Ceramic Ball Grid Array Packaging, Assembly & Reliability
Outline for Discussion

• Why BGA????
• CBGA Introduction and Package Description
• PC Board Design for CBGA
• CBGA Assembly
• Rework
• Board-Level Solder Joint Reliability
Why BGA?????

• Advantages:
  – Higher Surface Mount Assembly Yield.
    » Coarse Pitches Compared to Fine Pitch Leaded.
    » Self-Centering.
    » Not Easily Susceptible to Handling Damage.
  – High I/O Density and High Pin Count Capability.
  – “Drop-In” Multi-Chip Capability.
  – Potentially Better Electrical and Thermal Performance Than Leaded Packages.
  – Compatibility with Most or All Existing Surface Mount Equipment.

• Limitations:
  – More Difficult to Inspect (But You Do Not Need To).
  – Any Defect Requires Removal and Replacement of Entire Package (No Touch-Up).
BGA Assembly Yields
(As Reported by IBM and Compaq)

DPPM_j*  

Year


IBM Data on CBGA
COMPAQ Data on PBGA (Collapsing Ball)

.5 mm Pitch QFP
.65 mm Pitch QFP
1.27 mm Pitch CBGA

< 0.5 (COMPAQ)
15 15 12

<5 2 5 1

85 80
20 30

85

200
100
50

* DPPM_j = Defective Joints Per Million Assembled
CBGA Construction

- **Alumina (Al₂O₃) Ceramic Substrate**
- **10Sn/90Pb Solder Balls**
- **97Pb/3Sn C4* Joint**
- **Silicon Die**
- **Epoxy Underfill**
- **0.89 mm Solder Ball for 1.27 mm Pitch**
- **63Sn/37Pb Solder Ball to Substrate Attach**

*C4 = Controlled Collapse Chip Connection (Evaporated 97Pb/3Sn Bump)*

Freescale Semiconductor
CBGA Construction (Cont.)

• 304 Pin C4*/CBGA

MPC106 PCI Bridge Chip

* C4 = IBM Developed Controlled Collapse Chip Connection

Freescale Semiconductor
CBGA Construction (Cont.)

• SEM Micrograph of Board-Mounted CBGA Substrate (No Chip) Showing

  - C4 Pads
  - Via
  - Power and/or Ground Planes
  - Trace
  - BGA Pad
CBGA Solder Balls

Typically 0.89 to 0.94 mm

0.89 mm Diameter 90% Pb and 10% Sn (by Weight) “High Melt” Solder Ball Held to CBGA Substrate and Motherboard with Eutectic (63Sn/37Pb) Solder.

Eutectic Solder Applied to the Motherboard Before Reflow. Must Have Minimum Eutectic Solder Paste Volume of 4,800 mils³.
### Products in CBGA

<table>
<thead>
<tr>
<th>Body Size</th>
<th>Pin Count</th>
<th>Pitch</th>
<th>Array</th>
<th>Comments/Products</th>
</tr>
</thead>
<tbody>
<tr>
<td>14 x 22</td>
<td>119 and 153</td>
<td>1.27</td>
<td>7 x 17, 9 x 17</td>
<td>Fast SRAMs</td>
</tr>
<tr>
<td>21</td>
<td>255</td>
<td>1.27</td>
<td>16 x 16</td>
<td>60X PowerPC™ products.</td>
</tr>
<tr>
<td>21 x 25</td>
<td>303</td>
<td>1.27</td>
<td>16 x 19</td>
<td>MPC105 and 106 PCI Bridge chips.</td>
</tr>
<tr>
<td>25</td>
<td>360</td>
<td>1.27</td>
<td>19 x 19</td>
<td>7XX and 7410 PowerPC™ products.</td>
</tr>
<tr>
<td>29</td>
<td>483</td>
<td>1.27</td>
<td>22 x 22</td>
<td>7450 PowerPC™.</td>
</tr>
<tr>
<td>32.5</td>
<td>503</td>
<td>1.27</td>
<td>25 x 25 (7 perim rows)</td>
<td>MPC107 PCI Bridge chip.</td>
</tr>
<tr>
<td>37.5</td>
<td>838</td>
<td>1.27</td>
<td>29 x 29</td>
<td>C-Port C-5 product. High CTE ceramic.</td>
</tr>
</tbody>
</table>

Freescale Semiconductor
Example CBGA
Case Outline Drawings: 119, 360 and 483 Pin CBGAs
119 Pin, 14x22 mm
CBGA Case Outline Drawing

Note: This is an uncontrolled copy
360 Pin, 25 mm
CBGA Case Outline Drawing

Note: This is an uncontrolled copy
483 Pin, 29 mm

CBGA Case Outline Drawing

Note: This is an uncontrolled copy

Freescale Semiconductor
Incoming PCB Requirements for CBGA

• Solder Pad Metal Finish Options
  – HAL or HASL (Hot Air Solder Leveled).
  – Entek Plus™ (from Enthone) or Other Organic Coatings.
  – Both Used Successfully with CBGA.

• Board Flatness
  – Planarity Within Industry (i.e., IPC) Standards Is Fine.
  – CBGA Typically More Forgiving Than Fine Pitch QFPs.

• Recommended Material Physical Properties
  – Glass Transition Temperature (Tg) ~115 °C.
  – Planar Coefficient of Thermal Expansion (CTE): Industry Typical 16~18 ppm/°C.
  – Lower CTE May Be Used to Increase Reliability.
  – Large Number of Vias within Area of BGA Increases CTE Locally.
Stencil Printing Solder Paste

- Nominal CBGA Solder Paste Volume Needs to Be 7,000 Mils\(^3\). Minimum Volume of 4,800 Mils\(^3\).
  - 7.5 Mil Stencil With 34 Mil Apertures Accomplishes This.

- Stencil Thickness Can Be From 5 to 14+ Mils Thick to Achieve This.

- May Want to Make Stencil Openings the Same Size As Pad Size (or Slightly Smaller).
  - Large Round Apertures Print Easily.

- Print Accuracy Needs to Be Only ± 5 Mils, With No Paste Touching Adjacent Exposed Metallization.
# CBGA Solder Paste Stencil Thickness Versus Aperture Diameter Table

(Use to Achieve Nominal 7,000 mils$^3$ or Minimum 4,800 mils$^3$ Solder Paste Volume)

<table>
<thead>
<tr>
<th>Stencil thickness (mils)</th>
<th>deposit base diameter (mils)</th>
<th>28</th>
<th>29</th>
<th>30</th>
<th>31</th>
<th>32</th>
<th>33</th>
<th>34</th>
<th>35</th>
<th>36</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td></td>
<td>2925</td>
<td>3137</td>
<td>3358</td>
<td>3585</td>
<td>3820</td>
<td>4063</td>
<td>4313</td>
<td>4570</td>
<td>4835</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>3510</td>
<td>3765</td>
<td>4029</td>
<td>4302</td>
<td>4584</td>
<td>4875</td>
<td>5175</td>
<td>5484</td>
<td>5802</td>
</tr>
<tr>
<td>7</td>
<td></td>
<td>4095</td>
<td>4392</td>
<td>4701</td>
<td>5019</td>
<td>5348</td>
<td>5688</td>
<td>6038</td>
<td>6398</td>
<td>6876</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td>4680</td>
<td>5020</td>
<td>5372</td>
<td>5736</td>
<td>6112</td>
<td>6500</td>
<td>6900</td>
<td>7312</td>
<td>6769</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td>5265</td>
<td>5647</td>
<td>6044</td>
<td>6453</td>
<td>6876</td>
<td>7313</td>
<td>7763</td>
<td>8226</td>
<td>8703</td>
</tr>
<tr>
<td>10</td>
<td></td>
<td>5850</td>
<td>6275</td>
<td>6715</td>
<td>7170</td>
<td>7640</td>
<td>8125</td>
<td>8625</td>
<td>9140</td>
<td>9670</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>6435</td>
<td>6902</td>
<td>7387</td>
<td>7887</td>
<td>8404</td>
<td>8938</td>
<td>9488</td>
<td>10054</td>
<td>10637</td>
</tr>
<tr>
<td>12</td>
<td></td>
<td>7020</td>
<td>7530</td>
<td>8058</td>
<td>8604</td>
<td>9168</td>
<td>9750</td>
<td>10350</td>
<td>10968</td>
<td>11604</td>
</tr>
<tr>
<td>13</td>
<td></td>
<td>7605</td>
<td>8157</td>
<td>8730</td>
<td>9321</td>
<td>9932</td>
<td>10563</td>
<td>11213</td>
<td>11882</td>
<td>12571</td>
</tr>
<tr>
<td>14</td>
<td></td>
<td>8190</td>
<td>8785</td>
<td>9401</td>
<td>10038</td>
<td>10697</td>
<td>11375</td>
<td>12075</td>
<td>12796</td>
<td>13538</td>
</tr>
</tbody>
</table>

*notes: 0.95 multiplier for typical non-cylindrical deposit shape minimum allowable paste volume for CBGA = 4800 cubic mils recommended target paste volume for CBGA = 7000 cubic mils non-tapered aperture opening*
CBGA NSMD Motherboard

Solder Pad Geometry

- Recommended Non-Soldermask Defined (NSMD) Pad Dimensions.
  - 0.72 mm solder pad diameter.
  - Minimum solder paste volume of 4,800 mils$^3$ (0.079 mm$^3$) per pad.
  - Surface finish may be any consistently solderable surface such as organic solderability protectant (OSP), HASL, electroless or electrolytic nickel/gold or immersion silver.

```
Solder Pad

Via

0.30 mm (12 mil) Finished Plated Through Hole (Adjustable)
0.65 mm (25 mil) Annular Pad (Adjustable)
0.30+ mm (12+ mil) Wide Line Between Pads

Liquid Photoimageable Solder Mask

0.72 ± 0.037 mm (28.5 ± 1.5 mil) Copper Pad Diameter
0.076 ± 0.025 mm (3 ± 1 mil) Clearance Between Copper Pad and Solder Mask (Adjustable)

Top View

Solder Mask Away From Copper Pad

Cross Section

PCB Laminate

Freescale Semiconductor
```
BGA Placement

• Most Newer Pick and Place Equipment Have Upward Looking Vision Systems That Can Recognize BGA.
  – Most Vendors of Older Equipment Have Retrofits Available.

• Alignment Can Be From All Balls, Package Body or Perimeter Row of Balls Only.

• No Vision Required for BGA With Mechanical Centering Device.
  – Place in Vibratory Nest, or on Precisor Stand, Then Pick From That “Known” Pick up Point.

• Loose Requirement for Placement Accuracy: Half the Diameter of the Pad, Compared to ±4 Mil Pitch QFPs.

• BGAs (Both PBGA and CBGA) Will Perfectly Self Center in Reflow.

Example of a Mechanical Precisor
Reflow Guidelines for BGA

- Follow Guidelines Recommended by Solder Paste Supplier.
  - Flux Requirements Must Be Met for Best Solderability.

- Recommended Profile: 210-215°C Peak, ≥75 Seconds Above 183° C (or Any Standard Surface Mount Reflow Profile).

- Temperature Profile Should Be Carefully Characterized to Ensure Uniform Temperature Across the Board and Package.
  - Solder Ball Voiding May Be Affected by Ramp Rates and Dwell Times Below and Above Liquidus.

- Nitrogen Atmosphere Not Required, but Will Make the Process Even More Robust, and Can Make a Big Difference in the Case of Marginally Solderable PC Board Pads.

- Full Convection Forced Air Furnaces Work Best, but IR, Convection/IR or Vapor Phase Can Be Used.
Profiling Furnace for BGA

• Profile Board Must Be Fully Populated to Simulate Thermal Load of the Production Board.

• Reflow Profile Board to Keep All Parts in Place.
  – Guess at a Profile. Start Low and Increase Temperature Until High Mass Parts Solder.

• Select Components to Monitor.
  – Highest Mass Part Near Center of PC Board.
  – Low Mass Part on One Corner of Leading Edge of Board.
  – Low Mass Part on Diagonal Trailing Corner.
  – CBGA, Whatever Location.
  – Any Other Critical or High Mass Part.
Before Soldering Parts on Profile Board:

- Drill Hole in PC Board (Just Large Enough for Thermocouple Wire) Through a Solder Pad in the Center of the CBGA Site.

Assemble All Components and Reflow by Normal Method.

Using Same Drill Bit, Drill Into Ball, About 20 Mils Deep.

- Use Small Amount of Thermally Conductive Epoxy to Fix Thermocouple Weld Into Center of Solder Ball, (the Exact Point of Critical Temperature).
Thermocouple Attachment Method for CBGA (Cont.)

- Thermocouple Attachment Inside CBGA Solder Ball
Secondary Reflow of BGA in Wave Soldering

- A Wave Solder Operation Is Often Required After Surface Mount Devices Have Been Soldered on the Board.
- The Partial Re-Reflow of BGA Solder Joints in This Secondary Operation Must Be Avoided.
  - The Result Can Be Open BGA Solder Joints.
    - PBGAs Are Low Thermal Mass Packages (They Can Heat Rapidly).
    - The Large Number of Plated Through Holes Transmit Heat Well.
Avoiding Secondary Reflow Of BGA in Wave Soldering

- Optimize Wave Solder Operation to Minimize BGA Temperature (As Measured Below the BGA).
- Utilize Wave Solder Pallets to Block Solder From BGA Area.
- Block Heat Transfer Through BGA Vias by Plugging Them or Tenting With Soldermask on Bottomside.
  - May Not Be Desirable From Test Standpoint.
    - Eliminates Direct Test Contact to Via Pads.
- Apply Polyimide Tape to BGA Area on the Bottomside.
  - Undesirable Hand Operation, but Can Be Used Temporarily.
- Redesign Board for Surface Mount Devices Only.
  - Eliminate Wave Solder Operation.
Cleaning After Reflow

- The Industry Is Moving to No-Clean Fluxes, but Due to Large Pitch and High Standoff, Any Conventional Cleaning Process Can Typically Be Used.
  - Ensure That Spray Nozzles Direct Flow at an Angle to Potentially Clean Under the Part.

BGA Inspection

- External Solder Joints Can Be Visually Inspected. If Uniform Reflow Temperature Is Maintained, and Outer Joints Look Good, the Internal Joints Will Be Good.
- Internal Solder Joints Can Be Inspected by X-Ray.
- The Entire Assembly Process Should Be Carefully Characterized; Then, Due to the Very High Yielding Nature of the Product, Solder Joint Inspection Can Be Eliminated.
BGA Assembly Yield - Top Ten Causes of BGA Assembly Defects

• In No Particular Order:
  – Popcorning of Plastic BGA Parts Due to Not Following Out of Dry-Pack Requirements. Results in Shorting Balls. This Is a PBGA Issue Only Since CBGA Is Level I.
  – Poor Incoming Board Quality (Unsolderable Pads Due to Oxidation, Soldermask or Other Contamination or Intermetallics).
  – Complete Lack of Flux on the Solder Pad (i.e., Solder Paste Skips) Insufficient Solder Paste Volume.
  – Part Misorientation (i.e., Rotated 90, 180 or 270°).
  – False Indictment of the BGA by the Test Program or Operator.
  – Operator “Tweaking” Before Reflow (Causes Solder Paste Smearing and Shorts).
  – Board Design Errors (i.e., Solderable Surface Under Soldermask or No Soldermask at All in the Area of the BGA Which Causes Solder to Wick Down Vias with Dog-Bone Pads).
  – Incorrect Reflow Furnace Profile (Usually Due to Improper Thermocouple Placement on Profile Board or No Profiling).
  – Partial Secondary Reflow During Wave Soldering.
  – Component Defects (Missing Balls, Unsolderable Balls, Coplanarity).
BGA Rework

• Due to Lack of Solder Joint Visibility BGAs Are Often Falsely Blamed for Board Electrical Problems.
  – In Study Done by Compaq Computers (Suzanne Fauser, et al) ¹ 75% of All BGAs Removed From Production Boards During the Period of Study Were Found Have Been Incorrectly Removed Due to Inability of Test Technicians to Identify the Real Cause.
  • Easy to Blame.
  • Hidden Solder Joints.
  • BGA Is Typically the Largest, Highest Pin Count Device.

• Special Care Must Be Taken to Develop Good in-Circuit Test Programs That Accurately Pin-Point the Failing Part.

¹ S. Fauser Et Al, Compaq Computer Corp., “High Pin Count PBGA Assembly: Solder Defect Failure Modes and Root Cause Analysis” Originally Published in the Proceedings of Surface Mount International
BGA Component Removal in Rework

- If the Board Is Thick With Internal Ground Planes:
  - Use Additional Localized Bottom Side Heat.
  - Preheat Entire Board to at Least 80°C (or Higher, As Permitted by Least Heat Tolerant Components Involved).
    -- Elevating the Temperature of the Entire Board Will Reduce Total Cycle Time and Prevent or Reduce Board Warpage.
- Avoid Collateral Moisture Damage (i.e., Popcorning) to Neighboring Moisture Sensitive Components. If It Is Unavoidable That They Go Over 150°C, Bake Board Overnight at Low Temperature (≥80°C) to Remove Moisture If Necessary.
- CBGA Typically Require Top and Bottomside Heating for Removal Due to High Thermal Mass of Ceramic.
- Apply Localized Heat to BGA Package.
  - When Solder Is Molten, Lift Package off Immediately With Vacuu
Example of BGA Rework Nozzle With Exhaust (One Type of Patented Air-Vac Nozzle Shown)

- Advantages:
  - Minimum Heating of Neighboring Parts.
  - Rapid Heating (Lower Rework Cycling Time).

- Potential Issues:
  - Gradients Across Part May Make One Side Reflow First (Part Tilt).
  - Narrow Process Window (i.e., Can Heat Extremely Rapidly).
Examples of Non-Shell Type BGA Versus QFP Rework Nozzles

QFP Nozzles
(Note Slots for Perimeter Heating Only)

BGA Nozzles
(Note Pipes and Shrouds for Directing Gas Under Package)

• Advantages:
  – Smaller Clearance Around Part Required.
  – One Nozzle Can Potentially Work for More Than One Body Size.
  – Wide Process Window.
Removing Excess Solder from Motherboard Solder Pads After BGA Removal

- Redress of the Site without Damaging Soldermask Required to Achieve Relatively Flat Pads.
  - CBGA Sites May Have Solder That Has a Higher Liquidus Due to Higher Lead Content as a Result of Dissolution from the Solder Balls.
- Avoid an Additional Heat Cycle by Removing Excess Solder From the Pads Immediately After Package Removal, While Board Is Hot.
- Skill and Experience of Rework Operator Should Be the Key Factor in Determining the Removal Technique.
  - A Hand-Held Solder Vacuum Tool Can Do an Excellent Job Quickly.
  - Solder Wicking Braid Requires Heat and Pressure, Usually Requires More Time and Is More Likely to Damage the Board or Pads.
Flux/Solder Paste Application in Rework

• CBGA Requires Exact Volume of Solder Paste (4,800 Mils³ Minimum for 1.27 mm Pitch) to Be Reapplied.

• Several Options for Reapplying Solder Paste.
  – Custom Mini-Stencil (Preferred).
  – Screen Solder Paste Directly Onto CBGA Component.
  – Dispense of Solder Paste in Known Volumes on Each Pad.
  – Use Solder Preform Disks That Match the Array.
Solder Paste Application in Rework

- Mini-Stencil Required to Apply Paste for CBGA/CCGA and Leaded Part Rework (Optional for PBGA).

(Photos Courtesy of OK Industries)

Freescale Semiconductor
Placing New Component on Site

• The Rework Station Should Have Good Look-Up/Look-Down Capability.
  – Either Video or Optical Vision.
  – Split Prism Optics to Simultaneously View the BGA Balls and the Footprint Is Helpful.

• Once Again, High Placement Accuracy Not Required Due to the Self-Aligning Nature of the BGA.
Rework Reflow

– Batch Oven Preheating (Typically 80°C) of Entire Board May Be Advisable, Especially With Large Multilayer Boards, to Prevent Board Warpage or Localized “Oil-Canning”.
  • Pre-Heat Temperature Cannot Exceed Rating of Least Heat Tolerant Part on the Board.

– Characterize Temperatures to Provide Uniform Heat and Ramp Rates Similar to Reflow Furnace.

Cleaning After Rework

– If Cleaning Is Required, Use Standard Production Cleaning Method or Use a No-Clean Flux for Rework (Even If a Non No-Clean Flux Used in Initial Assembly).
Summary of BGA Rework Process

• The Rework Process Goal Is Joint Structure and Quality at Least As Good As the Initial Assembly.
• Robust Initial Assembly Process Reduces the Need for Reworking BGA Packages.
• Removing and Replacing a BGA Can Be Easier Than for a QFP Due to Self Alignment.
  –However, Individual Solder Joints Cannot Be Repaired (i.e., No Touch-up).
• BGA Package Rework Can Be Accomplished with Off-the-Shelf Equipment.
• The Entire Rework Process Must Be Carefully Characterized.
CBGA Construction Influence on Attachment Reliability

- For CBGA, Ceramic Substrate Mismatched to FR4/Glass PCB.
- Causes Joints Farthest from the Center to Fail First (i.e., Greatest Distance from the Neutral Point or DNP)

FR4/Glass PCB (15-21 ppm/°C)

Ceramic Substrate (6.7 ppm/°C)

Increasing DNP

Freescale Semiconductor
Accelerated Thermal Cycling Conditions

- Accelerated Thermal Cycling (ATC) Performed on Mounted Daisy-Chain Packages to Obtain Failure Distributions.
  - Two Conditions Used Are 0 to 100°C (-48 cpd) & -40 to 125°C (24 cpd).
  - Testing Continues Until >50% Packages Exhibit at Least One Joint Fail.

Typical Profile for 0 to 100°C Testing

Typical Profile for -40 to 125°C Testing

Freescale Semiconductor
Motorola Board-Level Cycling
Typical Test Board Configuration

- Four Layer FR-4 Epoxy/Glass Test Board Used.
- Simulated Half-Ounce Power and Ground Planes Included.
- Nominal Solder Pad Diameters to Match the Device.
  - 0.0285” φ for 1.27 mm Pitch CBGA.
- 4.50” x 7.25” x 0.062” Thick Board.
- HASL and OSP (i.e., Bare Copper or Entek™) Solderable Surfaces.
- Copper Thieving Squares on Outer Layers.
28.5 mil board pad diameter. 7.5 mil thick stencil. 
25 mm CBGA Board-Level Thermal Cycling Data Plotted on a Log-Normal Plot (Both HASL and OSP Surface Finishes Tested)

25x25 mm sq. 1.2 mm Thick, 0 to 100 C, 2cph

Freescale Semiconductor
Comparison of Ceramic Substrate Technology
1.27 mm Pitch Parts, 2 Cycles per Hour TC

<table>
<thead>
<tr>
<th>Number of Temperature Cycles (0 to 100 C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
</tr>
<tr>
<td>2712</td>
</tr>
<tr>
<td>6328</td>
</tr>
</tbody>
</table>

Standard Ceramic Substrate - HTCC

High Temp. Coefficient of Expansion Ceramic Substrate - LTCC

Dotted Line Assumes 838 Pin Failure Today - No Failures As of 4700 Cycles.