Application Note
AN1005

RoHS Compliant ClearONE™ Printed Circuit Assembly Guidelines

Introduction

The standard ClearONE terminator is the industry leader in performance and reliability. The need to provide a lead free solution for resistor terminator networks in compliance with the RoHS initiative has required CTS to add a new material set to the ClearONE product line.

Many manufacturers have chosen to utilize Tin/Silver/Copper (Sn/Ag/Cu or SAC) spheres in their BGA products as a means of eliminating lead. The SAC spheres collapse into the interface solder during the reflow process, providing an unwanted and less than desirable variation in performance.

This new RoHS ClearONE material set takes advantage of the existing designs while eliminating lead-bearing solders. ClearONE solder spheres (10/90 Tin/Lead) and the eutectic interface solder (63/37 Tin/Lead) are replaced with Tin/Nickel-plated copper spheres and high temperature Tin/Silver/Copper (Sn/3.5Ag/0.5Cu) interface solder for the RoHS ClearONE terminators. As with the 10/90 spheres, the copper spheres do not collapse during solder reflow. By utilizing spheres that do not collapse, the RoHS ClearONE terminators maintain a predictable, consistent standoff height, and the performance established in the previous non-RoHS product.

The standard ClearONE Ball Grid Array (BGA) terminator material set is described in Figure 1. The ClearONE product designs provide identical performance in each channel through the use of symmetrical element layouts and by maintaining predictable standoff spacing.

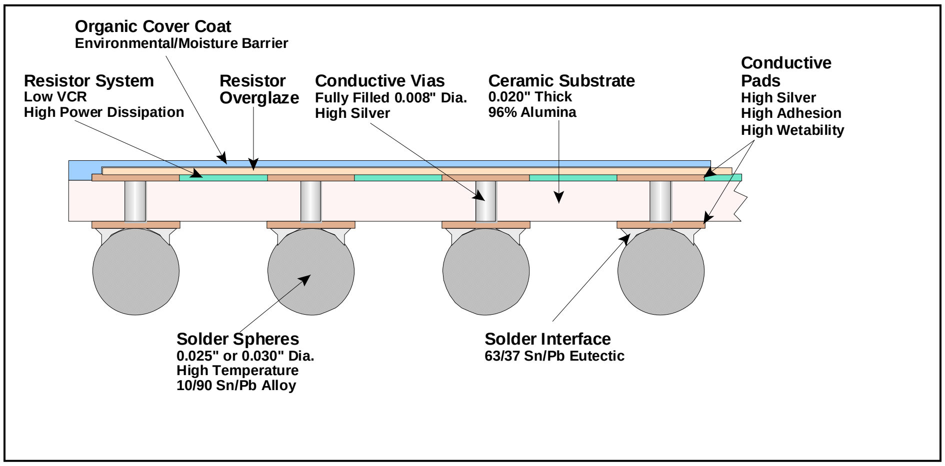


Figure 1. ClearONE terminator with lead bearing material set

The RoHS Compliant ClearONE terminators continue to take advantage of the symmetrical layout designs and predictable standoff spacing by utilizing copper spheres that maintain their shape, as shown in Figure 2.

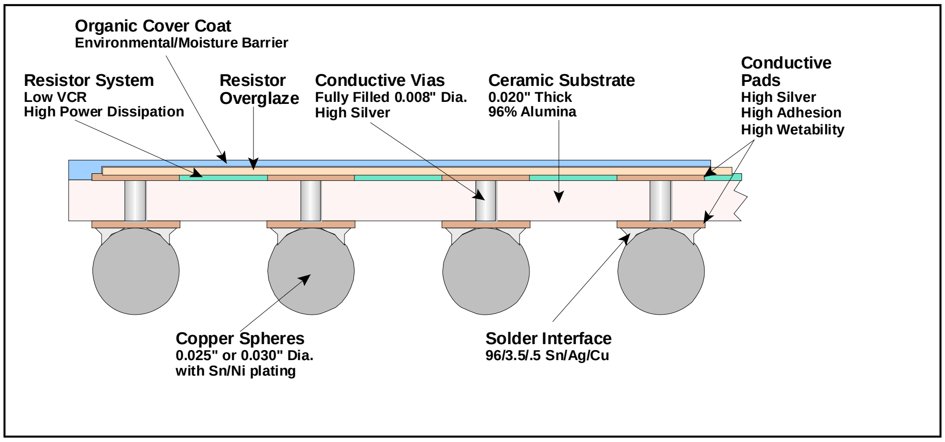


Figure 2. ClearONE Terminator with RoHS compliant material set

Description

PCB Land Pad Design

- The shape of the CBGA land pads on the PCB should be round. The diameters of the PCB land pads are recommended to be equal to the nominal CBGA solder ball diameter as provided on the product data sheets, see Figure 3. Tolerance for the CBGA land pad diameter is ± 0.001 inches (±0.025mm). The absolute minimum land pad diameter shall be 0.020” and 0.025” for 1.0mm and 1.27mm pitch parts respectively.

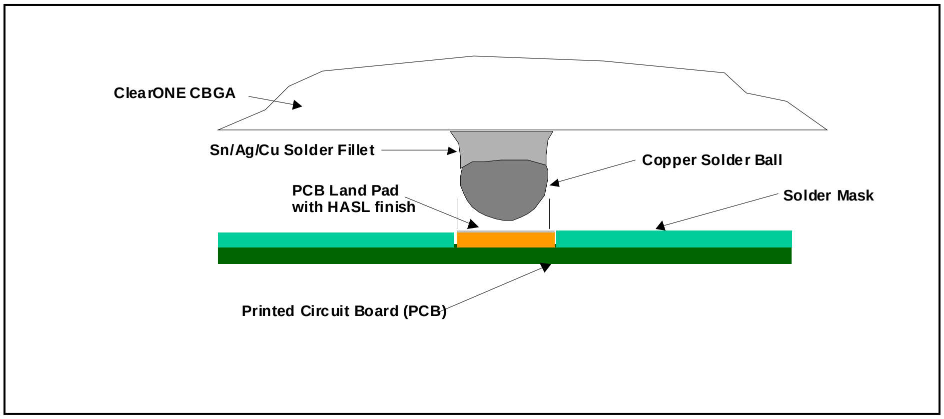


Figure 3. CBGA Solder Ball and PCB Land Pads are equal size

- The solder mask surrounding the land pad should not overlap the land pad to ensure maximum thermal cycling reliability, see Figure 3.

- 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 4.

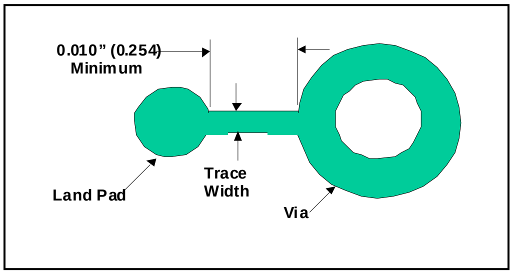


Figure 4. Land pad to via

- 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 5.

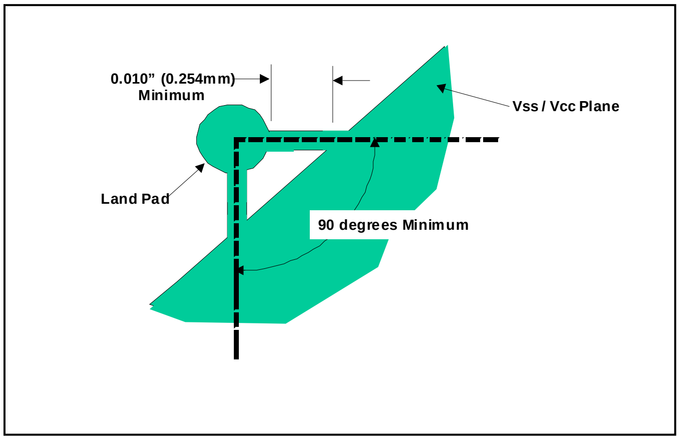


Figure 5. Land pad to Vss/ Vcc Plane

PCB Solder Mask Opening

The PCB solder mask opening shall be larger than the PCB land pad by 0.001 inches (0.025mm) per side as shown in Figure 6.

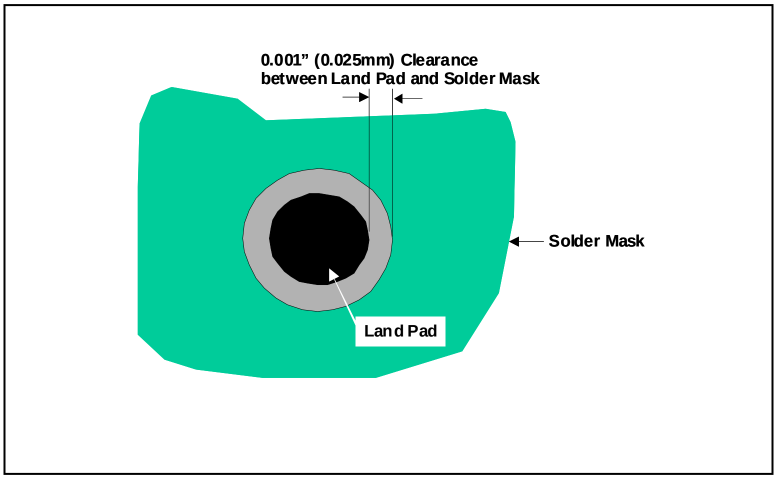


Figure 6. Solder Mask to Land Pad Clearance

PCB Mounting Pad Finish

PCB land pad finish will depend on the system requirements for the application. Since, each customer will have to take in to account a wide range of variables, CTS cannot make a single recommendation for all applications.

Cost and environmental concerns will need to be considered for each application and the manufacturer will need to thoroughly evaluate any selected board pad finish to insure solder joint quality is maintained. Available choices for lead-free pad finishes are: Electrolytic nickel/gold, electroless nickel/immersion gold (ENIG), hot air Pb-free solder leveling (HASL), immersion tin, immersion silver, and organic solderability preservative (OSP).

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

Stencil Thickness:

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

Stencil Opening Design:

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 7a. For square stencil openings, the sides should be equal to the PCB land pad diameter + 0.001 inches (+0.025mm), as shown in Figure 7b.

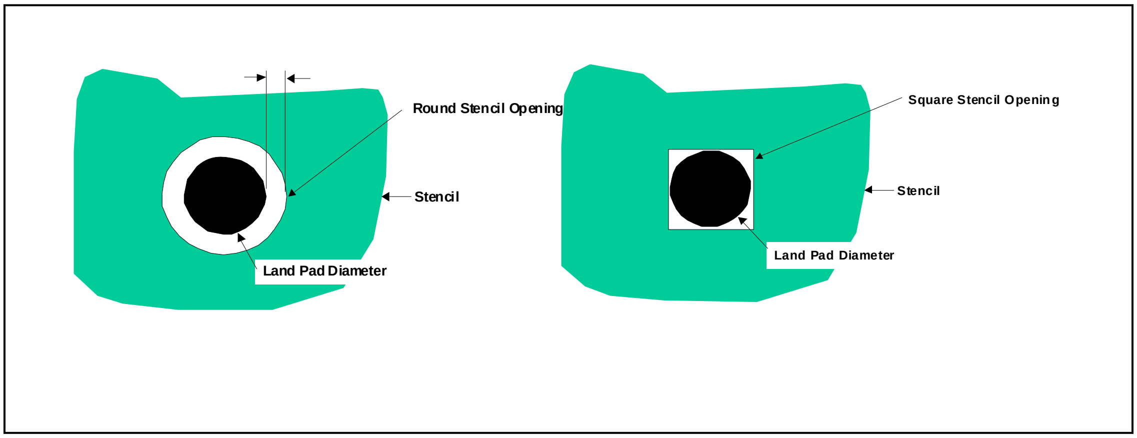


Figure 7a & 7b. Stencil Opening to CBGA Land Pad Diameter

The profile of the stencil opening should be tapered for optimal paste release as shown in Figure 7c, where the dimensional relationship is held **A**<**B** such that the taper is between 1 and 3 degrees.

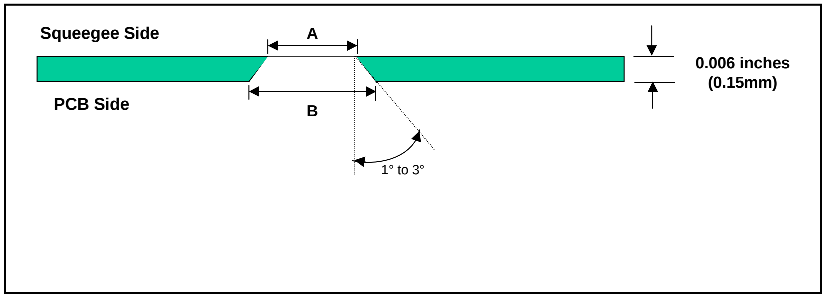


Figure 7c. Stencil Opening – Profile View

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.

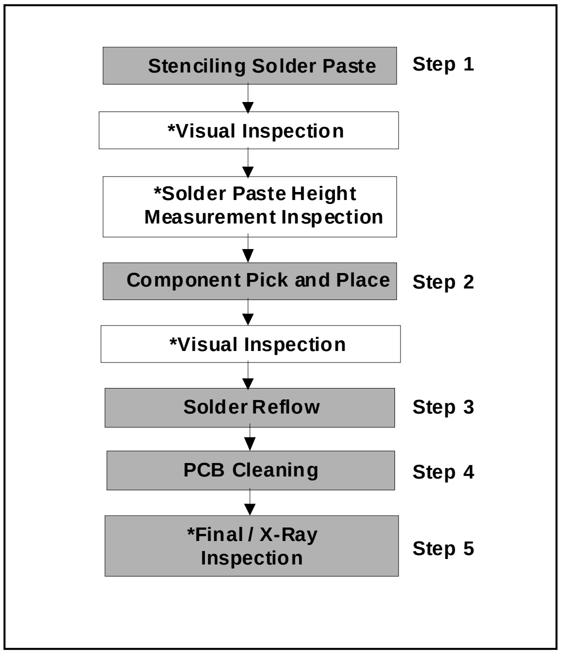
1) Land pad to land pad pitch
2) Land pad diameter
3) Solder mask openings
4) Alignment of solder mask opening with PCB land pad

RoHS ClearONE CBGA Mounting Recommendations

Typical Assembly Procedures

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

Typical Assembly Process Flow



*Sample inspection frequency for inspection steps should be determined by customer's familiarity and experience with SMT assembly.

Figure 8. Assembly Process Flow

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 9a.

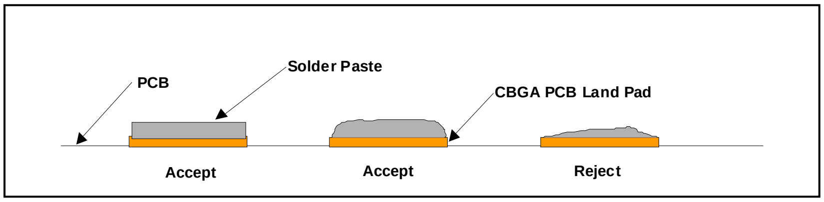


Figure 9a. Solder Paste Deposition

Printed solder patterns should not run together, see Figure 9b.

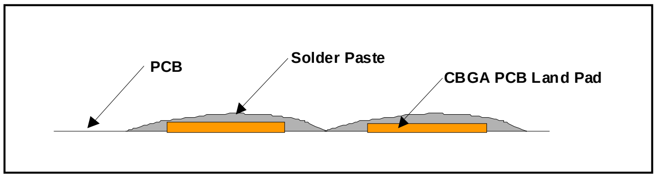


Figure 9b. Rejectable Shorted Solder Paste Deposition

If solder paste starts to dry out, remove and discard paste from stencil, clean stencil thoroughly, and then apply fresh solder paste to stencil.

The solder paste height should be measured as needed to confirm stencil thickness and tolerance of +0.0005 inches (+0.001mm) per Figure 10a/10b.

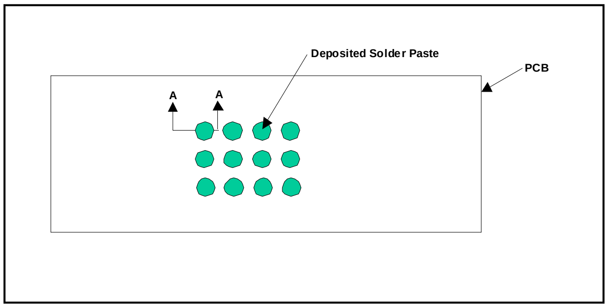


Figure 10a. Solder Paste Deposition (Top View)

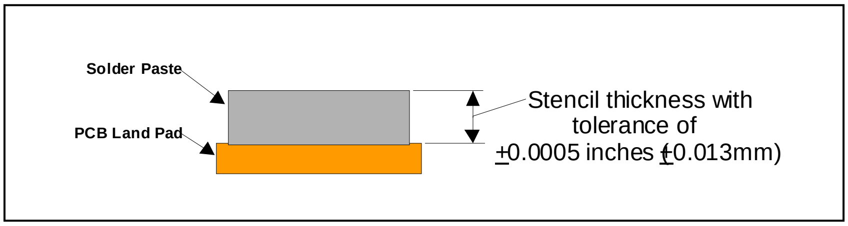


Figure 10b. Paste Height of un-reflowed paste (Cross Section View)

- Any SnAgCu 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 using 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 11.

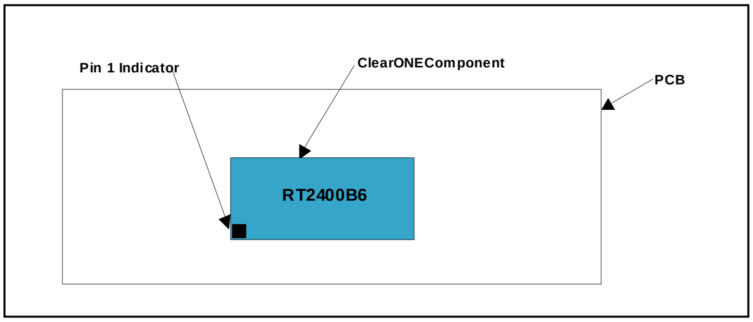


Figure 11. Package Orientation with Pin 1 Indicator

Verify assembly accuracy as needed. The side view of the intended alignment is shown below, in Figure 12.

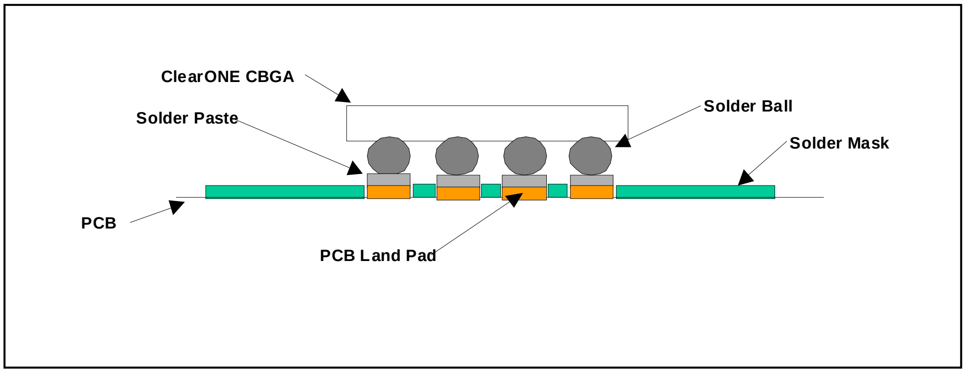
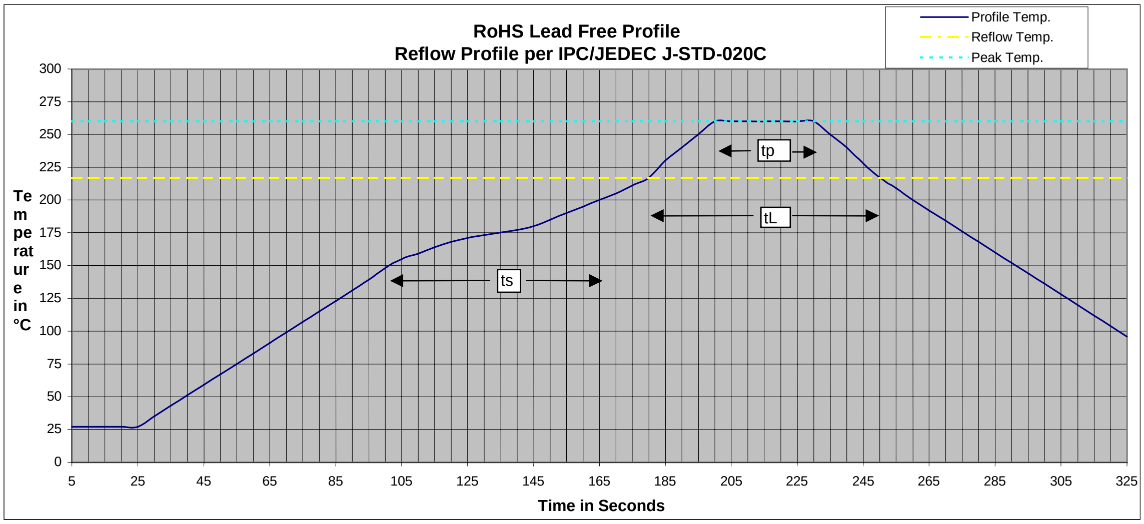


Figure 12. ClearONE CBGA Placement on Land Pad with Solder Paste (Side View)

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 13 using the solder paste referenced. A list of viable parameters is given in Figure 14. If an alternate solder paste is used, consult the manufacturer for the proper reflow temperature profile ensuring that the device maximums listed in Figure 15 are not exceeded.



	Time	Limits	Comments
tp	30	20-40 sec.	Time within 5°C of Actual Peak
tL	70	60-90 sec.	Time above 217 °C
ts	65	60-180 sec.	Time in Preheat between 150°C & 200°C

Figure 13. Suggested PCB Assembly Time-Temperature Profile for Solder Reflow

	Ramp Rate	Temperature Range	Duration
Pre-Heat Period t_S	(0°C – 1.0°C)/sec	150°C-200°C	60-180 sec
Ramp to Reflow Temperature	(1°C – 4°C)/sec	200°C-260°C	20-60 sec
Time above Reflow	N/A	>217°C	60-90 sec
Maximum Temperature	N/A	260°C	40 sec Maximum
Time within 5°C of Actual Peak	N/A	255°C-260°C	20-40 sec.
Cool-down	(2°C – 4°C)/sec	260°C-25°C	60-120 sec

Figure 14. Lead-Free Solder Reflow Temperature Profile Parameters

Parameter	NOT TO EXCEED
Heating/Cooling Ramp	4C/second Maximum
Peak Reflow Temperature	260°C for 40 seconds
Time at 260°C	40 seconds Maximum
25°C to Peak Temperature	8 Minutes Maximum

Figure 15. RoHS ClearONE Device Maximum Ratings

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 16. 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.

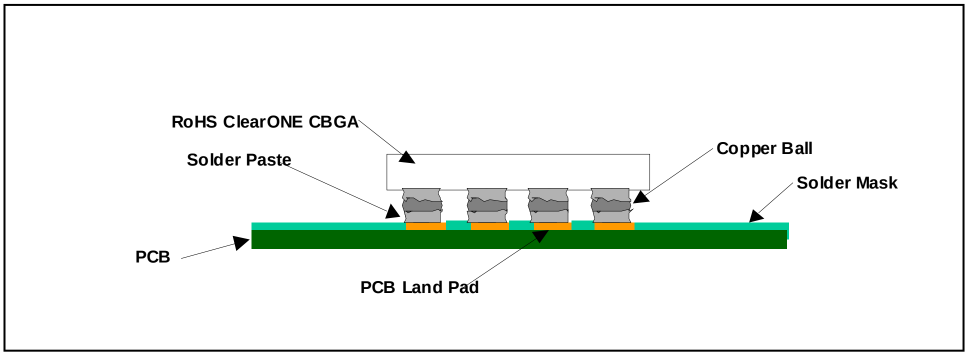


Figure 16. Final CBGA Assembly

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

Typical Equipment

Any industrial brand is recommended.

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	96.5/3.0/0.5 Sn/Ag/Cu 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.

In the actual assembly process, control of dispensing and quality of the solder paste become important. 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 solder-ball bond.