CHAPTER 3 BOARD ASSEMBLY

INTRODUCTION

The electronics industry is continually challenged to meet customers' growing demand for smaller, high-performance semiconductor devices. The market demand for portable electronic products, in particular, is driving manufacturers toward increased miniaturization of components. Accommodating these smaller packages has necessitated improvements and changes to the methods by which components are assembled onto a Printed Circuit Board (PCB). This, along with increased efforts to eliminate ozone-depleting substances in all phases of manufacturing, have led to the following changes:

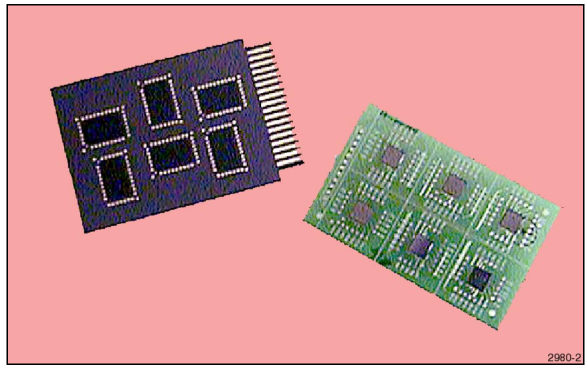


Figure 3.1 Packages are typically surface-mounted onto a printed circuit board using a solder reflow assembly method.

- Conventional vapor phase reflow assembly is being phased out because it requires the use of fluorinated solvents.
- Reflow systems are being designed to have forced convection, which improves the efficiency and uniformity of the heat transfer process.

AMD recognizes the impact that the board assembly process can have on the reliability of the components, especially the reflow temperature profile that is used (e.g., the peak temperature and temperature gradient). The qualification and reliability testing AMD conducts on its plastic Surface-Mount Devices (SMDs) includes simulating common board assembly techniques. This chapter provides an overview of the board assembly methods commonly used by AMD.

PCB DESIGNS

The primary functions of the PCB are to provide mechanical support for the components, provide the electrical interconnection paths, and help dissipate heat. The most common substrate design is a multilayer laminate, with four to six layers being prevalent for designs that accommodate SMDs. Designing more layers in the substrate reduces the circuit density, but it increases the thickness and cost of the PCB.

PCB Materials. In choosing the substrate material, designers must consider cost, the material's Coefficient of Thermal Expansion (CTE), the glass transition temperature, and the end-use application. (The glass transition (Tg) temperature is the temperature above which the physical structure of the laminate changes to a soft form and loses its mechanical strength. Tg is characteristic of polymer.) Common substrate materials are shown in the table below.

Substrate Materials

Substrate Resin	Coefficient of Thermal Expansion (ppm/°C)	Glass Transition Temperature (°C)
FR-4	80 to 90	130
Bismaleimide Triazine (BT)	41	180
Polyimide	50	270

BOARD ASSEMBLY METHODS

PCB assembly methods vary depending upon the type of components being mounted. For through-hole components, wave soldering is the popular method. For complex surface-mount devices, such as plastic leaded chip carriers and plastic quad flat packages, reflow soldering methods are used.

Wave Soldering. The wave soldering process begins with the component leads being inserted into the plated through holes on the board. As the assembly passes through the wave soldering system, all oxidation and contamination is cleaned from each hole barrel and component lead, and the underside of the board is exposed to a wave of molten solder (having a maximum temperature of 240°C to 260°C). The solder is flowed through each through hole by capillary action. As the lead tip touches the solder wave, the wetting force causes the solder to climb up the lead through the hole, spreading the solder to form the required solder fillet.

Reflow Soldering. The reflow methods currently in use are infrared and forced convection. These methods rely on the board assembly being heated to melt the solder paste on the land pattern so that it wets with the solder coating on the component leads. This allows the required solder joint to form, electrically and mechanically bonding the component to the surface of the PCB. The primary differences between these reflow methods are the source of the heat and the means by which the heat is transferred.

- ***Infrared -*** The InfraRed (IR) reflow method employs the thermal energy of halogen lamps radiating at a given wavelength, usually 1 to 5 mm. The light is condensed by reflecting mirrors which raise the temperature enough to reflow the solder paste. The board assembly passes through 4 to 20 zones, each of which are independently controlled for temperature. The zones are classified as preheat, preflow, and reflow. The IR method is attractive for high-volume, large board assembly operations because it allows the use of line-light and point-light sources.
- ***Convection Reflow -*** The forced convection reflow system is basically an improved modification of the infrared technique. Within the forced convection system, fans circulate hot air or inset gas. The heat transfer coefficient of forced convection is higher when compared to IR. Based on the uniformity and efficiency of heat transfer, convection heating is the optimum source for reflow when doing a mass reflow of the entire PCB assembly.

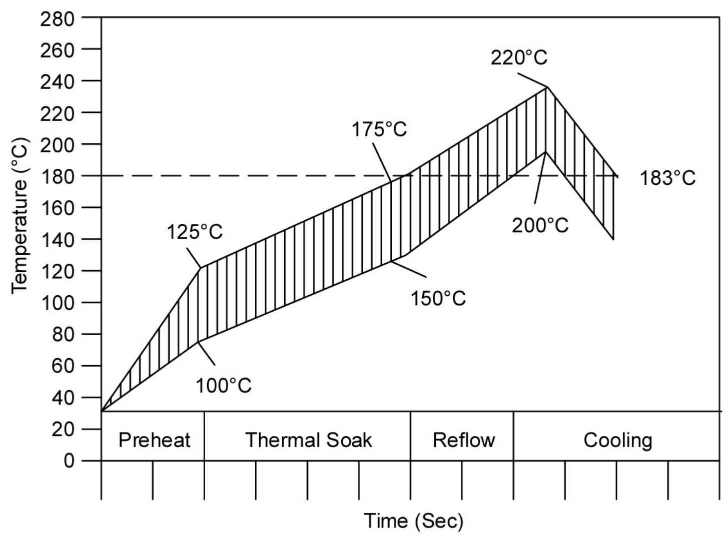
Reflow Profile Recommendation. *Figure 3.2 on page 3-4* shows a typical reflow profile of an IR/ convection oven for tin/lead solder. The band has a width of approximately 25°C over the entire range of heating and its upper and lower limits define the process window. Each temperature zone of the reflow profile is described below.

- ***Preheat -*** During preheat, the solvent will begin to evaporate and a 1° to 3°C/second rise rate should be maintained. If the rate exceeds this slope, thermal shock or cracking of component is risked.
- ***Thermal Soak -*** During thermal soak, there is typically a 60 to 120 second exposure to allow the paste to dry and the flux to activate. Too high or too low a temperature can lead to solder spattering or balling, as well as oxidation of the paste, pads, and the component terminations.
- ***Reflow -*** A critical parameter in the reflow temperature zone is the wetting time, the amount of time each solder joint is molten. The wetting time should be no more than 120 seconds, and the peak temperature of the SMD lead should be in the range of 220°C to 225°C. For information on specific packages, see your AMD sales representative.

- ***Cooling*** - A cooling rate less than 4°C/second is recommended to help achieve bright solder fillets with a good shape and low contact angle.

The total profile time varies by SMD package, board size, board density, throughput requirements, type of reflow equipment, and solder paste.

Profiling Method. Attaching a thermocouple to the assembly, using adhesives or Kapton tape, is the most common profiling method. When profiling for a small component, the thermocouple should be placed toward an edge or corner of the assembly. For a component of higher mass, however, the thermocouple should be placed toward the center of the board. (During the soldering process, the outside edges and corners of an assembly will heat up faster than the center. Also components of greater thermal mass will heat more slowly than components of lesser thermal mass.) Additional thermocouples should also be placed on other parts of the board or on components that are most subject to thermal shock or high temperature damage.



Reflow Profile (Temperature Zones)	Description of Characteristics	Process Windows
Preheat	Initial heating of the component leads or balls	1°C to 3°C/second 100°C to 125°C
Thermal Soak	Solder paste dries out and flux activates	100°C to 175°C Maximum 120 seconds
Reflow	Time above 183°C peak temperature	60 to 120 seconds
	Peak reflow temperature	220°C +5/-0°C maximum
Cooling	Cooling rate[1]	Maximum 6°C/second

Note:
1 The peak temperature applies to the temperature of the leads in the solder joints; not the temperature of the component package itself.

2981

Figure 3.2 Solder Reflow Temperature Profile

For Ball Grid Array (BGA) packages, AMD recommends that a thermocouple bead be soldered to a ball under the package and then placed in the center of the package. It is here where the package is likely to exhibit the minimum peak temperature and, therefore, is the worst case position for a cold solder joint.

For Ceramic Column Grid Array (CCGA) packages, AMD recommends special care be taken with handling and control measures. See your AMD sales representative for further details.

LAND PATTERNS

The solder land pattern is a critical design feature for a surface-mount component because it affects the strength of the solder joints, the ease with which the component can be tested and repaired, the type of solder defects that can occur (e.g., solder bridging), and the thoroughness with which the board assembly can be cleaned. Refer to ANSI/IPC-SM-782 specification for land pattern recommendations.