APPLICATION NOTE

PRINTED CIRCUIT ASSEMBLY GUIDELINES

INTRODUCTION

Ball grid array packaging technology has been demonstrated to be compatible with industry standard printed circuit board assembly processes for over twenty years. However, this packaging technology has traditionally been used with high pin-count semiconductor products and implemented using standard plastic molding and wire bonding techniques common to many semiconductor packaging approaches. This packaging construction was commonly known as a plastic ball grid array (PBGA) because of the thermoset plastic used to encapsulate the semiconductor. The PBGA package typically incorporated eutectic (63/37 Tin/Lead) spheres as the device leads, and it found frequent application in commercial-grade standard reliability components.

An alternate construction used for more demanding applications utilized a ceramic (typically alumina) substrate onto which the semiconductor die was bonded. This became known as the ceramic ball-grid array package or CBGA. Because of the difference in thermal expansion coefficients between the ceramic substrate and the circuit board material, 10/90 Tin/Lead spheres were used for the device leads in order to provide a compliant interface and maximize the device reliability under thermal cycling conditions. A metallic cover was then attached to the substrate using a glass material that provided a hermetic seal. This construction found frequent use in the most demanding commercial and most military/aerospace applications where high reliability is of paramount concern. Because of the processing steps required to attach the die to the substrate and form the hermetic seal, this construction tended to be more expensive than the PBGA construction.

CTS' CBGA is illustrated in Figure 1. This method of packaging termination products revolutionizes the concept of digital signal line termination by providing a high-density array of resistors custom configured to terminate popular signal schemes and buss architectures.

Use of industry leading circuit deposition methods and equipment enables us to provide the highest packaging density and reliability while eliminating the expensive steps of die attachment and seal formation.

The construction of the ClearONE termination device utilizes a base of high-purity ceramic (Alumina) upon which the thick-film resistors and conductors are deposited. An overglaze of glass covering the resistors and circuitry provides a near-hermetic seal that minimizes moisture or chemical attack and therefore essentially eliminates moisture absorption. As such, prebaking of the ClearONE device is unnecessary prior to assembly of the printed circuit board, thereby eliminating a step in the board assembly process.

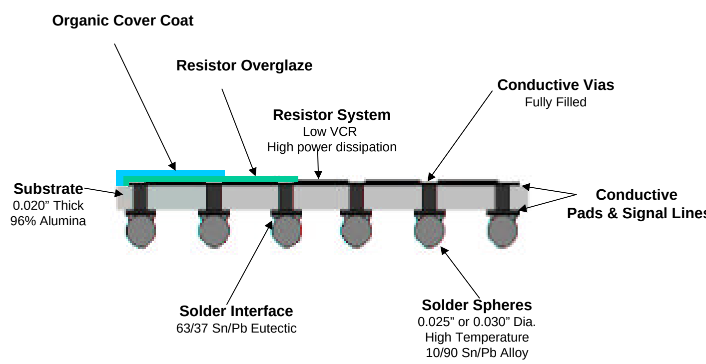

Figure 1 - ClearONE Construction

PCB LAND PAD DESIGN

- The shape of the CBGA land pads on the PCB should be round. The diameters of the PCB land pads should be equal to the nominal CBGA solder ball diameter as provided on the product data sheets, see Figure 2. Tolerance for the CBGA land pad diameter is ± 0.001 inches (± 0.025mm).

- The solder mask surrounding the land pad should not overlap the land pad to ensure maximum thermal cycling reliability, see Figure 2.

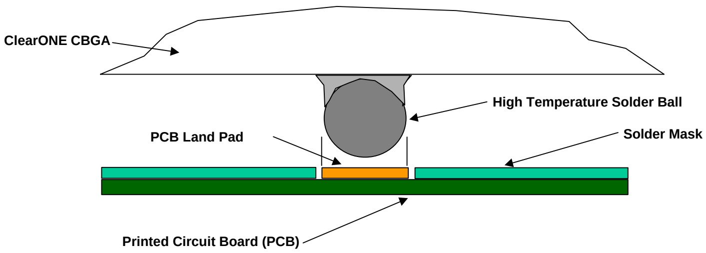

Figure 2 - CBGA Solder Ball and PCB Land Pads

- The electrical trace leading into the CBGA land pads requires a minimum width trace per the PCB board manufacturing technology used.
- CBGA land pads are not to be located on top of any vias.
- Vias connected to the CBGA land pads should have a minimum width connecting trace with a minimum length of 0.010 inches (0.254 mm), see Figure 3.

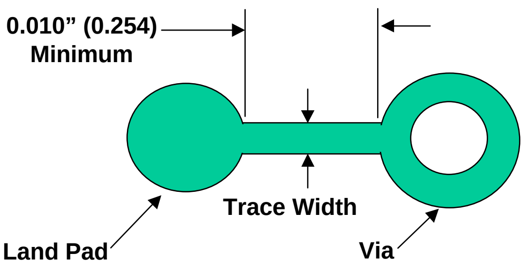

Figure 3 - Land Pad to Via

- CBGA land pads that are directly connected to a Vss or Vcc plane are to be connected with a maximum of two minimum-width traces of 0.010 inches (0.254 mm) minimum length. The two traces should be at least 90 degrees apart, see Figure 4.

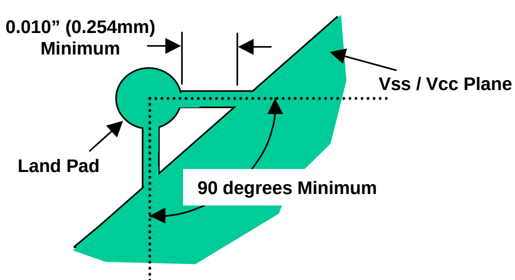

Figure 4 - Land Pad to Vss/Vcc Plane

PCB SOLDER MASK OPENING

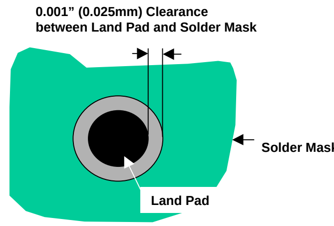

Figure 5 - Solder Mask to Land Pad Clearance

The PCB solder mask opening shall be larger than the PCB land pad by a minimum of 0.001 inches (0.025mm) per side as shown in Figure 5.

PCB MOUNTING PAD FINISH

PCB land pads are recommended to be solder-plated and reflowed with HASL finish.

CBGA PLACEMENT LOCATION TARGETS

Targets (fiducials) for pick and place equipment with pattern recognition systems should be included in the PCB layout. The in-house assembly group of the PCB assembly subcontractor should be consulted for exact requirements.

STENCIL DESIGN RULES

The recommended stencil thickness is 0.006 inches (0.15mm) with a tolerance of ± 0.0005 inches (± 0.001mm).

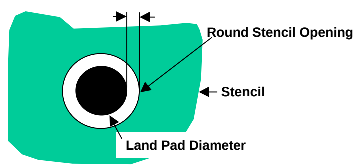

Figure 6a

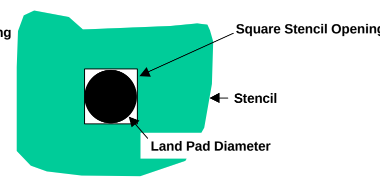

Figure 6b

The shape of the stencil opening may be round or square. For round stencil openings, the diameter should be equal to the PCB land pad diameter +0.001 inches (+0.025mm), as shown in Figure 6a. For square stencil openings, the sides should be equal to the PCB land pad diameter +0.001 inches (+0.025mm), as shown in Figure 6b.

The profile of the stencil opening should be tapered for optimal paste release as shown in Figure 6c, where the dimensional relationship is held **A**<**B** such that the taper is between 1 and 3 degrees.

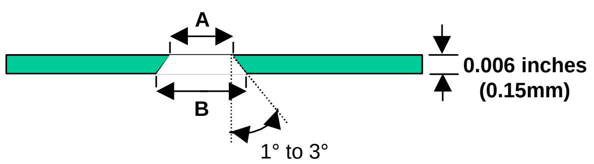

Figure 6c - Stencil Opening Profile Parameters

ARTWORK CRITICAL DIMENSION CHECK

During the data conversion from design software to PCB fabrication artwork, the critical dimensions may be changed by rounding-off errors in the software. The critical dimensions need to be verified on the artwork before PCB manufacture.

1) Land pad to land pad pitch
2) Land pad diameter
3) Solder mask openings
4) Alignment of solder mask opening with PCB land pad.

ESD PRECAUTIONS

This device is sensitive to ESD and as such standard industry practices should be observed when handling this component or assemblies incorporating this component, especially when the device or assembly has been removed from its protective packaging.

MOUNTING RECOMMENDATIONS

The assembly procedure for ClearONE CBGA packages is compatible with industry standard surface mount procedures, as exemplified in IPC-A-610A class 2 standards. Modern practices usually include a controlled environment with a temperature of 22°C ± 3°C, and a relative humidity of 50% ± 10%.

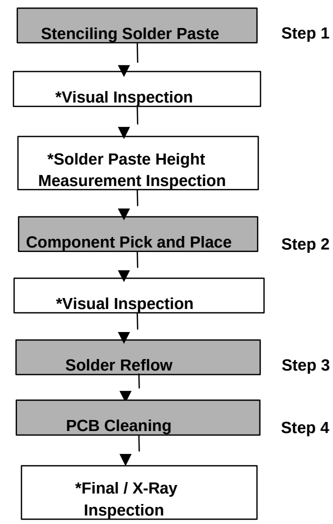

Figure 7 - Assembly Process Flow
(* Sample inspection frequency for inspection steps should be determined by customer's familiarity and experience with SMT assembly.)

STEP 1 – STENCILING SOLDER PASTE

Application of solder paste to the PCB is performed through a stencil with optical alignment capability and a metal squeegee. The squeegee speed and pressure should be adjusted per the solder paste manufacturer's recommendation. The PCB should be supported during the printing process to prevent bending the PCB. The stencil may require wiping between print passes in order to ensure proper print quality.

Verify the print quality before moving to the next step.

Printed solder quality is uniform within the CBGA pattern, pattern to pattern, as well as board to board per Figure 8a.

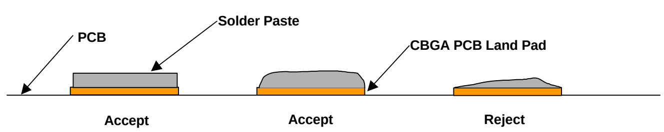

Figure 8a - Solder Paste Print Quality

Printed solder patterns should not run together, see Figure 8b

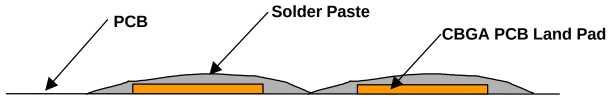

Figure 8b - Rejectable Shorted Solder Paste Deposition

Closely follow the paste manufacturer's guidelines for effective usage time, removing old paste and cleaning stencil thoroughly when paste has exceeded the recommended exposure limits.

The solder paste height should be measured as needed to confirm stencil thickness and tolerance of ± 0.0005 inches (± 0.001mm) per Figures 9a & 9b.

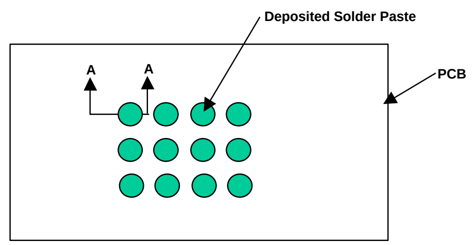

Figure 9a - Solder Paste Deposition (Top View)

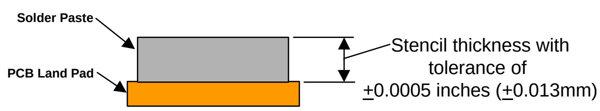

Figure 9b - Paste Height of Unreflowed Paste (Cross Section A-A)

- Any Type 3 SN63 Eutectic paste with good stencil life is recommended.
- Fresh solder paste should be used per manufacturer's recommendations.
- If cleaning the PCB assembly is appropriate based on the solder paste/flux used, the method and cleaning solution should follow the solder paste manufacturer's recommendations. The MSDS (Material Safety Data Sheet) should be consulted for details.
- The PCB should be transferred after solder paste printing to the pick and place machine within the time limit specified by the solder paste manufacturer.

STEP 2 – COMPONENT PICK AND PLACE

Placement of the CBGA on the PCB should be performed by a pick and place machine using automated optical alignment. Placement by hand is not recommended.

If the pick and place equipment uses the CBGA package perimeter for mechanical location instead of optical alignment, care should be exercised to ensure proper device alignment is achieved.

Verify CBGA polarity using the pin 1 indicator dot per Figure 10.

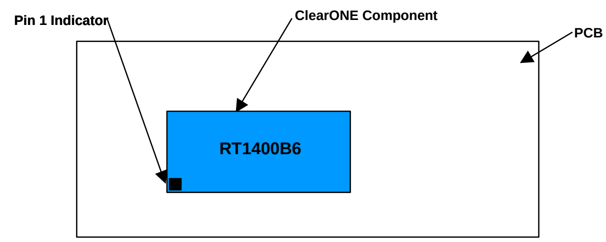

Figure 10 - Package Orientation with Pin 1 Indicator

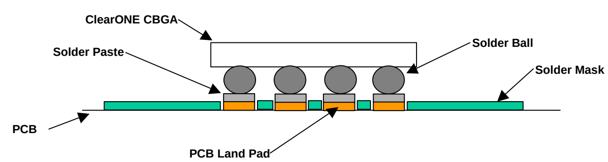

Figure 11 - Proper ClearONE to PCB Alignment

Verify assembly accuracy as needed. The side view of the intended alignment is shown in Figure 11.

STEP 3 – REFLOW

Place the assembled PCB onto the reflow oven conveyor belt, with an established reflow temperature profile. Good results are obtained with a profile shown in Figure 12 using the solder paste referenced. A list of viable parameters is given in Figure 13. If an alternate solder paste is used, consult the manufacturer for the proper reflow temperature profile ensuring that the device maximums listed in Figure 14 are not exceeded.

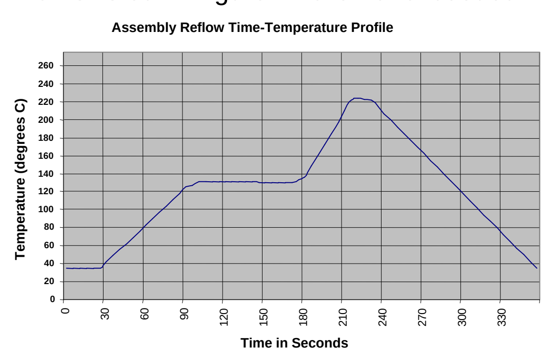

Figure 12 – Suggested PCB Assembly Time-Temperature Profile

	Ramp Rate	Temp Range	Duration
Initial Heating	(2°C–4°C)/sec	25°C– 125°C	30–60 sec
Soak Period	(0°C-0.5°C)/sec	125°C–150°C	100–120 sec
Ramp to Reflow Temperature	(1°C – 4°C)/sec	125°C– 200°C	20–60 sec
Time above Reflow	N/A	>183°C	30–90 sec
Maximum Temperature	N/A	240°C	10 sec Maximum
Cool-down	(2°C – 4°C)/sec	240°C– 25°C	50–200 sec

Figure 13 – Reflow Parameters for Typical SN63 Paste

Characteristic	Limit
Heating/Cooling Ramp Rate	4ºC/second Maximum
Peak Temperature	235 ºC Maximum
Time at 235ºC	20 seconds Maximum

Figure 14 – ClearONE Device Limits

STEP 4 – PCB CLEANING

Depending upon the type of flux system used during board assembly, cleaning of the PCB after reflow may be required. The solder-paste manufacturer should be consulted to determine the necessary cleaning process.

STEP 5 – FINAL/X-RAY INSPECTION

Verify assembly quality using available inspection machines, such as x-ray and side-looking optical microscopy. The side view of the final configuration is shown in Figure 15. There should be no shorts, no solder bridges between solder balls, and no loose solder balls under or around the CBGA. Inspection frequency should be based upon familiarity and experience with SMT processing.

PCBs with mounted CBGAs which have defects such as misalignment, shorts, or bridges should be reworked using recommended procedures. Please refer to www.ctscorp.com for this procedure.

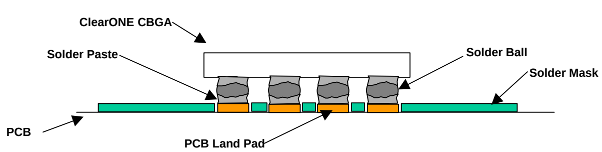

Figure 15 - Final CBGA Assembly

TYPICAL EQUIPMENT

Item	Equipment	Important Considerations
1	PC baking oven	For PCB drying (Moisture removal)
2	Screen printer with stencil	Equipped with optical alignment, stencil, and metal squeegee
3	Pick and place machine	Equipped with optical alignment accuracy of ±0.004 inches (±0.10mm)
4	Reflow oven	Forced air convection, 8 zone recommended
5	Wash system, if needed	Consult paste manufacturer for recommendations
6	Magnifying glass	X3
7	Paste height measuring system	Measure paste height to determine proper volume
8	Microscope	X40
9	X-ray inspection machine	Check alignment
10	Rework station	Computer temperature control and split-optics vision alignment system.

MATERIAL

Item	Material	Notes
1	Solder Paste	63% Sn/37% Pb or similar
2	Cleaning solution must be compatible with solder flux system	Review material safety data sheet and follow solder-paste supplier's recommendations.

SUMMARY

CBGA technology is uniquely designed for successful integration into industry standard SMT processes. Nonetheless, maximum quality and reliability of the final assembly can be assured only if assemblers take certain steps to prevent board related features from changing the shape of the liquid solder during attachment. These steps include adhering to strict definitions of mounting pad dimensions and tolerances, width of leads connected to pads, vias in the pads, and solder mask openings.

It is important to note that the mounting pad finish and stencil opening affect the quality of the reflowed solder bond.

SUMMARY (CON'T)

In the actual assembly process, control of solder paste dispensed volume and the quality of the solder paste are critical to achieving high manufacturing yields and acceptable assembly reliability. The smaller size of the device and leads makes the use of pick and place with automated optical alignment mandatory. The actual reflow procedure is controlled by a typical time-temperature profile. Recommended inspection procedures will verify the correct attachment and ideal shape of the solderball bond.

ADDITIONAL INFORMATION

Additional information on ClearONE CBGA products and technology can be obtained from the following reports, which are available from the CTS website at www.ctscorp.com.

1) Rework of ClearONE CBGA components – August 2001.
2) ClearONE Reliability Report – June 2001.
3) High Frequency Termination and the SCSI Buss
4) LVD SCSI Load-Humidity Test Report
5) ClearONE Product Selector Guide