APPLICATION
NOTE

Mobile Pentium® II Processor Mini-Cartridge 240-Pin BGA Connector Assembly

Development Guide

April 1998 **Order Number: 243759-001**

Contents

List of Figures

PAGE

List of Tables

PAGE

1. INTRODUCTION

The ball grid array (BGA) package is becoming one of the industry's most popular packaging alternatives for high pin-count packages. One of its many advantages over other high pin-count packages is that the solder-ball interconnects self-align (within limits) during the reflow process, thereby reducing placement accuracy requirements during surface mount and improving board yields.

2. BGA CONNECTOR

The 240-position BGA board-to-board stacking connector consists of two halves:

- Plug
- Receptacle

The plug (see Figure 1) and the receptacle (see Figure 2) both utilize 0.76 mm diameter lead-eutecticting solder balls in a 1.27 mm pitch, 8×30 array (see Figure 3) for surface mount. The plug surface mounts to the processor and mates to the receptacle, which surface mounts to the OEM motherboard.

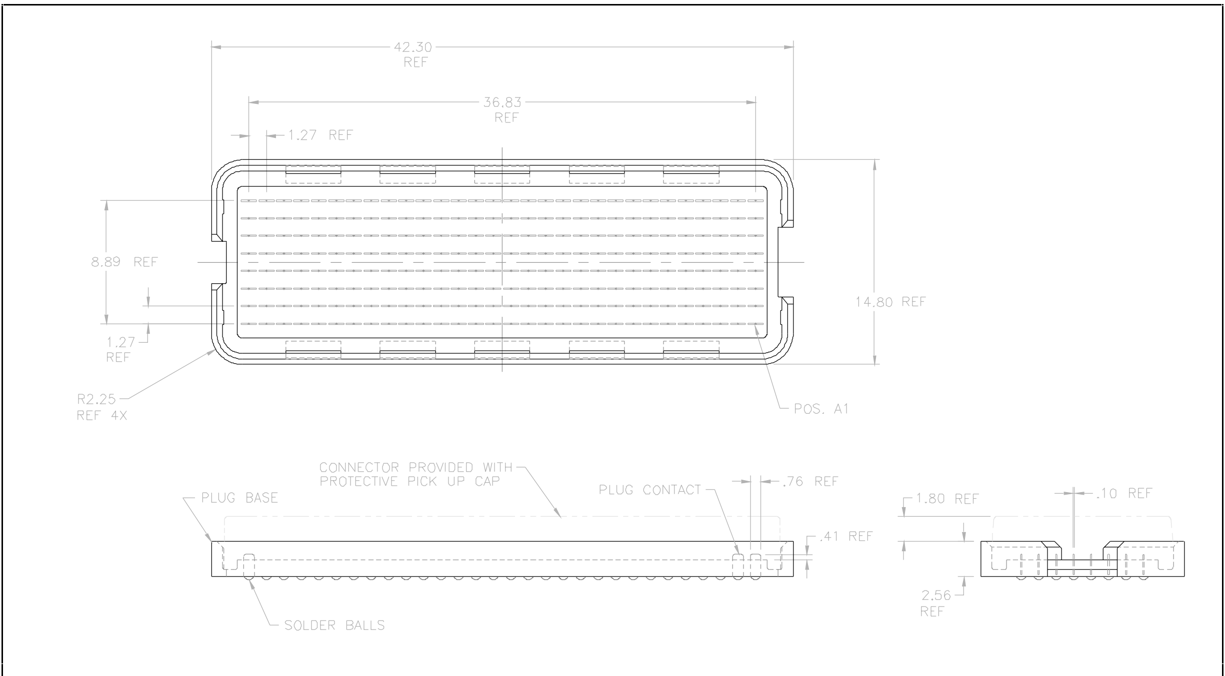

Figure 1. 240-Position BGA Connector (Plug Assembly) (Dimensions in mm)

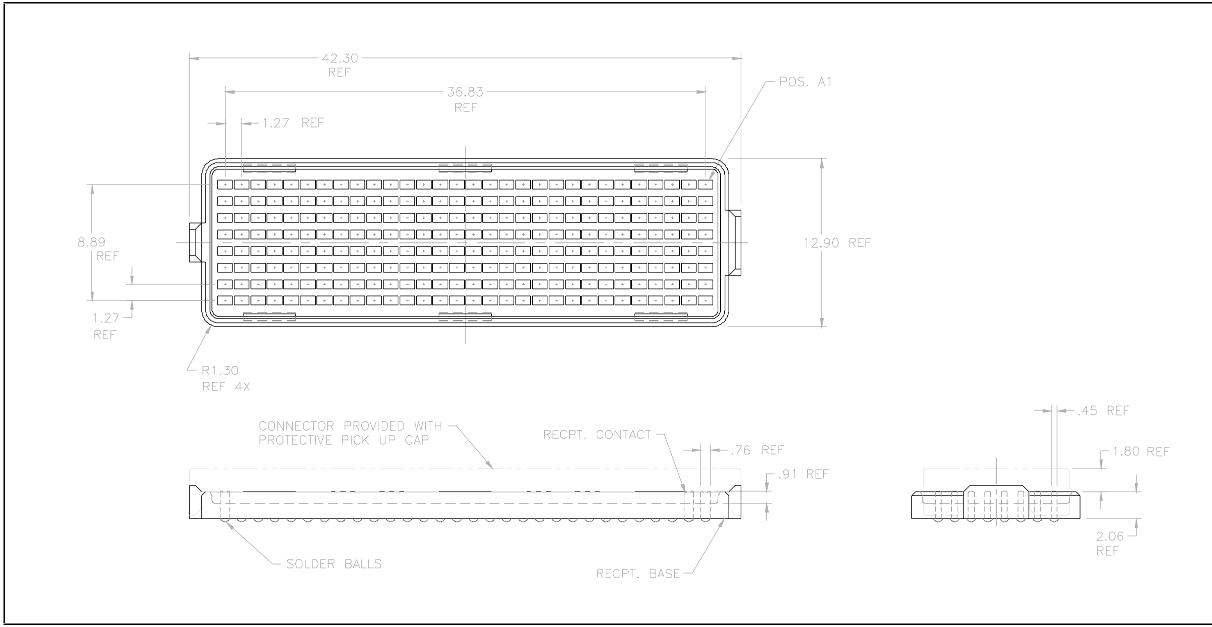

Figure 2. 240-Position Connector (Receptacle Assembly) (Dimensions in mm)

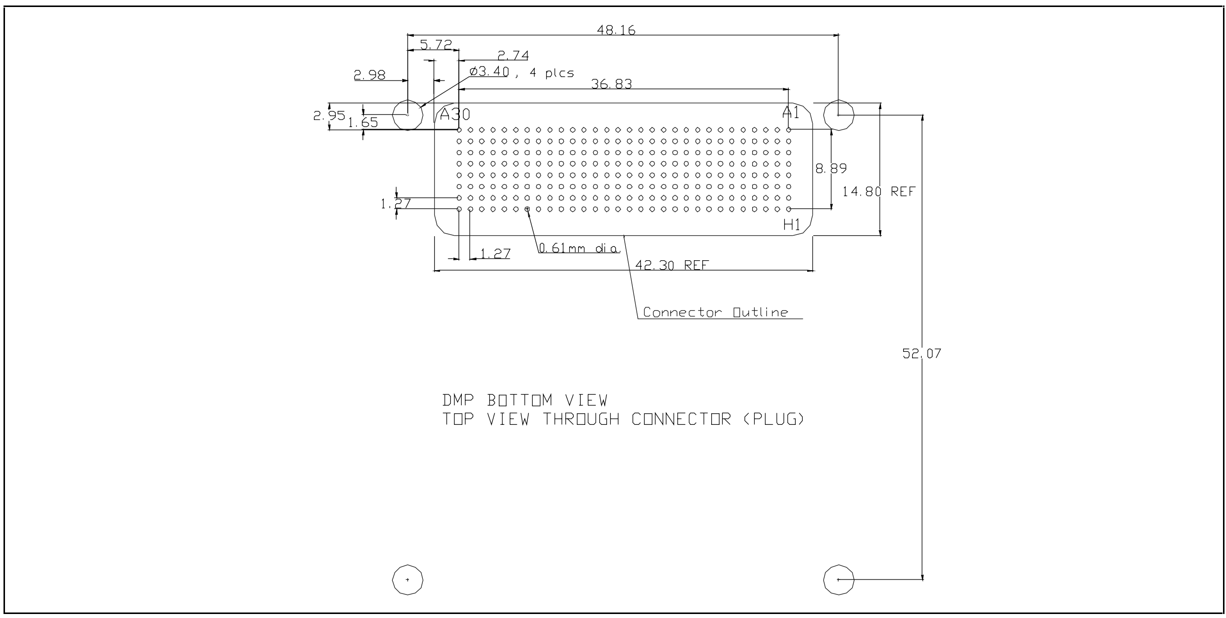

Figure 3. 240-Position Connector Footprint (Dimensions in mm)

3. COMPONENT PACKAGING

The 240-position BGA connector is available only in Tape and Reel in quantities of 300 pieces per reel. The tape is 56 mm wide on a 24 mm pitch (see Figure 4). The reel is 13 inches in diameter with a 150 mm hub diameter.

4. SOLDER STENCILS

The stencil thickness, as well as the etched pattern geometry, determines the precise volume of solder alloy deposited onto the device land pattern. Stencil alignment accuracy and consistent solder volume transfer is critical for uniform reflow-solder processing. Stencils are usually made of brass or stainless steel, with stainless steel being more durable. A stencil with a rounded corner, square hole with a five-degree tapered opening has been shown to be the best design to use for most BGA applications. Stencil thickness is typically between six and eight mils. The paste materials tend to dry out when proper environmental controls are not applied. A squeegee durometer of 95 or harder should be used. The blade angle and speed must be fine tuned to ensure even paste transfer. To achieve higher reflow yields, it is important to inspect the stenciled board before placing the parts.

5. SOLDER PASTE

Both no-clean and aqueous-clean fluxes may be used to mount the 240-position BGA connector (no-clean fluxes such as Kester R-244NC* are reported to have good performance). If the customer chooses to use aqueous-clean fluxes it is strongly advised that an ionic contamination study be performed using an appropriate SIR pattern (industry standard requirements for surface insulation resistance is 100 mega-ohms).

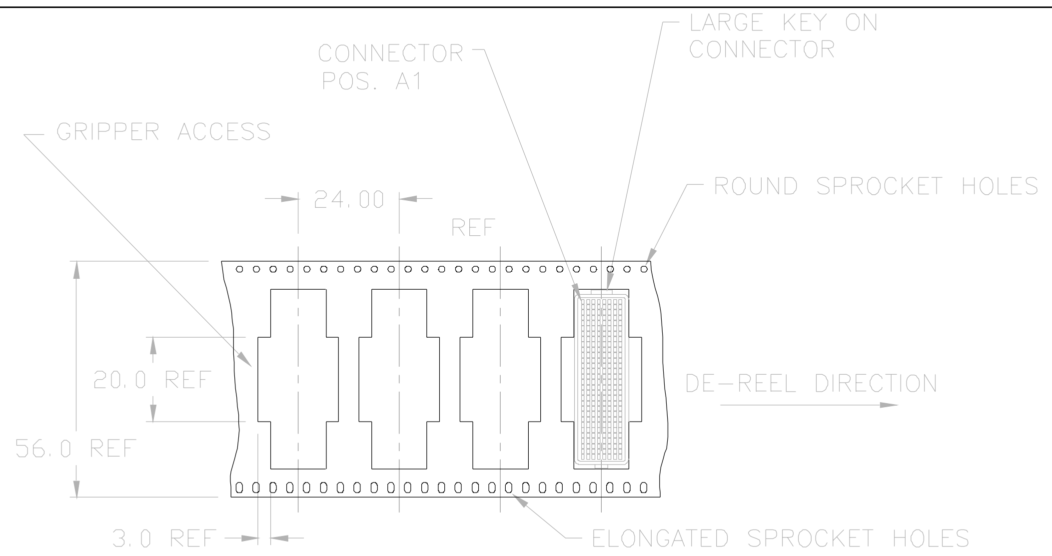

Figure 4. Tape and Reel Packaging (Dimensions in mm)

6. COMPONENT PLACEMENT

The 240-position BGA connectors have no special equipment placement requirements other than support for the previously mentioned tape and reel format. The customer has the option of using mechanical grippers or a vacuum for pick and place. Both sides of each pocket in the tape have a 20 mm × 3 mm cavity for mechanical gripper clearance during pick and place. The connectors can be supplied with a protective pick-up cap with a flat area for vacuum pick and place (see Figure 5). This cap can be easily removed after reflow.

7. COMPONENT ALIGNMENT

The pick-and-place accuracy dictates the package placement and rotational (theta) alignment, and is equipment/process dependent. Slightly misaligned parts (less than 50% off the pad) will self-align automatically during reflow. Grossly misaligned packages (greater than 50% off pad) should be removed prior to reflow as they may develop electrical shorts as a result of the solder bridges.

8. MOTHERBOARD LAYOUT (PADS AND VIA'S)

Intel currently recommends non-soldermask defined pads in sizes ranging from 23-25 mils and dog-bone connecting via's.

Non-soldermask defined pad sizes in the 23-25 mil range offer improved solder-joint reliability (as compared to soldermask defined pads) while also allowing two traces to be routed between them. For a 20 mil pad, routing can be accomplished using 6 mil traces and 6 mil spaces, while a 24 mil pad would require 5 mil traces and spaces.

For a possible reduction of the overall height of the receptacle the OEM may desire to use larger land pads on the motherboard. Data shows that there is no significant impact to the overall height by using a metal-defined land pad of 30 mils, rather than 22 to 26 mils. Also, reducing the amount of solder paste may result in more solder ball collapse.

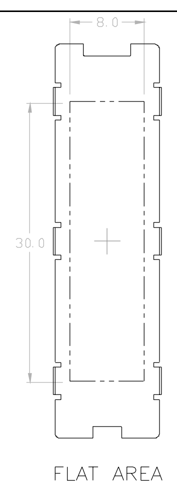

Figure 5. Vacuum Pick-Up Cap (Dimensions in mm)

12 mil plated via's on 24 mil pads also allow for two-trace routing. Intel recommends tenting the via's on the bottom side of the board to minimize heat transfer to the balls during reflow. Tenting via's on the top-side of the board to minimize the risk of solder-wicking is optional and left to the customer's discretion. It should be noted that tenting on both sides allows no relief for trapped gasses.

9. SOLDER-WAVE

Do not use solder wave to mount BGA's.

The ball temperature must be maintained below 160 °C in order to guarantee that no solderballs are accidentally reflowed during the solder-wave process.

10. REFLOW

Figure 6 shows a typical reflow profile from a Heller 1700 reflow oven, which is a 100% convection oven with independent upper and lower controls in seven zones.

The first ramp shows that the temperature increases less than 3 ºC per second before reaching the 130-160 ºC range. This reduces the amount of motherboard warpage which may cause inadequate contact between the solder ball and the solder paste. After the initial ramp, allow the board and components to reach thermal equilibrium (130-160 ºC) for approximately two minutes before approaching the 230 ºC peak temperature.

Finally, a ramp-down temperature of less than 3 ºC per second will again reduce motherboard warping during the period of time when the solder balls solidify.

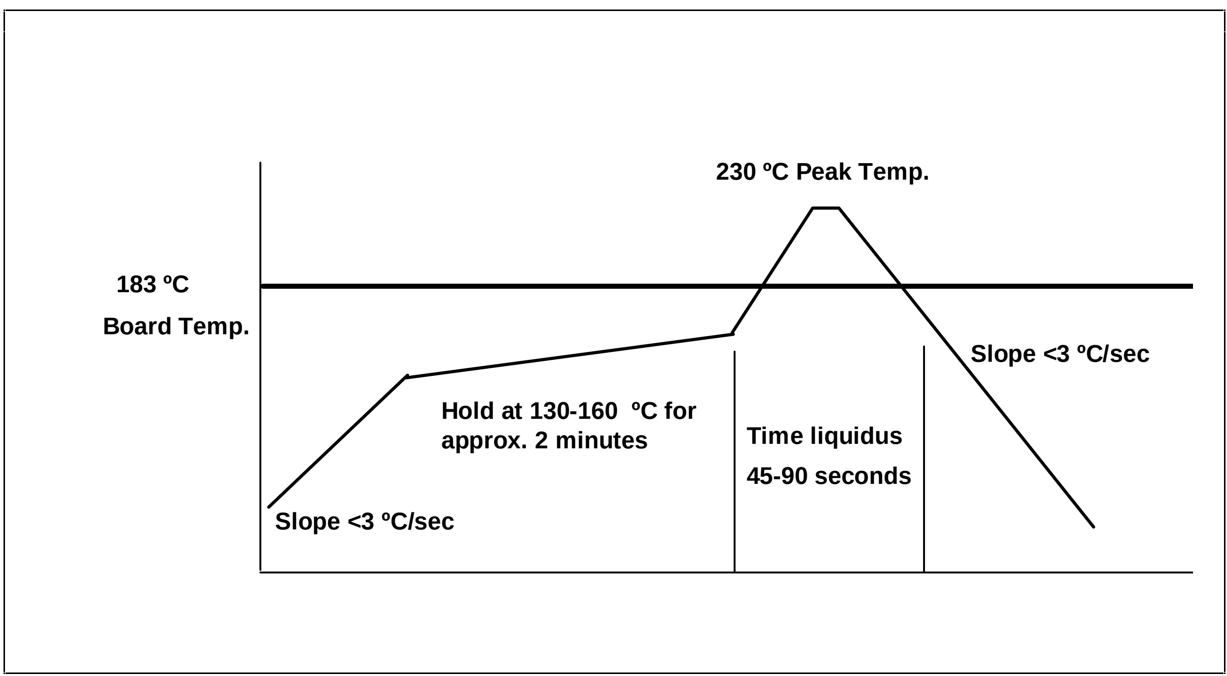

Figure 6. Typical Oven Reflow Profile (not to scale)

11. SMT PROCESS

Table 1 highlights several essential factors that contribute to a high-yielding BGA connector assembly process.

12. POST REFLOW INSPECTION

X-Ray is capable of consistently finding shorts between balls and voids, although it cannot detect cold-solder joints or opens.

In general, while good for process development and confidence building, X-ray is difficult to implement in high-volume manufacturing. Measurement of ball collapse has a shorter turn-around-time than X-ray, but is still primarily a development or sampling tool.

13. VOIDING

Voids in ball joints are induced by the SMT assembly process, primarily by trapped flux in the solder paste. The primary factors that increase the size and quantity of voids are the following:

- Increasing the reflow temperature profile
- Increasing the metal content of the solder paste
- Increasing the solvent volatility in the solder paste
- Decreasing the metal powder size in the solder paste

Table 1. Essentials for Quality Assembly

Essential Element	Recommendation
Solder Paste Quality	The solder paste should have uniform viscosity and texture and be free of foreign material.
Solder Paste Storage	The solder paste should be used before the expiration date. Ensure that the paste is maintained at a proper temperature during shipment and is protected from drying out on the solder stencil.
Motherboard	Use a clean, flat, well-plated solder ball land area and sufficient soldermask coverage.
Placement Accuracy	Tight placement tolerances are not usually required. The BGA can center itself as long as a major portion (more than 50%) of the solder ball contacts the fluxed solder paste land area on the board. Alignment marks (targets) on the PCB are helpful for placing parts.
Moisture Sensitivity Precautions	Know the moisture sensitivity rating of your components and adhere to IPC/JEDEC moisture control conditions to avoid package delamination or cracking.

14. REMOVAL AND REPLACEMENT PROCESS

This section provides a step-by-step example of how to remove and replace a BGA connector.

- Removing the BGA Connector from the Board

    1. Preheat the board to a minimum temperature of 80 ºC (max. temperature is 220 ºC). Any part of the board above 160-170 ºC approaches the 183 ºC solder melting temperature and risks damaging the joints of other components on the board. This is especially critical for the bottom side parts which are closer to the heat source. It is recommended that each customer conduct time and temperature experiments to determine the optimum conditions to minimize board/package warpage. Monitor both top and bottom-side board temperatures.

    2. Dispense a liquid, no-clean flux between the package and the board.

    3. Attach a vacuum pick-up tip onto the package and apply hot air (preheat, ramp time, and temperature to be determined by customer's own experimentation). Note that some problems with lifting the motherboard pad have been reported, possibly due to the machine type used and the amount of upward tension applied by the vacuum pick-up. This problem may be solved by first determining the typical release time when using the vacuum pick-up, adding 30 seconds, and then waiting this period of time before applying the vacuum pick-up for removal.

    4. Lift the package from the board.

    5. Carefully turn off the hot air and remove the board from the heat source, allowing it to cool to a safe, handling temperature. Inspect the board for any damage to the adjacent components or to the board itself.

- Inspecting, preparing and replacing the new package

    1. Remove any excess solder from the BGA solder pads using a solder wick or vacuum.

    2. Clean the BGA solder pads with isopropyl alcohol and a brush. Allow the board to dry, and then inspect it to ensure that it has a clean surface.

    3. Preheat the board to a minimum temperature of 80C (maximum temperature of 220 ºC). Any part of the board above the 160-170 ºC range approaches the 183 ºC solder melting temperature and risks damaging the joints of other components on the board, especially those on the bottom side which are closer to the heat source. It is advised that each customer conduct temperature experiments to determine optimum conditions to minimize board/package warpage. Monitor both top and bottom-side board temperatures.

    4. Stenciled solder paste is optional, but may later be required as BGA packages become larger. For applications where a solder stencil is not possible or not desired, acceptable results may be obtained by only applying flux to the pre-tinned pads. This is done by using a non-metallic spreader and applying a no-clean flux paste to the pads on the board. Care should be taken to prevent scratching the pads or the board.

    5. Apply liquid flux to the solder balls of the new package. After the liquid flux is applied, within two minutes, place the package on the board and then reflow (be sure to use the board's alignment features' fudicials and then mechanically or manually place the component).

6. Reflow the solder balls using hot air directed at the edge and under the package body. It is recommended that temperature experiments be conducted to determine optimum conditions.

7. Remove the board from the heat source and allow it to cool to a safe handling temperature.

8. Inspect the board and package to verify proper solder ball collapse and observe any defects that may have been caused by the rework procedure.

UNITED STATES, Intel Corporation
2200 Mission College Blvd., P.O. Box 58119, Santa Clara, CA 95052-8119
Tel: +1 408 765-8080

JAPAN, Intel Japan K.K.
5-6 Tokodai, Tsukuba-shi, Ibaraki-ken 300-26
Tel: + 81-29847-8522

FRANCE, Intel Corporation S.A.R.L.
1, Quai de Grenelle, 75015 Paris
Tel: +33 1-45717171

UNITED KINGDOM, Intel Corporation (U.K.) Ltd.
Pipers Way, Swindon, Wiltshire, England SN3 1RJ
Tel: +44 1-793-641440

GERMANY, Intel GmbH
Dornacher Strasse 1
85622 Feldkirchen/ Muenchen
Tel: +49 89/99143-0

HONG KONG, Intel Semiconductor Ltd.
32/F Two Pacific Place, 88 Queensway, Central
Tel: +852 2844-4555

CANADA, Intel Semiconductor of Canada, Ltd.
190 Attwell Drive, Suite 500
Rexdale, Ontario M9W 6H8
Tel: +416 675-2438