Board Solder Reflow Process Recommendations - Leaded SMT 9

9.1 Introduction

In reflowed board assemblies, the solder joint quality is affected by several variables including the component, SMT (Surface Mount Technology) assembly conditions, solder wave conditions, solder paste, reflow profile, wave profile and board design.

At the component level, the solder-ball alloy solder-ball pad plating, the lead base metal, plating quality and lead coplanarity can affect solderability. The type of base metal used for leaded component assembly is typically chosen for its mechanical and thermal properties and compatibility with other assembly processes while the pad metal used to attach solder balls is chosen for it's adhesion qualities. At Intel, a plastic-leaded SMD (Surface Mount Device) may use alloy 42 or copper alloy as lead base metal. Copper alloy is sometimes preferred for its superior lead compliance and thermal conductivity. Package internal structure may be invisible from the outside, but it affects the heat capacity of the package and therefore has a significant effect on the speed with which the leads heat up.

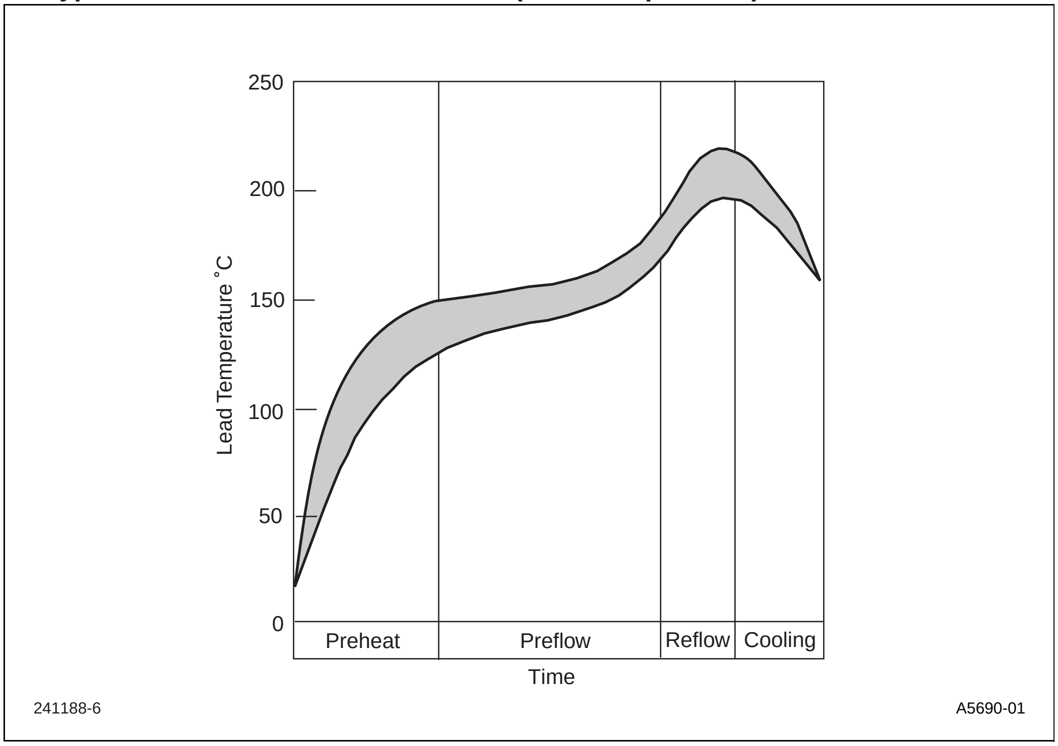
The lead base metal is plated with tin-lead solder to prevent corrosion and ensure good solderability. The tin-lead solder plate has a minimum thickness of 200 microinches and the plating composition can range from 75 to 95% tin. The leads are formed to meet industry specifications (JEDEC, EIAJ).

Another type of package that is used in reflow soldering is ball grid array (BGA), in various ball pitches from 1.27mm to 0.8mm or the shrink version that called chip scale package (CSP). In Intel, these components have balls that range from 0.3 to 0.36 millimeters. The balls composition is eutectic solder: 37% lead 63% tin.

In the reflow process, the solder paste must be heated above its melting point and be completely molten to fuse with the plating on the component lead to form the desired heel and foot fillets or to melt the ball and cause it to collapse and form the desired joint This solder joint formation mechanism depends on temperature and time which are reflected in the reflow profile. In leaded devices the volume of solder paste on the land is significantly greater than the plated solder volume on the component lead and is the key contributor to joint formation. However in BGA the balls on the component are the main contributor to the solder volume of the joint, never the less the paste volume which is 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. A typical IR/Convection reflow profile is shown in Figure 9-1.

Figure 9-1. A Typical IR/Convection Reflow Profile (Lead Temperature)



9.2 Methods Of Measuring Profiles

Profile measurement is a vital part of setting up the solder conditions. The measurements are usually carried out using thermocouples which either have long connecting wires allowing the recording equipment to be kept outside the furnace, or are attached to a high temperature resistant recording device which travels through the reflow furnace with the PCB under test. Special care must be taken to ensure proper placement of thermocouples to accurately measure the desired surface temperatures. For convenience, the thermocouple wires are taped/epoxied down on the surface which is to be measured or soldered to it with high temperature solder. For a large component, there may be significant temperature differences between the leads/balls, the package body and between different sides of the package. These differences should be minimized to achieve uniform solder joint formation. Obvious differences in lead/ball-temperatures between components also occur due to differences in the heating rates of components (as a result of differences in structure, for example, inner leads or balls of a square array will heat up slower than the leads of a peripheral array). It is recommended that a thermocouple be used to verify that the center balls on a Ball Grid Array package reach acceptable temperatures during the soldering process.

Attaching thermocouple to CSPor BGA is a challenging task. Usually a hole is drilled at the center and the thermocouple is routed through the bottom of the board and to the area of interest. In particular, since the CSP stand off is very small, a tunnel is milled next to the drilled hole and connected to the pad .This will allow the thermocouple to sit flush with the board. An easier method is to drill on the pad and attach the thermocouple from the bottom directly to the ball.

9.3 SMT Reflow Equipment

The major types of reflow technologies are listed below.

A. Vapor Phase: Uses the latent heat of condensation to reflow the solder. Has an oxygen-free atmosphere.

B. Infrared, Convection/Infrared, and Convection Ovens:

- Class I: Radiant Infrared Dominant: Relies on radiant IR energy as the principal heating medium.
- Class II: Convection/Infrared: Relies on a combination of radiant IR energy and convection as the principal heating mediums.
- Class III: Convection Dominant: Relies primarily upon convection as the principle heating medium. The gas ambient may be air or an inert gas, for example, nitrogen.

C. Laser: Uses laser scanning to heat up the lead surface to reflow the solder.

9.4 IR/Convection Reflow Profile

A typical reflow profile of an IR/Convection oven is shown in Figure 9-1 and key parameters are provided in Table 9-1. To establish a reflow profile, the conveyor speed and oven panel temperatures should be established. The repeatability of the reflow profile can be affected by the loading pattern and drastic changes in the surrounding environment such as exhaust rate and ambient temperature.

Table 9-1. Process Window for IR/Convection

Reflow Profile (Lead Temperature) Zones	Characteristic Description	Window/Limits
Preheat	Initial Heating of Lead/Component Peak Temperature in Preheat	1° C to 3° C/Sec 100° C to 140° C
Preflow	Dryout and Solder Paste Activation Soak Time Equalizes Component Temperatures	120° C to 170° C 60 to 120 Secs
Reflow	Time above 183° C Peak Reflow Temperature (Lead) Component Body Temperature Cooling Rate	60 to 120 Secs 205° C to 225° C (45s min >205C) Max 220° C 2° C/sec to 4° C/sec

NOTE:

1. The maximum body temperature for devices classified as moisture sensitive should be limited to 220° C. Maximum ambient exposure limits for devices must be observed prior to reflow to avoid excessive moisture absorption, see Chapter 8.
2. For Reflow, 60-120 seconds is only a guideline or starting point, and depending upon the solder paste manufacturer rec ommendations and line yield, these times could change. The board reflow process should also be aware of maximum and minimum temperature ramp rates which would also be provided by the solder paste manufacturer, but which is also NOT mentioned in this Packaging Databook.

9.5 Heating Rate And Preheat

To avoid submitting sensitive components to a thermal shock, board warping and excessively drying the solder-paste, the heating rate or ramp must be controlled. This ramp should be in the range of 1 to 3° C/sec and should minimize solder paste splatter. Special consideration should be given to the heating ramp for low-mass devices like CSP's.

9.6 Preflow

In the preflow zone the bulk of solvents in the solder paste evaporate and the flux in the solder paste is chemically activated. Larger components and boards may require longer dwell times to achieve uniform temperatures. Excessive temperature or time in this region may evaporate the flux especially if it is of the no-clean type (causing voids in BGA/CSP solder joints).

9.7 Reflow And Time Above Solder Melting Point At 183° C

While eutectic solder paste melts at 183° C to achieve proper wetting/wicking a temperature significantly higher must be maintained for proper solder reflow (see Table 9-1). Sufficient time must be allocated for foot and heel fillet formation. The leads of larger components need more time to heat up to the required temperature. Excessive dwell should be avoided to minimize intermetallic growth in the solder joint.

9.8 Peak Reflow Temperatures

The peak temperature of the solder joint during reflow should be high enough for adequate fluxing action and solder flow to obtain good wetting. However, it should not be so high as to cause component or board damage. In infrared reflow ovens, the peak reflow temperature is primarily controlled by the panel temperatures in the reflow zone and secondarily by the conveyor speed. The peak temperature of the SMD (Surface Mount Device) lead is in the range of 205° C to 225° C. In vapor phase reflow, the peak reflow temperature of the solder joint is determined by the boiling temperature of the primary fluid and the dwell time of the board in the primary zone. For both reflow processes, the peak reflow temperature of the SMC package body should not exceed 220° C, higher temperatures could increase the risk of internal package delamination of components identified as moisture sensitive (see Table 9-3).

9.9 Cooling Rate

The cooling rate of the solder joint after reflow is important because the faster the cooling rate the smaller the grain size of the solder. Smaller grain size solder has a higher fatigue resistance. If fans are used to cool the boards as they exit the oven, then a cooling rate of up to 6° C per second to the solder freezing point can be easily achieved but should not be exceeded as damage to the components or excessive board warp may occur due to the different expansion/contraction rates of different package/board constituents.

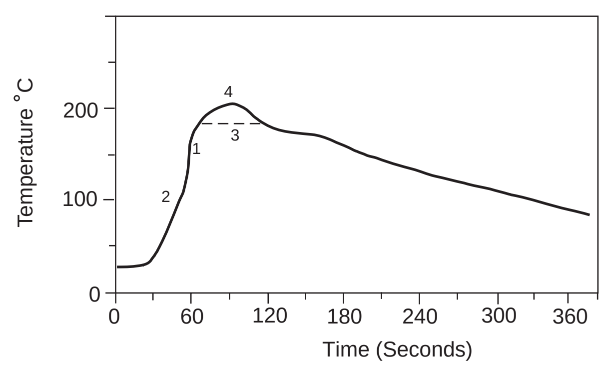
9.10 Nitrogen Atmosphere

Nitrogen atmosphere furnaces are strongly recommended if no clean fluxes are to be used. However, solder pastes are also available which achieve "no clean" results in a normal air atmosphere.

9.11 VPS Reflow Profile

A typical vapor phase (VPS) reflow profile of an SMD solder joint in batch-process equipment is shown in Figure 9-2. Table 9-2 lists the important characteristics and parameters for a VPS process. The actual board and lead surface temperatures may vary slightly. To establish a vapor phase profile for batch machines, the elevator speed and board dwell time in the primary zone must be established.

Figure 9-2. A Typical VPS Reflow Profile (Lead Temperature)



241188-4

A5691-01

Table 9-2. Process Window for VPS Reflow Profile (Lead Temperature)

Characteristic #	Characteristic Description	Vapor Phase
1	Maximum Heating Rate	6° C/sec
2	Peak Temperature in Preheat Zone	125° C
3	Duration of Time above Melting Point of Solder	Min—10 secs Max—80 secs
4	Peak Reflow Temperature	215° C +/- 5° C

9.12 Trouble Shooting

Table 9-3 outlines problems which could occur during board soldering.

Table 9-3. Board Soldering Diagnostics

Observation	Explanation	Probable Cause(s)
1. Insufficient or no Solder Fillet	Poor Fusing between Solder Paste and Plated Lead	1. Uneven Land and Lead Temperatures 2. Lead Coplanarity 3. Insufficient Wetting Time 4. Insufficient or Old Solder Paste 5. Peak Reflow Lead Temperature Too Low 6. Poor Solderability of Leads or Lands
2. Bridges and Icicles	Solder Connecting or Partially Connecting Adjacent Leads or Lands	1. Excessive Solder Paste 2. Excessive Component Placement Pressure 3. Solder Paste Misplacement 4. Solder Paste Integrity 5. Bent Leads 6. Board Vibrations during Soldering 7. Lead Plating Integrity 8. Excessively large voids in solder balls (BGA-only)
3. Dewetting	Solder Does Not Adhere to Lead or Land	1. Poor Solderability of Lands 2. Poor Solderability of Leads 3. Solder Paste Integrity 4. Lead Plating Integrity 5. In BGA apparent dewetting when excessive board warp occurs while solderballs are >160°C (actually a separation).
4. Solder Balls	Solder Agglomerates on Land or solderballs in adjacent PCB-area	1. Solder Paste Misplacement 2. Solder Paste Integrity 3. Insufficient Preheat 4. Solder-splatter Due to Excessive Heating Rate
5. Exposed Metal at Lead Tip	Poor or No Fillet Formation at Toe	1. Lead Trimming Exposed Base Metal, No Reliability Risk
6. Solder wicking up component leads	No or a poor heel-fillet	1. Lead temperature too high 2. Plating defect (porosity)
7. Package integrity failure such doming or cracking	Excessive expansion of absorbed moisture has separated internal parts of the package and deformed the plastic body	1. Excessive humidity during exposure of the devices to ambient 2. Out of bag time was exceeded 3. Excessive exposure time of devices to ambient air 4. Excessive humidity absorption of devices during transport or before dispatch

Table 9-3. Board Soldering Diagnostics

8. Voids in BGA Device	Voids in the BGA ball. The void could be near the ball and PCB interface or near the ball and the component substrate interface. It can be detected by X-rayor cross section	1. Plugged via under pad 2. Reflow parameters, temperature,time 3. Trapped flus in the solder paste which may be caused by: Excessively hot reflow profiles, Excessively small paste-powder sizes, Excessively high paste-solvent volatility, Excessively high paste metal content.

It is important to ensure that the thermal profile selected or set up is consistent with the properties of the solder-paste used. There are a few considerations which emphasise this;

1. The solderpaste must not dry out too much during the time that the PCB is stored prior to reflow
2. The solderpaste must not slump excessively
3. The size and distribution of solderballs in the paste must be compatible with stencil-printer minimum dimensions.
4. The solderpaste balls must not be excessively oxidised or deviate from a spherical shape.
5. The quantity deposited by the printing machine must be reproducible. Inspection and measurement may improve the prediction of the optimum stencil cleaning interval and paste replacement interval.
6. The preflow time and temperature must achieve activation of the flux but not excessively deplete the flux content, else adequate cleaning action prior to solderjoint formation may not occur.
7. The peak temperature must not be so high that the flux is burned, if it has to be cleaned off afterwards, nor must it allow the solder to wick up the leads of components and so starve the joints. The temperature difference between Leads and Lands also influences this effect.

9.13 Three Special SMT Challenges

SMT process engineers should be aware of internal advances in new large mass/low thermal resistance packages. For example, Intel's 208 lead Shrink Quad Flat Pack (SQFP) package has five different internal constructions. The internal construction features are designed to lower the thermal resistance between the die and the package leads. Thermal resistance between die and ambient is an important consideration for the design of system cooling. While these features were designed to conduct heat out of the component, they also conduct heat away from the leads during SMT mass reflow.

Intel has observed that this evolution of *larger* mass/*lower* thermal resistance packages may have contributed to a rise in open solder joints resulting in lower SMT line yields. Traditional solutions identified in Table 9-3 may be supplemented by these latest findings.

9.13.1 Challenge Number One

Large mass / low thermal resistance packages heat at a significantly slower rate than typical SMT components. To demonstrate these differences Intel ran experiments using various components. In Table 9-4, the leads of 208 LD SQFP (with heat slug) package are 33° C cooler than the 32-lead TSOP package and the leads arrive at peak temperature 30 seconds later.

Figure 9-3. Lead Solder Profile in IR Oven

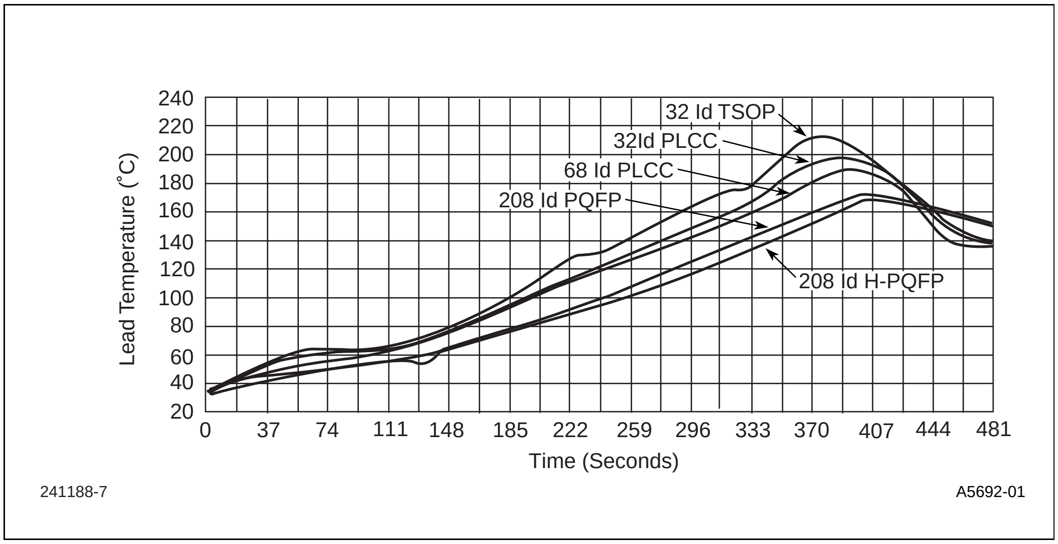


Table 9-4. Lead Peak Temperatures for Large Mass/Low Thermal Resistant and SMC

Package	Device	θ_{Jc}	Mass	Span	Peak Temp
32-lead TSOP	28F010	20° C/W	.37 gm	.79 in	220° C
32-lead PLCC	28F010	27° C/W	1.1 gm	.59 in	208° C
68-lead PLCC	80C196KB	12° C/W	4.8 gm	.99 in	194° C
208-lead PQFP	82425EX	8.5° C/W	6.2 gm	2.21 in	191° C
208-lead H-PQFP	80486 DX4	1° C/W	11.0 gm	1.21 in	187° C

To minimize open solder joints the following considerations should be given special emphasis.

9.13.1.1 Suggested Actions

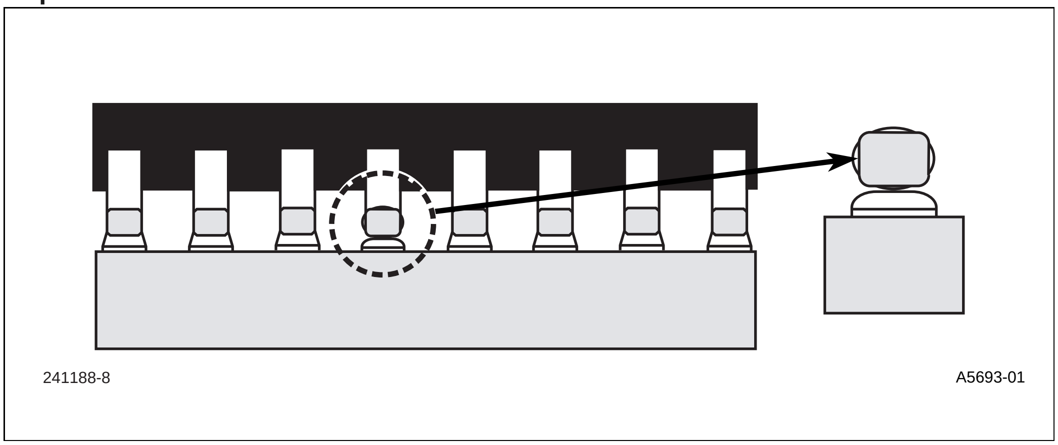
1. Review all IC data sheets to determine the largest mass / lowest thermal resistance devices.
2. Customize the oven profile.
   - — Key on *lead*, not package temperature.
   - — Evaluate ball temperatures in the center as as well as the corner of ball arrays (BGA-only).
   - — Slower belt speed may be required.
   - — More control zones may be needed.
   - — Change of PCB orientation on belt may help.
   - — Use of a PCB supporting frame may affect local board-temperature.
   - — Convection reflow oven may be needed .

Note: Bring all devices to temperature as uniformly as is possible.

9.13.2 Challenge Number Two

First-pass yield due to board (static/dynamic) warpage is harder to control with large components than with small. Intel evaluated several board cross-sections to determine yield problems associated with surface mounting large components. Analysis revealed that open solder joints occurred almost exclusively on center leads along the leading and trailing sides of large components (See Figure 9-4).

Figure 9-4. Open Solder Joints on Center Leads

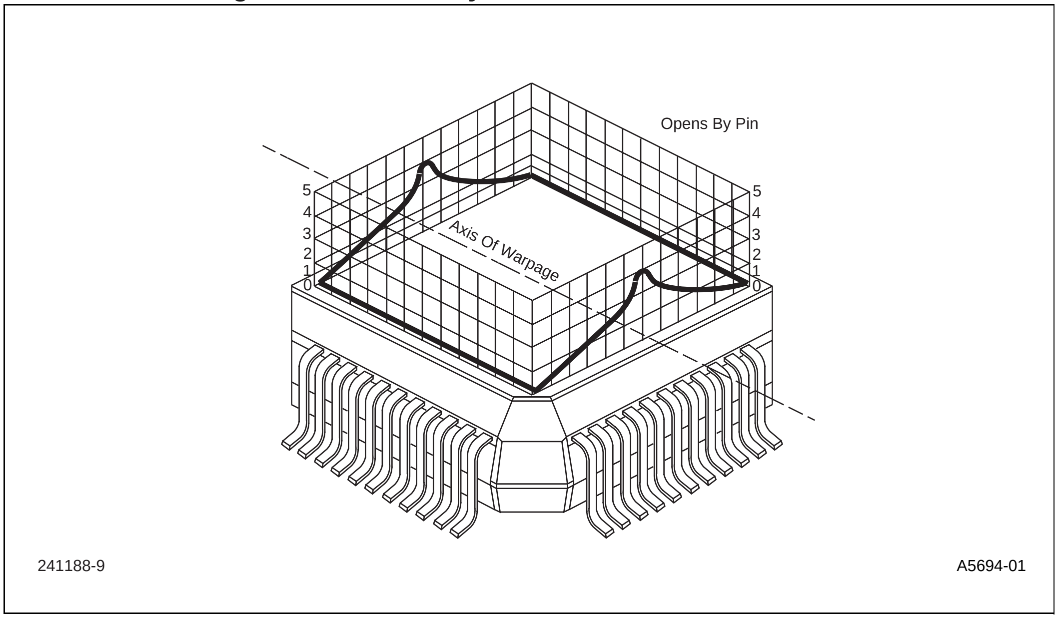


To minimize open solder joints, the following considerations should be given special focus when mounting large components.

9.13.2.1 Suggested Actions

1. Analyze in-circuit test (ICT) data to determine open pin failure locations using a histogram for each side of the component (see Figure 9-5).
2. Printed circuit boards may need custom support fixtures to minimize dynamic warpage during reflow.

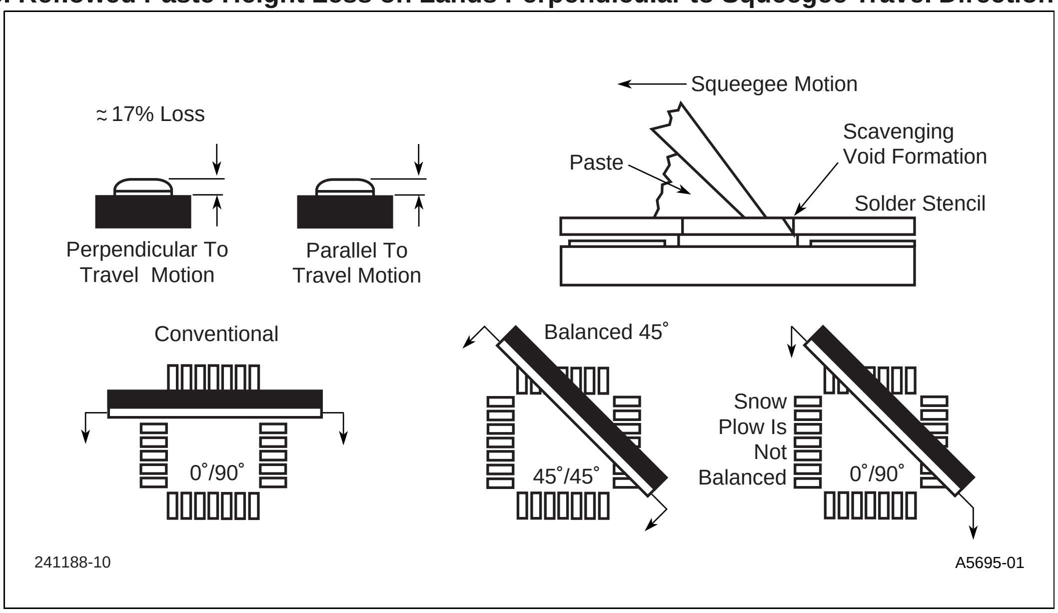
Figure 9-5. Four-Sided Histogram Used to Analyze ICT Data



9.13.3 Challenge Number 3

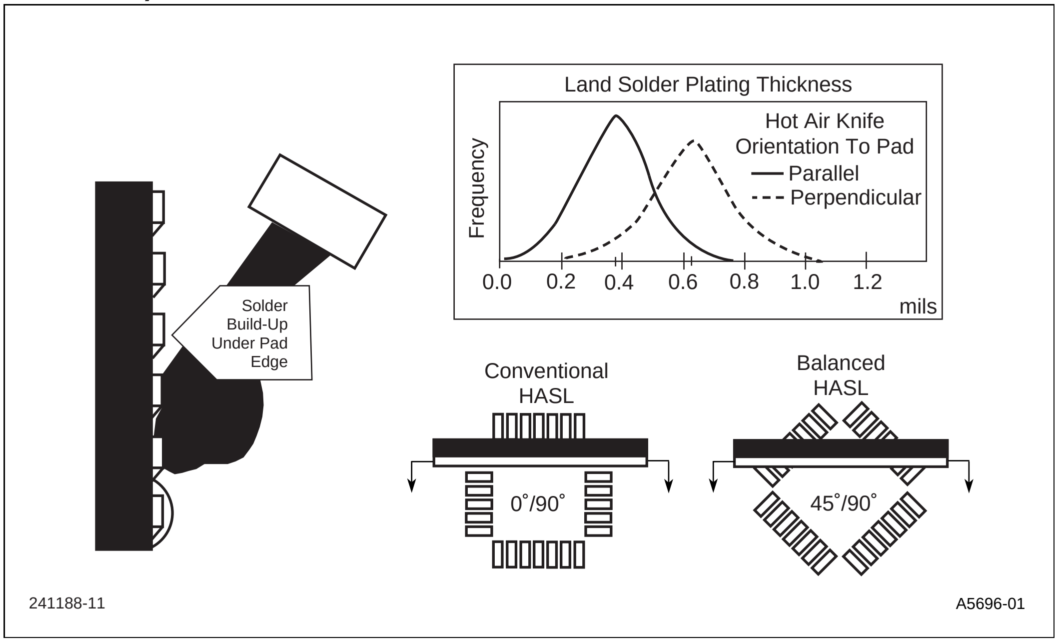
Solder paste height varies depending on land orientation. Intel evaluated cross-sectioned boards that were screen printed with solder paste and reflowed without components. This experiment revealed that the reflowed paste height was consistently .5 mils less on lands perpendicular to the squeegee travel direction (Figure 9-6).

Figure 9-6. Reflowed Paste Height Less on Lands Perpendicular to Squeegee Travel Direction



In addition, Intel evaluated board cross-sections that were reflowed without solder paste or components. Height measurements of the solder plated lands after reflow showed consistently 0.25 mils less on the lands parallel to the Hot Air Solder Level (HASL) direction of travel (see Figure 9-7).

Figure 9-7. Balance Land Solder Plating Thickness from HASL (Hot Air Solder Leveling) (PWB Fabrication)



To optimize yields, the solder paste height and/or land solder plated volumes should be controlled. The following suggestions may be helpful.

9.13.3.1 Suggested Actions

1. Move the squeegee and solder paste at a 45° angle onto the lands. This achieves more uniform solder paste volume within a given land pattern.
2. Move the hot air solder level at a 45° angle to the lands. This achieves more uniform solder volumes within a given land pattern.
3. If procedures #1 and/or #2 above are not feasible, then specify squeegee and HASL orientations so that they go in opposite directions to minimize the stacking tolerances.
4. Evaluate the use of a harder squeegee.

9.13.4 Challenge Number 4

Reflow first pass yield could be effected during wave soldering process. Intel evaluated numbers of low yield products which a BGA component was reflowed during wave soldering. Heat coming from the wave is increasing the temperature of the topside of the board. When the temperature is close to 183 deg BGA joints may weaken and be suceptable to stresses caused by board warp. BGA and CSP components are more suceptable to this kind of defect than leaded components as there is less strain relief.

To optimize yields BGA solder joints should be below 160 deg C during wave soldering. Special attention is needed regarding wave solder operation. The following suggestions may be helpful.

9.13.4.1 Suggested Actions

1. Reduce wave soldering preheat and pot temperature.
2. Vias should be capped or plugged by soldermask on the bottom side of the board.
3. External shielding on the bottom side of the board (opposite side of BGA component) to protect it from preheat and wave temperatures.
4. Special non metalic pallet with a shield under the BGA.

9.14 Rework Profiles

The three methods for reworking fine pitch packages (<50 mils) are: (1) manually reworking the package with a soldering iron, (2) directing a hot air stream at the package leads and removing the package when the solder melts, and (3) using focused InfraRed (IR) energy to heat the component leads and removing the package when the solder melts.

Manually reworking a package by a skilled rework operator has the least likelihood of damaging the printed circuit board. This method entails clipping all the leads of the defective package manually and removing the package. The leads, still attached to the lands, are then removed by melting the solder with a small tip, low wattage, soldering iron. The remaining solder on the lands is removed using solder wick. Putting a new package on the board is the most difficult part of the process. The best way to do this is to manually align the package on the lands with the aid of a microscope, then solder a couple of leads to the lands to keep the package from shifting during rework. The leads of the package are soldered one at a time. This method requires the most skill and is not applicable to BGA.

Hot air removal is by far the most widely used method of removing fine pitch packages. Hot air is directed onto the leads of the defective package until the solder melts and the package is lifted off by vacuum (either a wand or nozzle) or with tweezers. The lands on the board are cleaned with solder wick. Solder paste is reapplied (see Figure 9-7). The new package is placed. Hot air is then applied again to the solder leads to reflow the solder paste.

Focused IR is a relatively new rework process but works similarly to hot air. An IR lamp is used with a template that focuses the light on the defective package leads until the solder melts. The package is then lifted off by a vacuum wand or tweezers. The advantage of the IR rework method is that the heat is directed onto the leads and the package body stays cooler than when using the manual or hot air methods.

Table 9-5. Pros and Cons of Three Rework Methods

Rework Method	Pros	Cons
Manual Rework	• Inexpensive Rework Equipment • Minimum Package and Circuit Damage	• High Degree of Skill Needed • Process Good to 20 mil Pitch (0.5 mm) • Tedious and Time Consuming • Not applicablt to BGA
Hot Air Rework	• Repeatable Process • Less Time Consuming • Process Good to 16 mils Pitch (0.4 mm)	• Center of Package may become very hot • May Burn and Warp Circuit Board • Expensive Equipment

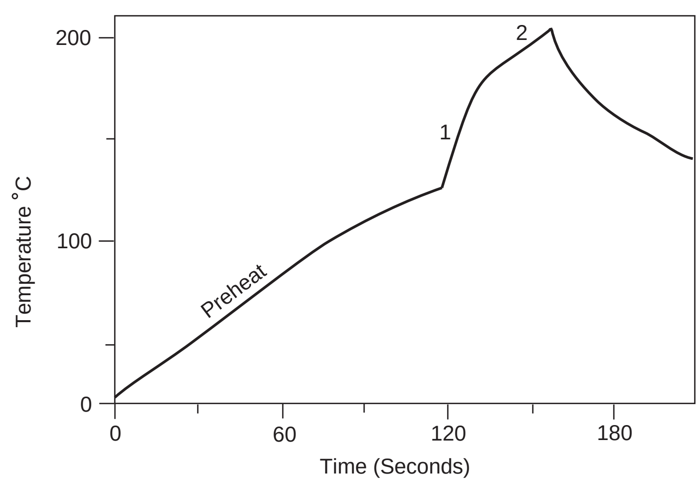
Table 9-5. Pros and Cons of Three Rework Methods

Focused IR Rework	• Center of Package Stays Cooler • Repeatable Process • Less Time Consuming • Process Good to 15 mils Pitch	• May Burn and Warp Circuit Board • Expensive Equipment

9.14.1 Peak Temperature

The peak temperature of the leads during the rework procedure should obviously be greater than the melting point of the solder in the solder joint. However, if the peak temperature is too high the component or the printed board can be damaged by excessive heat. Figure 9-8 shows a rework heating profile. Ensure that the package body temperature is kept at 220° C or less.

Figure 9-8. A Typical Temperature-Time Profile of a SMT Component While the Package was Being Reworked with a Hot Air Rework Machine



241188-5

A5697-01

9.14.2 SMD Board Removal Techniques

Intel recognizes the need for SMD board removal procedures that maintain the electrical and mechanical integrity of the component leads. The requirement for electrical testability of removed SMDs is driven by the need to

- confirm device failure as the cause of board-level nonfunctionality
- verify that the device also fails Intel test conditions
- allow electrical testing to be carried out as part of the component failure analysis procedure.

Therefore, the following removal procedure has been developed to ensure component mechanical integrity, including lead coplanarity and lead finish integrity.

9.14.3 Materials for Removal of SMDs

Materials used in the removal of SMDs are as follows:

- PCB with device to be removed
- SRT 1000 (or newer) series, Pace Craft-100, Air-Vac DRS-21*, or equivalent hot air solder reflow system
- Nozzle for optimized air flow. The nozzle is package dependent.
- Vacuum cup. The vacuum cup should be large enough to cover the center of the device without extending over the leads.
- Tweezers
- Safety glasses
- Pallet to retrieve component once it is removed
- Solder wick or hot air gun/blade

9.14.4 Board Preparation

Many plastic surface mount components and some ceramic chip components are moisture-sensitive. In addition, glass fiber-reinforced PCB materials can be subject to measeling and other thermal humidity-induced damage.

Moisture-sensitive surface mount components (and PCBs) must be dried by a prebake prior to rework to prevent damage to the board and the component. The duration of the prebake depends on the maximum temperature the populated PCB is exposed to. For example, a time of 24 hours at 70° C (in a low humidity environment) has been found to be workable. If the component selected for replacement has been exposed to excessive moisture, then it is necessary to dry it also. The bake time must be at 125° C for 24 hours or 40° C for 192 hours. If the component for rework is not rated moisture-sensitive, then the standard assembly handling and preheat procedures should be followed (see Section 9.14.5 “Removal Procedures”). See Chapter 8 for more information regarding Rebake.

In the board rework process, the component package body is not heated directly by hot air, but it can reach significant temperatures due to radiant and convective heating from the reflow air nozzle. Therefore, rework procedures should be calibrated to prevent the package body temperature from exceeding 220° C. Hot air or focused IR rework equipment may be used. Focused IR is advantageous because the plastic package body stays cooler than the package leads.

Heating of the package may also significantly raise the PCB temperature in adjacent areas. This may cause deformation of the board and in extreme cases may make it impossible to resolder a component in the repair location. To reduce this warping of the PCB it is recommended that the board is heated from underneath to a temperature of 80-120° C. This reduces the amount of heat which needs to be added by directing a hot-air stream at the component-leads, and reduces the temperature differential for the PCB between the repair area and other parts of the PCB, thus reducing sagging. The PCB must be adequately supported to reduce sagging.

---

* The above is not an endorsement of any kind nor a warranty of performance of the equipment or company.

9.14.5 Removal Procedures

The following procedure is useful for removing SMDs from the PC board using a hot air system.

1. Turn on the rework equipment at least 20 minutes before rework to ensure that the air flow is at the correct temperature. Insert the PCB into the work holder.
2. Adjust the work holder so that the device is centered under the hot gas nozzle. Lower the nozzle over the device and align. The alignment of the nozzle to the device is critical to ensure a uniform heating rate of the leads.
3. Raise the nozzle and lock it into position. Lower the vacuum cup and check to ensure that it is the proper size.
4. Add flux to the solder joints to improve solder reflow and reduce icicling.
5. To reflow the solder, set the timer for the approximate dwell time for the package type and size. Table 9-6 gives some guidelines. Pressure and time can be adjusted to optimize the process, but any modifications for increased throughput should not increase the temperature of the component body above 220° C.
6. Lower the nozzle until it is 5 mils –10 mils above the shoulder of the leads. This height is crucial. If the nozzle is too high, then the heating time must be increased. If positioned too low, then the nozzle can damage the leads.
7. Using the automatic mode, reflow the solder. Once the air flow has shut off, immediately raise the component from the board. Icicling can be minimized or eliminated by removing the component from the board surface quickly using a smooth upward motion. The direction of removal must be normal to the plane of the board to prevent bridging. Do not twist or tilt the component during the removal process.
8. To retrieve the component once it has been removed from the board, use the appropriate tool. Turn off the vacuum and let the part drop onto the container. The drop distance should be minimized to prevent damage to the leads.

Note: If resistance is felt when lifting the component off the board surface, do not force the part off the board. Determine if the component has been adhesively attached to the board. If so, then alternative removal procedures not covered here are required for removal.

Table 9-6. Parameters for Component Removal (Based on the Pace Craft-100)

Package Type	Lead Count	T_{lead} Center (°C)	T_{lead} Edge (°C)	Min Time (sec)
PLCC	32 44 68	282 291 277	277 281 276	25 +5/-0 25 +5/-0 35 +5/-0
PQFP	100 132 164	270 280 280	250 250 250	35 +5/-0 35 +5/-0 35 +5/-0

NOTE:
1. Pace air stream temperature = 350° C
2. Pace N_2 pressure = 40 psi

If the component has not been glued on re-expose the unit to the heating cycle and then repeat the removal process. If the component still does not separate from the board, terminate the removal procedure.

9.14.6 Rework Of Lead Surface Morphology After Removal

It has been found that careful removal of SMDs using the above procedures greatly reduces the need to rework leads. If bridging or icicling occurs, then using a solder wick or hot air gun for rework can remedy the problem.

Bridging can be removed by using a solder wick. Holding the solder wick over the bridged leads, bring a fine-tip soldering pencil in contact with the bridged area. When the solder reflows, touch the wick to the molten solder, absorbing the excess solder. For BGA clean-up a broad-blade and solder wiick provides for quick and easy clean up of solder remnants.

Icicling can be eliminated by reflowing the solder with a hot air gun (T > 190° C). Place the part, leads up, on a heat-resistant horizontal surface. Run the hot air blade or gun across the affected leads, reflowing the solder. Allow the solder to fully resolidify before moving the component. Be sure that the hot air knife does not increase the localized package body temperature above 220° C.

Note: This procedure was developed on a Pace Craft-100 rework station. Critical configuration requirements include a hot air source capable of maintaining 350° C, a nozzle design that evenly distributes air flow to all four sides of the package, and a timer. *Intel recommends that customers calibrate their removal systems to achieve commensurate parameters.* Calibrate the component lead temperature, time to reflow, and capability for component removal normal to the plane of the PCB to correlate to this set of parameters.

9.15 Summary

There are three critical parameters to meet to ensure electrical testability of a component* removed from printed circuit boards:

- the solder must be completely molten before removal is initiated,
- the removal direction must be normal to the plane of the board, and
- no movement, in-plane or tilting of the removed component should occur until the solder completely resolidifies. Specific temperatures and dwell time at temperature are a function of package type and size.

* This applies to components that have not been adhesively attached to the board for double-sided board usage.

9.16 References

[1] Prasad and R. Aspandiar: “Systems Manufacturing with Fine-Pitch Devices,” *Printed Circuit Assembly,* March 1989, pp. 9-14.

[2] C. Alger, W. A. Huffman, S. F. Gordon, S. Prough, R. Sandkuhle, K.Yee, “Moisture Effects on Susceptibility to Package Cracking in Plastic Surface Mount Components,” *IPC Technical Review,* February 1988.

[3] R. Lin, E. Blackshear, G. May, G. Hamilton, D. Kirby, “Control of Package Cracking in Plastic Surface Mount Components During Solder Reflow Process,” Proceedings 7th International Electronics Packaging Conference, 1987, pp. 995-1009.

[4] M. McShane, “New Reliability Aspects of Surface Mount Devices,” Proceedings 1987 IEEE International Electronics Manufacturing Technology Symposium, pp. 87-88.

[5] H. Nurser, Techniques in the Development of High-Reliability, High-Density Surface Mount Packages, and Recommendations for Their Use,” European Surface Mount and Related Technologies (SMART) Conference, February 1989.

[6] L. Clifton, “Preconditioning of Plastic Surface Mount Components to Simulate Solder Reflow Prior to Reliability Stressing,” SMTA Conference Proceedings, August 1989.

[7] R. McKenna, “Surface Mount Device Package Cracking: An Overview,” SMTA Conference Proceedings, August 1989.

[8] Institute for Interconnecting and Packaging Electronic Circuits, “Impact of Moisture on Plastic I/C Package Cracking,” IPC-SM-786.

[9] B. K. Bhattacharyya, W. A. Huffman, W. E. Jahsman, and B. Natarajan, “Moisture Absorption and Mechanical Performance of Surface Mountable Plastic Packages,” IEEE Components, Hybrids, and Manufacturing Technology Proceedings, 1988, pp.49-58.

[10] Intel Package Connections, Spring 1992.

[11] P. Zarrow, “Optimizing IR Reflow”, Electronic Packaging and Production, Nov. 1990.

[12] C. Hutchins, “SMT/FPT Soldering Problems and Solutions”, Surface Mount Technology, July 1990.

[13] R. Bowlby, “Finish First”, Circuits Manufacturing, Nov. 1987.

[14] Manufacturing Advantage Tips , see URL : http://developer.intel.com/design/quality/index.htm

9.17 Revision Summary

- Complete review and edit of chapter.