



# Reflow Process Guidelines for Surface Mount Assemblies

## REVISION HISTORY

<i>Revision</i>	<i>Date</i>	<i>Change Description</i>
PACKAGING-AN101-R	3/5/03	Changed peak reflow temperature.
PACKAGING-AN100-R	1/8/03	Initial release.

Broadcom Corporation  
P.O. Box 57013  
16215 Alton Parkway  
Irvine, CA 92619-7013  
© 2003 by Broadcom Corporation  
All rights reserved  
Printed in the U.S.A.

Broadcom® and the pulse logo are registered trademarks of Broadcom Corporation and/or its subsidiaries in the United States and certain other countries. All other trademarks are the property of their respective owners.

---

## TABLE OF CONTENTS

<b>Purpose</b> .....	1
<b>Scope</b> .....	1
<b>Reference Standards</b> .....	1
<b>Applicable Documents</b> .....	1
<b>Responsibilities</b> .....	1
<b>Procedure</b> .....	2
Introduction .....	2
General Surface Mount Process .....	2
Reflow Processes .....	3
Infrared Reflow .....	3
Forced Convection .....	3
Vapor Phase Reflow .....	3
Reflow Stages .....	3
Ramp-Up .....	3
Soak .....	3
Solder Reflow .....	4
Cool Down .....	4
Solder Profile Development .....	4
Eutectic Solder .....	4
Pb-Free Solder .....	6
Other Considerations .....	8
Moisture Sensitivity .....	8
Methods of Measuring Profiles .....	8
Maximum Package Case Peak Temperature .....	10
Component Rework .....	10
Summary of Reflow Recommendations .....	11
References .....	11





---

## LIST OF FIGURES

Figure 1: General Convection Reflow Profile for 63Sn/37Pb Solder ..... 4

Figure 2: General Convection Reflow Profile for Pb-free Solder Alloy ..... 6

Figure 3: Thermocouple Attachment Methods ..... 9



---

## LIST OF TABLES

Table 1: Eutectic Solder Reflow .....	5
Table 2: Pb-Free Solder Reflow .....	7
Table 3: Classification of Reflow Profiles .....	7
Table 4: Package Peak Reflow Temperatures .....	10





---

## PURPOSE

This document defines the guidelines for soldering surface mount components (SMCs) to printed circuit boards (PCBs) as a starting point for surface mount technology (SMT) process characterization. The data provided in this document is for informational purposes only and can be used as a general guide. It is the responsibility of the component users to ensure that all processes and parameters are properly characterized for their application.

---

## SCOPE

This procedure applies to all SMCs manufactured by Broadcom.

---

## REFERENCE STANDARDS

- J-STD-020B
- JESD22-A113-B (or the latest revision)

---

## APPLICABLE DOCUMENTS

- Assembly Instruction Procedure
- P-QAR-1007 (or the latest revision)

---

## RESPONSIBILITIES

Quality Assurance (QA) is responsible for ensuring compliance in the implementation of this procedure's requirements. Both QA and the packaging engineer are responsible for interpretation, maintenance, and revision of this procedure.

## PROCEDURE

### INTRODUCTION

Reflow is the process where solder is melted and reformed to make the electrical and mechanical connections between the components and the board. The development of an optimized reflow process is critical to maximize the assembly yield and reliability of the component-to-board solder joint interconnect. The solder joint quality is affected by several variables in reflowed board assemblies, including the following:

- Component
- SMT assembly conditions
- Solder wave conditions
- Solder paste
- Reflow profile
- Wave profile
- Board design

At the component level, the following can affect solderability:

- Solder-ball alloy
- Solder-ball pad plating
- Lead base metal
- Plating quality
- Lead coplanarity

There is no single reflow profile that is adequate for all board assemblies. This document is intended to provide some general guidelines in determining the proper reflow profile.

### GENERAL SURFACE MOUNT PROCESS

With a conventional surface mount process, solder paste is screen-printed onto the exposed terminal pads of the PCB. The component is placed onto the board so that the leads/balls of the component contact the solder paste deposits. Following the component placement, the assembly is reflowed to form the solder joints between the component and board.

---

## REFLOW PROCESSES

Within mainstream surface mount processes, the three commonly-used reflow techniques for soldering SMCs to a PCB are:

- Infrared (IR)
- Forced convection (FC)
- Vapor phase reflow (VPR)

### Infrared Reflow

The IR reflow technique involves transferring thermal energy supplied via IR lamps to the circuit board, solder, and component. Because this heating approach in its basic form is essentially a line-of-sight radiation surface-heating technique, the amount of thermal energy absorbed varies with board size, component size, component orientation, and materials used. The surface temperature of the board is typically not uniform throughout the soldering process, and board edges tend to run 10–20°C higher than the temperatures at the center of the board. Ensure that the temperature remains relatively uniform across the board, or component overheating or under-heating can occur.

### Forced Convection

The FC reflow technique involves transferring thermal energy supplied via heated air, either forced or blown on the circuit assembly from above and/or below. This method has some advantages over IR reflow processing:

- Board heating is more uniform compared with an IR process
- Component overheating and thermal shock are less likely, as the components tend to change temperature at a rate similar to that of the PCB
- Assembly size, component size, component orientation, and component material have less effect on the soldering process with FC as compared to the IR reflow process

### Vapor Phase Reflow

The VPR technique uses vapor from a boiling inert fluorocarbon liquid. The latent heat of condensation provides a thermal constraint dependent on the selected fluorocarbon liquid. A typical fluorocarbon liquid used in the industry has a boiling point of 215°C. The PCB temperature exposure should be uniform using this method. The temperature gradient across the surface of the board should be minimal. The component location design rules for even heating are not as sensitive as the rules for the IR reflow soldering process.

## REFLOW STAGES

This section describes each primary stage of the reflow process.

### Ramp-Up

The assembly experiences the initial heating in this stage and the organic solvents (that are present in the paste to help provide good paste-printing characteristics) begin to evaporate. To avoid thermal shock of the board and SMCs, the maximum heating rate (ramp rate) should be controlled and not excessive.

### Soak

In this stage, the fluxes present in the solder paste (that serve to increase the wetting of the solder to the terminal/lead surfaces) are chemically activated. This phase also serves to soak the assembly at a relatively constant temperature so that when reflow is initiated, the solder across the entire board reflows uniformly.

## Solder Reflow

In this stage, the solder particles in the paste and solder balls of the component are melted. The solder wets the surfaces to be joined. For leaded components, optimum fusing of the component leads with the solder paste on the board is achieved when the leads attain the melting temperature of the plated solder alloy. For ball grid array (BGA) packages, the optimum fusing of the component balls with the solder paste is achieved when solder balls attain the melting point of the ball material.

## Cool Down

In this final stage, the assembly is cooled in a controlled manner to solidify the solder into a strong metallurgical bond between the component and PCB leads/pads/terminals. The cooling rate must be controlled to form robust solder joints while not inducing any thermal shock to the PCB or SMCs.

## SOLDER PROFILE DEVELOPMENT

### Eutectic Solder

The previously described reflow stages (“Reflow Stages” on page 3) are shown in Figure 1 for eutectic 63Sn/37Pb solder. The following guidelines describe the typical parameters of a convection reflow profile. These are intended to be general guidelines. The exact profile is highly dependent on many factors, including the thermal mass of the assembly and the requirements/characteristics of the solder paste. Most solder manufacturers provide a recommended reflow profile based on solder composition and flux chemistry. The following figure is a typical reflow profile for eutectic solder.

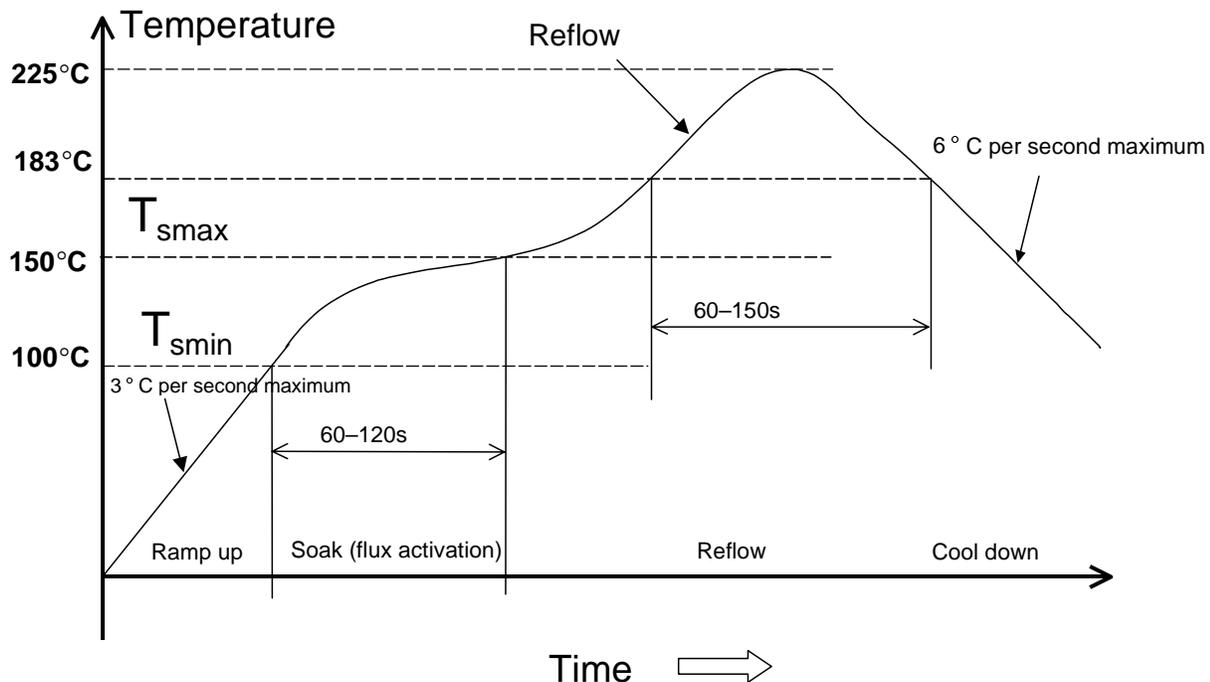


Figure 1: General Convection Reflow Profile for 63Sn/37Pb Solder

**Note** The profile shown in Figure 1 represents the solder joint temperatures.



Table 1: *Eutectic Solder Reflow*

<b>Reflow Stage</b>	<b>Description</b>
Ramp-up	To avoid thermal shock of sensitive components, it is preferable to hold the heating rate to less than 3°C per second.
Soak temperature	The temperature of the assembly in the soak stage should be in the range of 100–150°C, depending on the reflow method used (typically 130–150°C). Under-baking the solder paste flux can cause poor wetting, and also increase solder joint voiding. Over-baking the solder paste flux reduces the effectiveness of the flux and should be avoided. The exact temperature range is a function of the flux/paste requirements.
Soak duration	Typically, the amount of time in the soak stage is 60–120s.
Time above solder liquidus	The total time above the 63Sn/37Pb solder melting temperature (183°C) can range from 60–150s (typically 60–90 seconds), depending on the type of equipment used, board size, component size, assembly thermal mass, and so on. Extended duration above the solder melting point can damage the board and sensitive components, as well as cause the formation of excessive solder joint intermetallics that can reduce the strength and reliability of the solder joint.
Peak reflow temperature	The peak temperature of the solder joint during reflow should be high enough for adequate flux action/wetting and uniform solder reflow across the assembly. Typically, Broadcom recommends to keep the maximum temperature less than 225°C. Both package case and solder joint should not exceed the peak temperature. To avoid component or assembly damage, exposure time-to-peak temperatures (within 5°C to peak temperature) should be minimized (10–30 seconds).
Cooling rate	<p>The cooling rate of the solder joint after reflow is important to form robust solder joints. Faster cooling rates result in smaller solder grain size, which leads to higher fatigue resistance.</p> <p><b>IMPORTANT!</b> Due to differing rates of thermal expansion/contraction of the various materials in the assembly, it is critical to avoid excessive temperature gradients that can result in component damage.</p> <p>The cooling rate is typically less than 4°C per second, but can be as high as 6°C per second.</p>

Pb-Free Solder

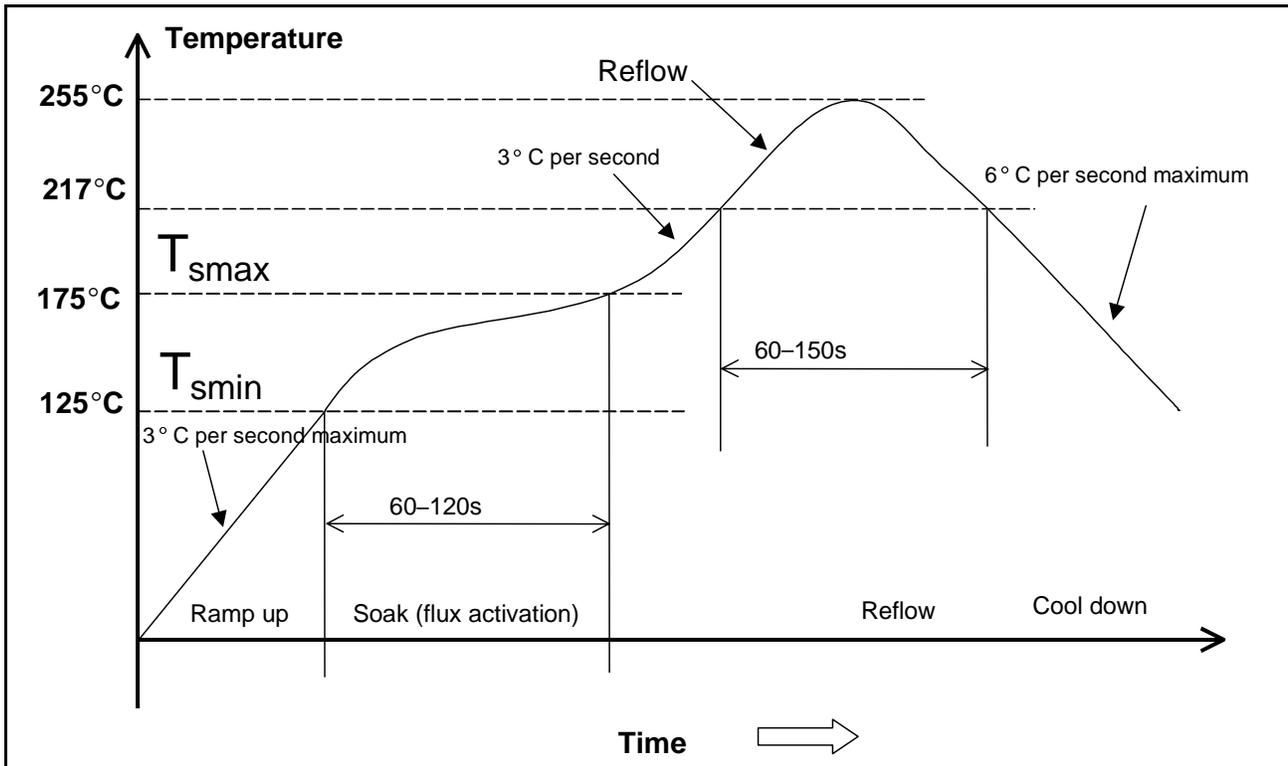


Figure 2: General Convection Reflow Profile for Pb-free Solder Alloy

**Note** The profile shown in Figure 2 represents the solder joint temperatures.



The profile requirements for Pb-free solder are similar to those described in “Eutectic Solder” on page 4, but Pb-free solder typically has a higher melting point that changes the profile. There are many different Pb-free solder compositions and fluxes/pastes. Each can have different requirements for the profile and melting temperature. Most solder manufacturers provide a recommended reflow profile based on solder composition and flux chemistry.

**Table 2: Pb-Free Solder Reflow**

<b>Reflow Stage</b>	<b>Description</b>
Ramp-up	To avoid thermal shock of sensitive components, it is preferable to hold the heating rate to less than 3°C per second.
Soak temperature	The temperature of the assembly in the soak stage should be in the range of 150–200°C, depending on the reflow method used. Under-baking the solder paste flux can cause poor wetting, and also increase solder joint voiding. Avoid over-baking the solder paste flux, as it reduces effectiveness. The exact temperature range is a function of the flux/paste requirements.
Soak duration	Typically, the amount of time in the soak stage is 60–120s.
Time above solder liquidus	The total time above the Pb-free solder melting temperature (217°C) can range from 60–150s (typically 60–90s), depending on the type of equipment used, board size, component size, assembly thermal mass, and so on. Extended duration above the solder melting point can damage the board and sensitive components, as well as cause the formation of excessive solder joint intermetallics that can reduce the strength and reliability of the solder joint.
Peak reflow temperature	The peak temperature of the solder joint during reflow should be high enough for adequate flux action/wetting and uniform solder reflow across the assembly. Typically, Broadcom recommends to keep the maximum temperature less than 255°C. To avoid component or assembly damage, exposure time-to-peak temperatures (within 5°C to peak temperature) should be minimized (10–30s).
Cooling rate	The cooling rate of the solder joint after reflow is important to form robust solder joints. Faster cooling rates result in smaller solder grain size, which leads to higher fatigue resistance.  <b>IMPORTANT!</b> Due to differing rates of thermal expansion/contraction of the various materials in the assembly, it is critical to avoid excessive temperature gradients that can result in component damage.  The cooling rate is typically less than 4°C per second, but can be as high as 6°C per second.

The following table summarizes the general solder reflow profile recommendations for both eutectic and Pb-free solders.

**Table 3: Classification of Reflow Profiles**

<b>Profile Feature</b>	<b>Sn-Pb Eutectic Assembly</b>	<b>Pb-Free Assembly</b>
Initial ramp-up rate	3°C per second maximum	3°C per second maximum
Preheat		
Temperature minimum ( $T_{smin}$ )	100°C	150°C
Temperature maximum ( $T_{smax}$ )	150°C	200°C
Time (minimum to maximum) ( $t_s$ )	60–120s	60–180s
Ramp-up rate $T_{smax}$ to $T_L$	3°C per second maximum	3°C per second maximum
Liquidus Temperature ( $T_L$ )	183°C	217°C (depends on Pb-free composition)
Time above $T_L$	60–150s	60–150s
Peak temperature	225°C maximum	255°C maximum
Time within 5°C of actual peak	10–30s	10–30s
Ramp-down rate	6°C per second maximum	6°C per second maximum



## OTHER CONSIDERATIONS

### Moisture Sensitivity

Most electronic components absorb moisture from the ambient environment. If an excessive amount of moisture is absorbed, the component can be damaged during the reflow process as the moisture vaporizes and induces high vapor pressure-related stress on the interfaces within the component. All electronic components are rated for their specific moisture sensitivity. This rating translates into the maximum time allowed between the point when the component is removed from its protective packaging (typically a vacuum-sealed bag), and reflowed.

**IMPORTANT!** It is critical that these guidelines be followed.

If the time is exceeded, the component manufacturer typically specifies the process needed to bring the component back into compliance. Typically, this is a baking step to remove the moisture from the component. The JEDEC standard specification J-STD0-20A describes the different moisture sensitivity ratings.

## METHODS OF MEASURING PROFILES

**IMPORTANT!** It is critical to confirm that the assembly is within the required profile specification.

To ensure that required temperatures are being achieved, characterize the reflow profile for each board assembly using thermocouples at multiple locations on and around the critical devices. Profile measurement is a vital part of setting up the solder conditions. The measurements are usually carried out using multiple thermocouples, which either:

- Have long connecting wires allowing the recording equipment to be kept outside the furnace
- Are attached to a temperature-resistant data recording device which travels through the reflow furnace with the PCB under evaluation

To accurately measure the preferred temperatures, ensure proper placement of thermocouples. Differences in lead/ball temperatures between components can occur due to differences in component structure and lead location (such as internal leads/balls of a ball grid array). Minimize these differences to achieve uniform solder joint formation. For convenience, tape or epoxy the thermocouple wires onto the surface to be measured.

---

**Note**

Broadcom recommends the drill hole/applying epoxy/cure method over the use of tape.



---

Because there can be significant temperature differences for many components between the leads/balls, package body, and different areas of the package, Broadcom recommends to use thermocouples to verify that the balls in the different regions of a BGA package reach acceptable temperatures during the soldering process. Since thermocouple access to balls under the component of a CSP or BGA is a challenging task, Broadcom recommends that a hole be drilled through the PCB at the ball of interest. The thermocouple can be routed through the hole from the bottom of the board and epoxied into place. This allows the thermocouple to sit flush with the board, and measure the temperature directly at the ball. This process can be repeated for various locations on the board to verify that all critical solder joints are seeing the proper temperatures and profile.

The following summarizes the Broadcom recommended thermocouple attachment method. A minimum of four locations are recommended across entire board:

- PCB
- Highest thermal mass package, such as EBGA, FCBGA, and so on
- Lowest thermal mass package
- Densely populated area

Thermocouple locations for high thermal mass packages such as TBGA, PBGA, EBGA, and so on. At least three thermocouples are required, two of which are:

- Solder joint (from the bottom of the PCB), center and corner
- Mold compound (from the top of the package)

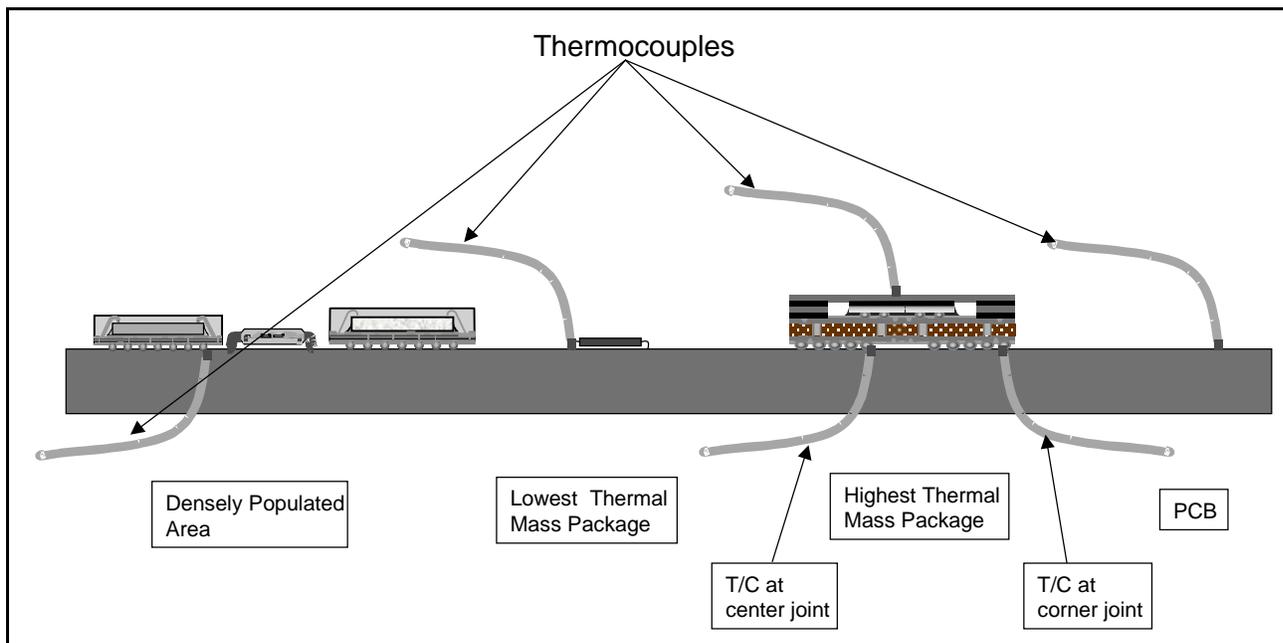


Figure 3: Thermocouple Attachment Methods

## MAXIMUM PACKAGE CASE PEAK TEMPERATURE

The temperature on the package top surface can be different from the temperature of the solder ball during the reflow. Excessive heat exposure during reflow can cause package overheat and create potential reliability issues. The following table lists the maximum package case temperature.

**Table 4: Package Peak Reflow Temperatures**

	<i>Eutectic Sn-Pb Components</i>	<i>Pb-Free Components</i>
Maximum Package Case Temperature (during reflow)	225°C	255°C

### Note



- In Table 4, the **Pb-Free Components** column refers to components that have been specifically designated by Broadcom as Pb-free or Pb-free compatible. These packages are designed to withstand the higher peak reflow temperatures associated with Pb-free solder. Components that have not been designated Pb-free or Pb-free compatible by Broadcom should not be run through a Pb-free reflow profile without prior review and approval by Broadcom.

## COMPONENT REWORK

Rework is the process where a defective component is removed (de-soldered) from the PCB, and replaced with a new device. In general, the same guidelines that were previously described for the standard surface mount process also apply to the rework process. Because rework is typically handled in a manual or semi-automated environment (versus fully-automated for the standard surface mount process), it is more difficult to provide the same degree of control and repeatability. The rework process also has additional constraints that adds to the difficulty and shrinks the process window. Rework varies significantly depending on the type/size of component being reworked, and the type of equipment being used. Because there is no single method for the rework process, Broadcom provides some best practice recommendations. Most of these recommendations are more critical for BGA components where access to the leads for de-soldering/soldering is limited.

- Before processing, bake all materials (components, boards, and so on) to remove latent moisture. Removing any moisture in the assembly reduces the chance of any damage to components during the high temperatures of the rework process.
- During the rework process, preheat the entire assembly to 100–125°C. This limits the amount of warpage of the board, as well as minimizes the degree of thermal shock during the reflow process.
- It is critical to uniformly heat the component during the reflow process. Uneven heating of the component can cause the solder joints to reflow at different times, and increase the probability for soldering defects or poor quality solder joints. While most heat is typically applied from above the component, applying some amount of heat from the backside of the board can reduce assembly warpage and improve the uniformity of temperatures across the component.
- To avoid unintended reflow of adjacent solder joints or adverse affects on adjacent component and/or solder joint quality, minimize excessive temperatures at adjacent components. Typically, focused nozzles or thermal shielding can be used to isolate the component of interest from the surrounding components.
- To ensure accurate component placement, use semi-automated or split-prism placement systems.
- Exact temperature profile requirements should depend on the type of paste/flux used and the manufacturer's recommendations.
- As with the standard surface mount process, perform temperature profiling using thermocouples to confirm that all temperature requirements at all critical times/locations are being the satisfied.

---

## SUMMARY OF REFLOW RECOMMENDATIONS

In general, the reflow process can have a wide process window provided certain critical factors are addressed:

- Follow temperature profile recommendations from solder paste/flux supplier
- Avoid excessive temperature ramp and cool rates
- Avoid excessive peak temperature exposure
- Avoid using surface mount process equipment outside of the operating conditions recommended by the equipment suppliers
- Use thermocouples and setup materials to confirm that all profile requirements are being satisfied

## REFERENCES

JEDEC Specification J-STD0-20B: Moisture/Reflow Sensitivity Classification for Nonhermetic Solder State Surface Mount Devices, August, 2002.

---

***Broadcom Corporation***

16215 Alton Parkway  
P.O. Box 57013  
Irvine, CA 92619-7013  
Phone: 949-450-8700  
Fax: 949-450-8710

Broadcom<sup>®</sup> Corporation reserves the right to make changes without further notice to any products or data herein to improve reliability, function, or design.  
Information furnished by Broadcom Corporation is believed to be accurate and reliable. However, Broadcom Corporation does not assume any liability arising out of the application or use of this information, nor the application or use of any product or circuit described herein, neither does it convey any license under its patent rights nor the rights of others.