating causes solder brittleness and may lift pads
- Move the soldering iron in the same direction with each stroke, rather than going back and forth. Going back and forth overheats pads at the ends of the row, increasing potential for damage

Paste Printing Methods at Rework

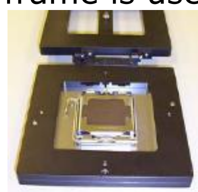
When new BGA's are installed at Rework, after pad cleanup, additional solder paste may not be required. The BGA ball can provide enough solder for a good joint. When new processor sockets are installed at Rework, additional paste is generally needed to ensure that good joints are formed, due to greater coplanarity differences.

Because of the difficulties of placing, aligning, and printing with a mini-stencil on the assembled PCB, a method is available that prints paste directly onto the socket BGA balls, rather than onto the PCB, as follows:

1. A stencil is inserted into the ball printing jig.

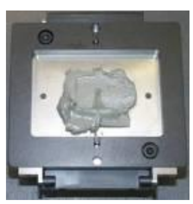


2. The part is hand placed with balls resting in stencil apertures.
(An alignment frame is used, but not shown.)



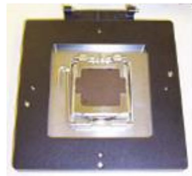
3. A clamping frame holds the part in place for later operations.

4. The jig is inverted, and paste is applied over the apertures.



5. A mini squeegee prints paste onto the balls, and removes excess paste.

6. The jig is inverted once more, the clamping frame is opened, and the part is removed by the rework machine's vacuum pick for placement onto the board.



Resulting side view of paste-printed balls.

Re-balling BGA's Not Recommended

Removed BGA's should be discarded. The re-balling process (placing new balls on removed BGA's, so that they can be re-used) is not recommended, for these reasons:

- Many BGA's (including Intel BGA's) are rated for three soldering cycles.
    - Re-balling exposes BGA's to more than three soldering cycles
    - (1) Initial installation, (2) Rework/removal, (3) Ball attach, and (4) Final installation
    - BGA's used on double sided boards could have even more soldering cycles
- Exposing BGA's to more than three soldering cycles may void the manufacturer's warranty (including Intel's)
- The intermetallic compound (IMC) layer, on the PCB and on the package, gets thicker with every reflow cycle. Wicking solder off the PCB or package pads does not remove IMC. Excessive IMC thickness can negatively affect solder joint reliability.

Sample Rework Parameters

Table 0-2 Sample Rework Parameters

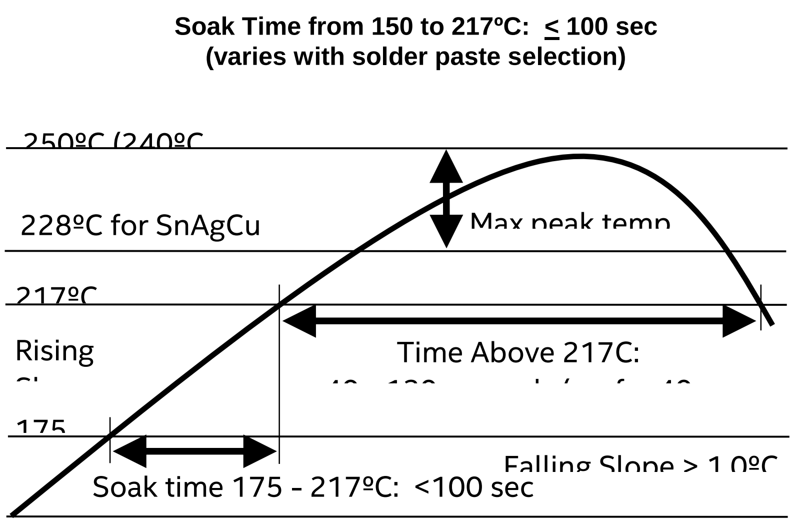
Reworked part type >	BGA's (and other array area packages)	Processor Sockets
Process parameters based on	Sn/4Ag/0.5Cu and Sn/3.5Ag ball alloys. PCB nominal thickness 0.062"-0.093" (1.6-2.4mm)	
Rework machine type	Hot air	
Flux	No clean, VOC-free, 18% solids, flux class ROL0, as per J-STD-004	
Flux applied to	Pads on board	
Solder paste	None	Same as used at SMT
Solder paste application	None	Printed on balls
Rising Ramp Rate below 205°C	0.5°C-2.5°C / sec	
Soak Time, from 150°C-217°C	< 100 sec (soak spec varies with solder paste selection)	
Critical Rising Ramp Rate between 205°C and 215°C	0.35°C-0.75°C / sec	
Peak Temperature Range	230°C-245°C	230°C-250°C
Time Above 217°C (TA217)	40-120 seconds	40-200 seconds
Delta-T (temp difference) across joints on part while above 217°C	≤10°C	≤15°C
Maximum Body Temp and Time	Not to exceed component supplier max specifications	
Except for body temps, all temperatures are measured at	At solder joints, with thermocouple in a hole at center of a pad.	
Falling Ramp Rate	0.5°C-2.0°C / sec	

Below is a graphical representation of information in the table above.

Figure 0-8 Sample Rework Parameters

Rework Profile Development

Because of the rework profile requirements, and because of the



interactions that each profile adjustment makes, it can be very difficult to develop a rework profile. This is especially true if all stages of the profile are targeted for development simultaneously. A recommended approach is to break the profile down into phases, and develop the first phase first, then the second, and so on.

Here are some recommendations for successful rework profile development.

- Maximize bottom heater temperature when creating the profile
- Keep the temperature of molten solder joints (above 217°C) within 10°C of each other across the component for BGA's, and within 15°C for sockets
- Create the profile in steps. Don't move on to the next step until the current step meets goals. Developing the entire profile at once can be overwhelming
  - Step 1: Board Preheat
    - Get joints into the 125°C to 150°C range before lowering the nozzle
  - Step 2: Soak
    - With nozzle down, get BGA joints into the 200°C to 220°C range and socket joints into the 190°C to 215°C range
    - Check the Soak Time spec

- Step 3: Peak Reflow
  - With nozzle down, meet specs for peak reflow range and Time Above 217°C
- Step 4: Cool Down
  - With nozzle up, get board cool enough to handle safely

- After developing the initial profile, check component body temperature to avoid exceeding Max component temps and times - adjust profile as needed.
- Place the body thermocouple at the topside location corresponding to the thermocouple with the hottest joint temperature

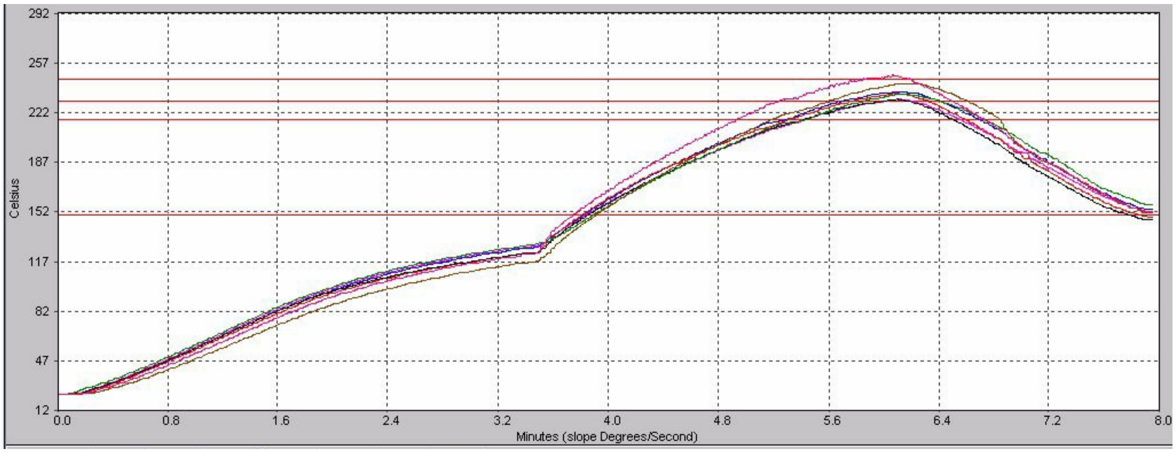
Below is a graphical representation of the steps listed above.

Figure 0-9 Sample Rework Profile Development

	Step 1 Board	Step 2 Soak, or	Step 3 Peak	Step 4 Cool
	Start with Preheat with bottom heater, before nozzle is	After nozzle is	Nozzle is	Nozzle rises
Target to exit	Solder Joint	BGA Solder Joint Temp:	Peak Temp Range,	Solder Joint Temp ≤ 80°C
Other specs	Rising Ramp	Rising Ramp Rate	Peak Temp Range.	Falling Ramp Rate

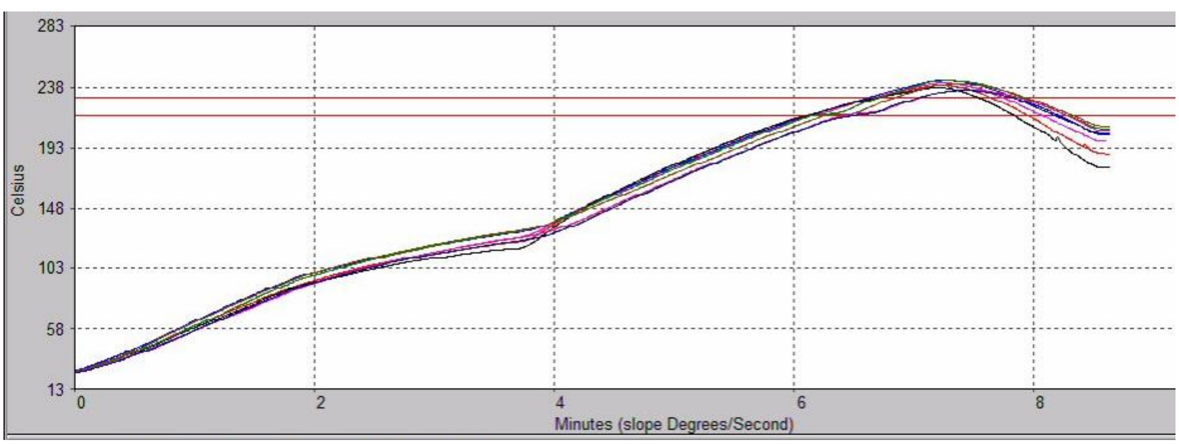
Sample Rework Profiles

Figure 0-10 Sample BGA Rework Profile



	Peak	Max Rising Slope	Max Falling Slope	Rising Time Between 150/217	Total Time Above 217	Total Time Above 230	Total Time Above 245
1 Outer (1)	231.3	1.13	-1.02	95.00	68.51	12.37	0.00
2 Outer	237.1	1.21	-1.20	88.50	84.56	42.38	0.00
3 Outer	235.5	1.50	-1.18	89.88	77.67	34.33	0.00
4 Outer	231.3	1.36	-1.16	93.83	65.76	11.94	0.00
5 Center	235.0	1.00	-1.09	93.33	77.60	33.74	0.00
6 Top of Pkg	242.6	1.43	-1.84	76.57	99.58	60.67	0.00

Figure 0-11 Sample Socket Rework Profile



	Peak	Max Rising Slope	Max Falling Slope	Total Time Above 217	Total Time Above 230
1 A1 Out Crnr	240.7	0.78	-0.72	114.25	64.67
2 Out Crnr (CW)	242.5	0.72	-0.68	121.74	72.70
3 Out Crnr (CW)	239.1	0.96	-0.91	109.73	58.32
4 Out Crnr (CW)	237.4	1.11	-1.33	101.64	50.05
5 A1 In Crnr	242.5	0.71	-0.65	125.46	71.75
6 In Crnr (CW)	240.1	0.73	-0.64	120.50	65.66
7 In Crnr (CW)	235.5	0.69	-0.71	110.57	49.52
8 In Crnr (CW)	235.4	0.70	-0.75	101.15	46.51

Revision Summary

May 2010: Original publication
Mar 2016: Updated and remove references to SnPb processes.