9

Board Solder Reflow Process Recommendations

CHAPTER 9
BOARD SOLDER REFLOW PROCESS RECOMMENDATIONS

9.1. INTRODUCTION

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

At the component level, the lead base metal, plating quality and lead coplanarity can affect solderability. The type of base metal used for component assembly is typically chosen for its mechanical, thermal properties and compatibility with other assembly processes. At Intel, a plastic SMC (Surface Mount Component) may use alloy 42 or copper alloy for lead base metal. Copper alloy is sometimes preferred for its lead compliance and thermal conductivity. Package internal structure may be invisible from the outside but have significant effect on the speed with which it heats 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 minimal 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).

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. This solder joint formation mechanism depends on temperature and time which are displayed in the reflow profile. 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.

There is no one best reflow profile for all board assemblies. Ideally, a reflow profile must be characterized for each board assembly. 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.

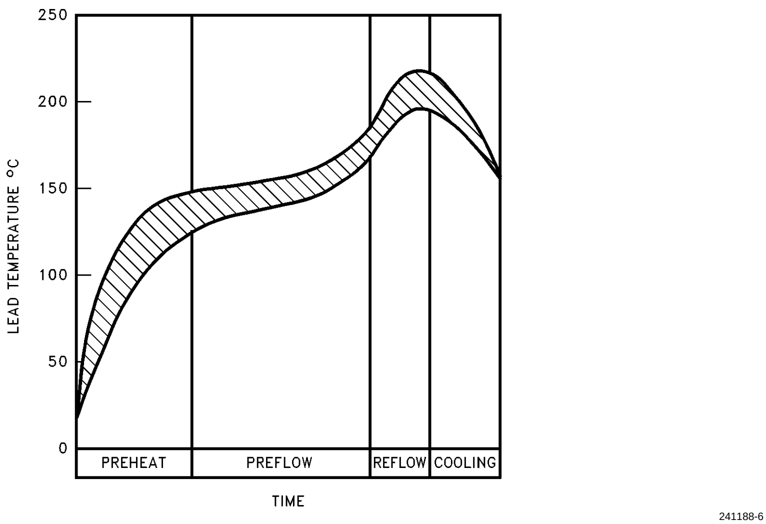

Figure 9-1. A Typical IR/Convection Reflow Profile (Lead Temperature)

9.2. METHODS OF MEASURING PROFILES

Profile measurement is a vital part of setting up the solder conditions. Special care must be taken to ensure proper placement of thermocouples to accurately measure the desired surface temperature. For convenience, the thermocouple wires are usually taped down on the surface to be measured. For a large component, there could be significant temperature differences between the lead and 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-temperature between components will also occur due to differences in heating-up rates of components as a result of differences in structure, e.g., square array versus peripheral. For example, it is recommended that a thermocouple is used to verify that the center balls on a Ball Grid Array package reach acceptable temperatures during the soldering process.

9.3. SMT REFLOW EQUIPMENT

There are various types of reflow technologies. The major types are:

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

B. **Conduction:** Essentially a conveyorized hot-plate. Solid to solid conduction of energy.

C. **Hot Bar:** Uses a metal contact bar to apply heat directly to the leads with minimal heat transfer to the package.

D. **Infrared, Convection/infrared, and convection ovens:**

Class I: Radiant Infrared Dominant: Relies on radiant IR energy as the principle heating medium.

Class II: Convection/Infrared: Relies on combination of radiant IR energy and convection as principle heating mediums.

Class III: Convection Dominant: Relies primarily upon convection as principle heating medium. The gas ambient may be air or an inert gas, e.g. nitrogen.

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

Types A, B, and D are considered mass reflow processes. Method D represents the vast majority of reflow operations.

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 have to 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 and Peak Temperature in Preheat	1°C to 3°C/Sec 100°C to 140°C
Preflow	Dryout & Solder Paste Activation Soak Time	120°C to 170°C 60 to 120 Secs
Reflow	Time Above 183C Peak Reflow Temperature (Lead) Component Body Temperature Cooling Rate	130 to 120 Secs 205°C to 225°C Max 220°C 2°C/sec to 4°C/sec

NOTE:

The maximum body temperature for devices classified as moisture sensitive should be limited to 220°C.

9.5. HEATING RATE & PREHEAT

To avoid submitting sensitive components to a thermal shock and dry the solder-paste, the heating rate or ramp must be controlled. This ramp could be in the range of 1 to 3°C/sec and should minimize solder paste spatter.

9.6. PREFLOW

In this zone, the bulk of solvents in the solder paste should have dried out and the flux in the solder paste is chemically activated prior to reflow. Larger components and boards may require longer dwell time to achieve uniform heating.

9.7. REFLOW AND TIME ABOVE SOLDER MELTING POINT AT 183°C

At 183°C, the eutectic solder paste will melt and wet the land, and subsequently wick up the component lead. Sufficient time must be allocated for foot and heel fillet formation. Larger components will need more time. 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 reflow 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 package body temperature should not exceed 220°C. Higher temperatures could increase the risk of internal package delamination for components identified as moisture sensitive. Package body temperature can be measured with a thermocouple attached to the center of the top surface of the package. In infrared reflow ovens, the peak temperature of the SMC (Surface Mount Component) 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. (See Table 9-1)

9.9. COOLING RATE

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, a cooling rate of up to 6°C per second, to the solder freezing point, can be easily achieved.

9.10. NITROGEN ATMOSPHERE

Nitrogen atmosphere furnaces are being used with the no clean fluxes. It should be noted, that solder pastes are offered which will achieve “no clean” results in a normal (air) atmosphere.

9.11. VPS REFLOW PROFILE

A typical vapor phase reflow profile of a SMC solder joint in a batch oven is shown if 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 have to be established.

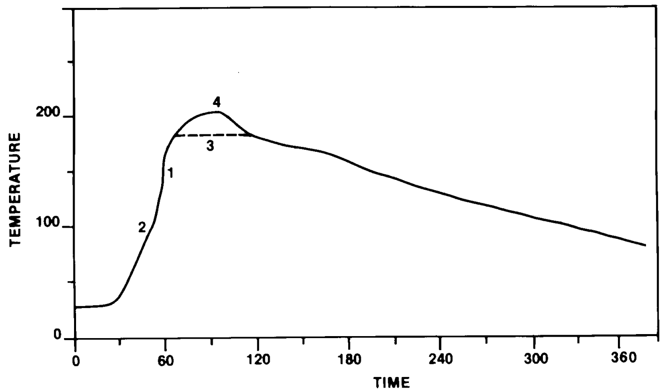

Figure 9-2. A Typical VPS Reflow Profile (Lead Temperature)

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 +/- 5°C

9.12. TROUBLE SHOOTING

Table 9-3 outlines potential board soldering problems.

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 Solder Paste 5. Peak Reflow Lead Temperature Too Low 6. Poor Solderability of Leads
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
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. Solder Balls	Solder Agglomerates on Land	1. Solder Paste Misplacement 2. Solder Paste Integrity 3. Insufficient Preheat
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)

9.13. 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. This factor 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 quiet 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-**Error! Bookmark not defined.** may be supplemented by these latest findings.

9.13.1. Challenge Number 1

Large mass / low thermal resistance packages heat significantly slower than typical SMC 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.

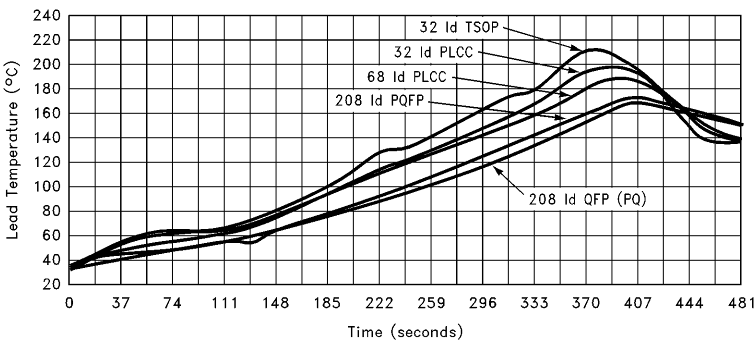

Figure 9-3. Lead Solder Profile in IR Oven

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 QFP (PQ)	80486 DX4	1°C/W	11.0 gm	1.21 in	187°C

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

9.13.1.1. CONSIDERATIONS

1. Review all IC data sheets to determine the largest mass / lowest thermal resistance devices.
2. Customize oven profile.
   - Key on *lead*, not package temperature.
   - Slower belt speed may be required.
   - More control zones may be needed.

Bottom line: Bring all devices to temperature as uniformly as possible.

9.13.2. Challenge Number 2

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

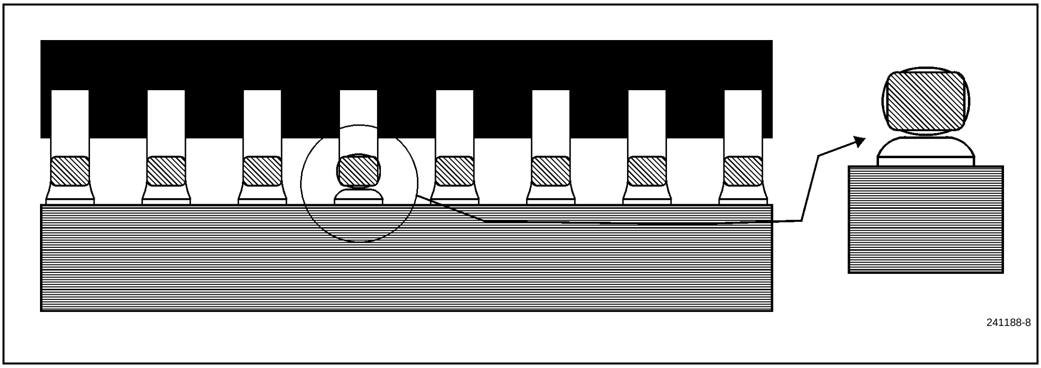

Figure 9-4. Open Solder Joints on Center Leads

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

9.13.2.1. CONSIDERATIONS

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

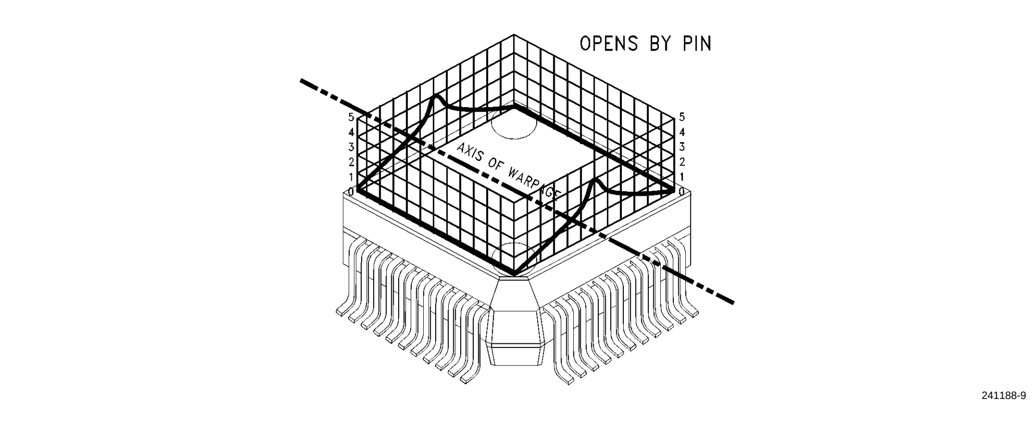

Figure 9-5. Four-Sided Histogram Used to Analyze ICT Data

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).

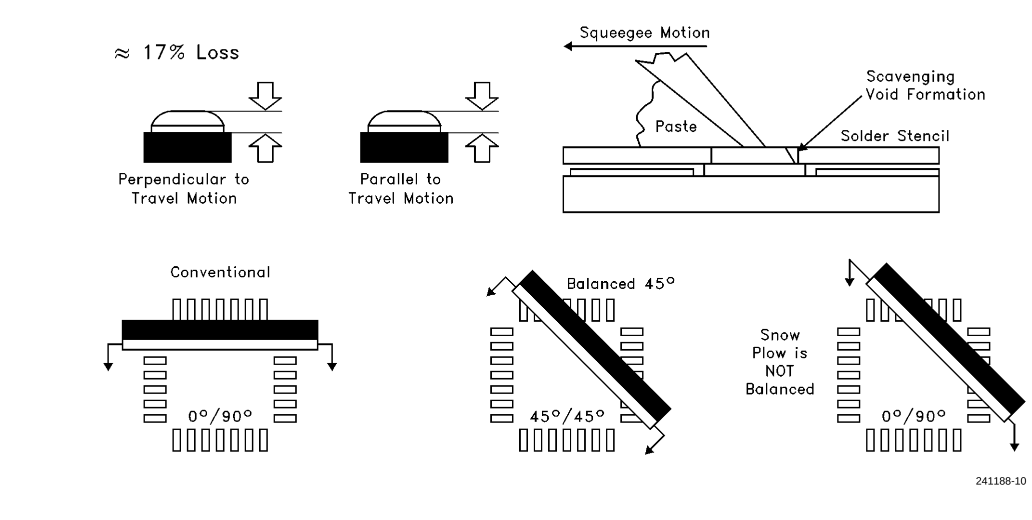

Figure 9-6. Reflowed Paste Height Less on Lands Perpendicular to Squeegee Travel Direction

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 that are parallel to the Hot Air Solder Level (HASL) direction of travel (see Figure 9-7).

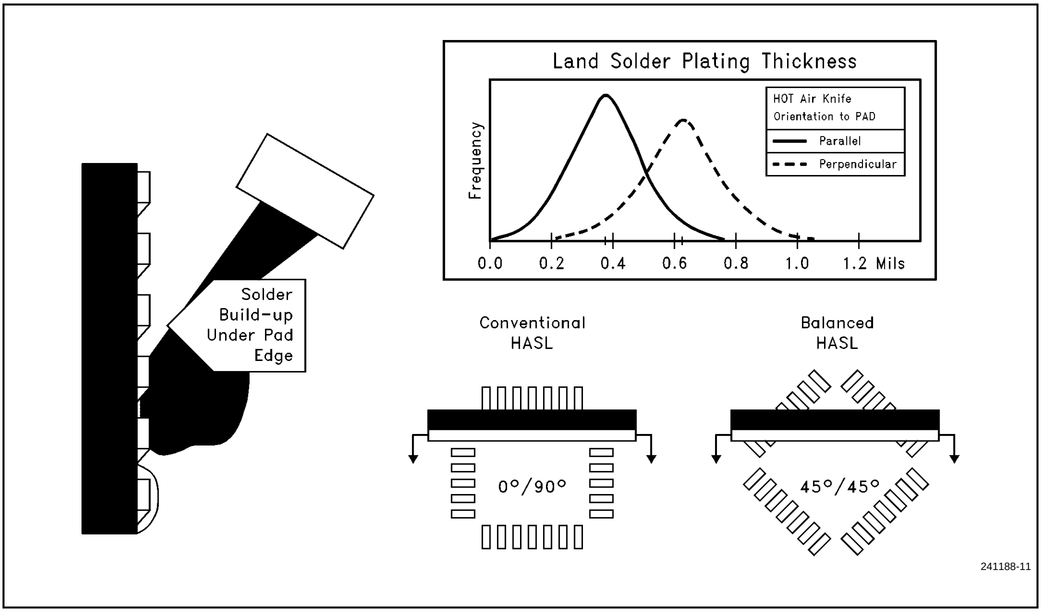

Figure 9-7. Balance Land Solder Plating Thickness from HASL (Hot Air Solder Leveling) (PWB Fabrication)

To optimize yields, the solder paste height and/or land solder plated volumes should not be overlooked. The following suggestions may be helpful.

9.13.4. Considerations

1. Move the squeegee and solder paste at a 45° angle onto the lands. This will achieve 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 will achieve more uniform solder volumes within a given land pattern.
3. If #1 and/or #2 above are not feasible, specify squeegee & HASL orientations such that they go in opposite directions to minimize the stacking tolerances.

9.14. REWORK PROFILES

The three methods for reworking fine pitch packages (<50 mils) are: (1) manual rework with a soldering iron, (2) direct a hot air stream at the package leads and remove the package when the solder melts, and (3) use focused InfraRed (IR) energy to heat the component leads and remove 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 the ensuing rework. The leads of the package are soldered one at a time. This method requires the most skill.

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 up with solder wick. Solder paste is reapplied (see Figure 9-2). The new package is placed. Hot air is then applied again to the solder leads to reflow the solder paste.

Focused IR is a fairly new rework process but works similar 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.

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

	Pros	Cons
Manual Rework	Inexpensive Rework Equipment Minimum Package and Circuit Damage	High Skill Needed Process Good to 20 mil Pitch (0.5 mm) Tedious and Time Consuming
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
Focused IR Rework	Center of Package Stays Cooler Repeatable Process Less Time Consuming Process Good to 15 mils Pitch	Expensive Equipment May Burn and Warp Circuit Board

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.

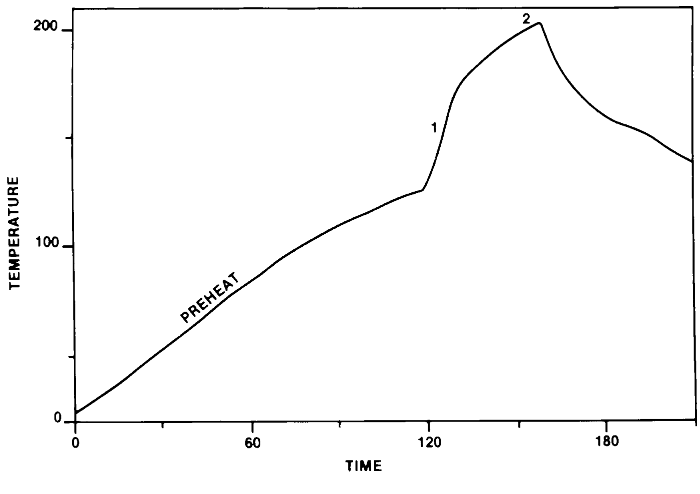

Figure 9-8. A Typical Temperature-Time Profile of a SMT Component While the Package was Being Reworked with a Hot Air Rework Machine

9.14.2. SMC Board Removal Techniques

Intel recognizes the need for SMC board removal procedures that maintain the electrical and mechanical integrity of the component leads. The requirement for electrical testability of removed SMCs is driven by the need to (1) confirm device failure as the cause of board-level nonfunctionality, (2) correlate device failure between the customer board assembly site and Intel, and (3) 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. Recommended Materials for Removal of SMCs

Materials used in the removal of SMCs are as follows:

- PCB with device to be removed
- Pace Craft-100, Air-Vac DRS-21, or equivalent[1] 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 PCB's) must be dried by a prebake prior to rework to prevent damage to the board and the component. The duration depends on the maximum temperature the populated PCB may be exposed to. For example a time of 24 hours at 70°C is recommended. If the component selected for replacement has been exposed to excessive moisture, it is necessary to dry it also. The bake time can be at 125°C for 24 hours or 40°C for 192 hours. If the component for rework is not rated moisture-sensitive, standard assembly handling and preheat procedures should be followed (see section on "Removal Procedures").

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.

---

[1] 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 procedures are recommended for removing SMCs from the PC board; using a hot gas system:

- Turn on the rework equipment at least 20 minutes before reworking to ensure that the air flow is at the correct temperature. Insert the PCB into the work holder.
- 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.
- Raise the nozzle and lock it into position. Lower the vacuum cup and check that it is the proper size.
- Flux can be added to the solder joints to improve solder reflow and reduce icicling.
- 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.

  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, the heating time must be increased; if positioned too low, the nozzle can cause damage to the leads.
- 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.
- 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 in order not to damage the leads.
- To prevent PCB damage and warpage, preheat the PCB by underboard heating an support the PCB both at the edges and underneath the component. For some package types (BGA) this is a necessity.

NOTE

If resistance is felt when lifting the component off the board surface, do not force the part. Re-expose the unit to the heating cycle and then repeat the removal process. If the component still will not separate from the board, terminate the removal procedure. Determine if the component has been adhesively attached to the board. If so, alternate removal procedures–not covered here–will be 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
Cerquad	32 44 68	269 289 251	271 277 252	55 +5/-0 55 +5/-0 60 +5/-0

NOTES:

Pace air stream temperature = 350°C

Pace N_2 pressure = 40 psi

9.14.6. REWORK OF LEAD Surface Morphology After Removal

Intel has found that careful removal of SMCs using the above recommended procedures greatly reduces the need to rework leads. If bridging or icicling occurs, Intel recommends the used of solder wick or a hot air gun for rework.

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.

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 recommendations.

9.15. SUMMARY

Intel recommends that three critical parameters be met to ensure electrical testability of a component[2] removed from printed circuit boards: (1) The solder must be completely molten before removal is initiated; (2) the removal direction must be normal to the plane of the board; and (3) no movement, in-plane or tilting, of the removed component should occur until the solder completely resolidifies. Intel recommends specific temperatures and dwell time at temperature as a function of package type and size.

[2] This pertains 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. L. Clifton, C. Graf, I. Levy, E. Pope, K. Wieneke, "Plastic Surface Mount Components in Desiccant Pack for Surface Mount Applications," Intel Corporation, November 1989.
10. 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.
11. Intel Package Connections, Spring 1992.
12. P. Zarrow, "Optimizing IR Reflow", Electronic Packaging and Production, Nov. 1990.
13. C. Hutchins, "SMT/FPT Soldering Problems and Solutions", Surface Mount Technology, July 1990.
14. R. Bowlby, "Finish First", Circuits Manufacturing, Nov. 1987.
15. P. Augur, "SMT Challenges for Large IC's with Low Thermal Resistance", Intel Customer Q & R training presentation, April 1995.

9.17. REVISION SUMMARY

- Expanded moisture sensitivity precautions during SMT Rework.
- Added "Special SMT Challenges."