Application Report
SWPA156–February 2009

Part – II Assembly Guidelines for 0.4-mm Package-On-Package Packages

Steven Kummerl OMAP

ABSTRACT

After the bottom circuit board is designed, the assembly guidelines for Package-On-Package (PoP) versions of the OMAP3 processor and the memory device must be considered.

PoP package assembly involves multiple variables, including the following factors, which have a major effect on the quality and reliability of final assembly:

- Circuit board design
- Solder paste characteristics
- Solder paste deposition
- Reflow profile
- Fluxing or solder paste deposition onto the PoP memory before assembly

There are two major techniques for mounting the PoP memory: one-pass and two-pass. Although this document describes both techniques , TI recommends the one-pass process because it is more economical and provides about the same assembly yields as the two-pass technique.

The BeagleBoard, discussed in the companion document, *Part-I, 0.4mm PCB Design Guidelines*, provides a basis for understanding 0.4-mm assembly guidelines and process parameters. This document, which is Part II, describes trial runs performed with daisy chain units. However, this is only one of many possible variations of the assembly process. To optimize PoP assembly, TI strongly recommends performing trial runs and studies with the assembly house.

Figure 1 shows POP after reflow.

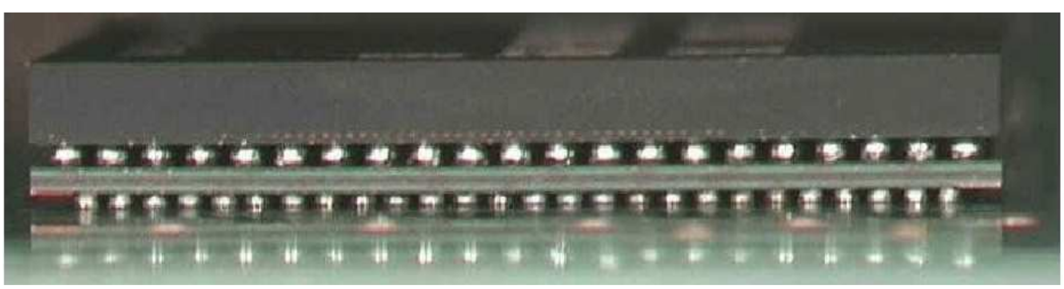


Figure 1. POP After Reflow

Contents

List of Figures

List of Tables

1 Introduction

This document is the second of two parts. Part I, *Part-I, 0.4mm PCB Design Guidelines*, describes general PCB design guidelines. The design of the bottom board to which the OMAP processor is mounted dominates the PoP failure mechanisms. Be sure to read and apply the guidelines in Part I before beginning PoP assembly.

This document, which is Part II, describes the guidelines for assembly of printed circuit boards that use the PoP package. Included are assembly options and suggestions for qualifying and working with assembly sites. Figure 2 is an overview of the OMAP3 PoP assembly guidelines.

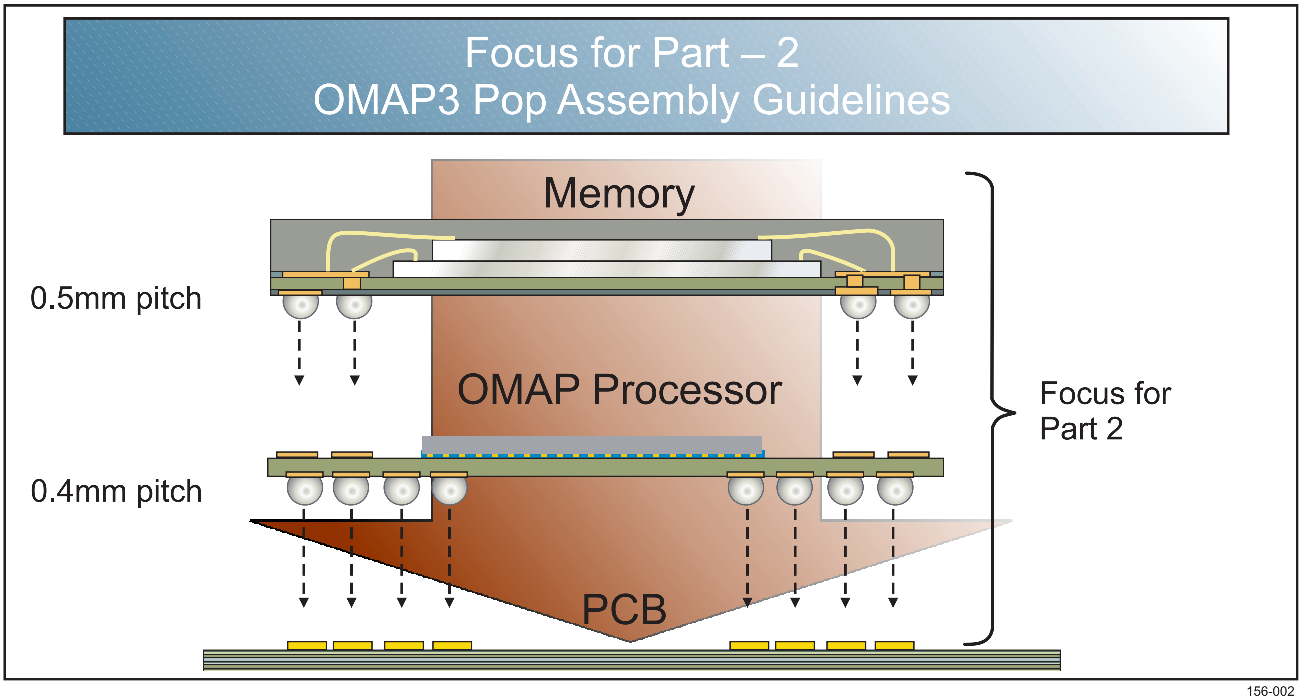


Figure 2. Overview of OMAP3 PoP Assembly Guidelines

Figure 3 shows a post-solder paste dip stack of the of the OMAP3 processor and memory.

Figure 3. Post-Solder Paste Dip Image of Memory and OMAP3 Processor

2 Focus of Part II

The focus of this document is the assembly of the OMAP3 BGA package with its companion memory mounted on top – hence the package-on-package name. These guidelines do not cover all aspects of automated assembly nor is this a study of the reliability of the PoP package.

Assembly of PoP devices and circuit boards for fine-pitch BGA packages at 0.4mm and smaller is still very new for most assembly shops. It is more of an art than a science. Thus, the material in this document will age and go out of date quickly.

Because this is a rapidly evolving technology, do as much research as possible on all aspects of fine-pitch board design. Spend time with the assembly house. Find out what they know, what experience they have, and how they document fine-pitch package assembly. Time spent pre-planning and discussing is time well spent.

Fine-pitch BGA assembly, especially PoP assembly, is considered a differentiator among assembly houses. Some assembly shops do not handle fine-pitch packages, and many do not have the appropriate equipment.

If the assembly house has never done PoP, allow time for several learning cycles before committing to volume production. The use of the OMAP3 BeagleBoard gerbers is a big help. Use this as a starting point to help the assembly shop develop its process. The processes and materials discussed will help determine the capability of the assembly site.

Assembly success comes from experienced companies willing to work together. The most common pairing is an assembly shop and a PCB board fabricator. Such companies openly share ideas and provide feedback to all parties involved in board design, fabrication, and assembly.

TI has explored several companies that have worked together and found many common points of agreement on assembly guidelines as well a few minor points of disagreement. The suggestions and recommendations, plus some precautions, in this document, originated with these companies. Always take advantage of any recommended changes from a vendor's DFM studies. And be flexible enough to change a board layout to fit the suggestions of the assembly house.

As a final word of advice, have lots of patience.

3 A Team Sport

Successful design and assembly of complex, fine-pitch circuit boards is a team sport. The days of tossing circuit diagrams over the cubicle wall to the board designer who then tosses them to the assembly shop are gone. Today's board design requires a team approach and the entire process, from component selection to assembly, requires careful coordination.

The typical team is composed of four members, each representing one of the four major steps in product fabrication: the device supplier (chips, passives, mechanical), the PCB designer, the PCB fabrication shop, and the PCB assembly shop (see Figure 4).

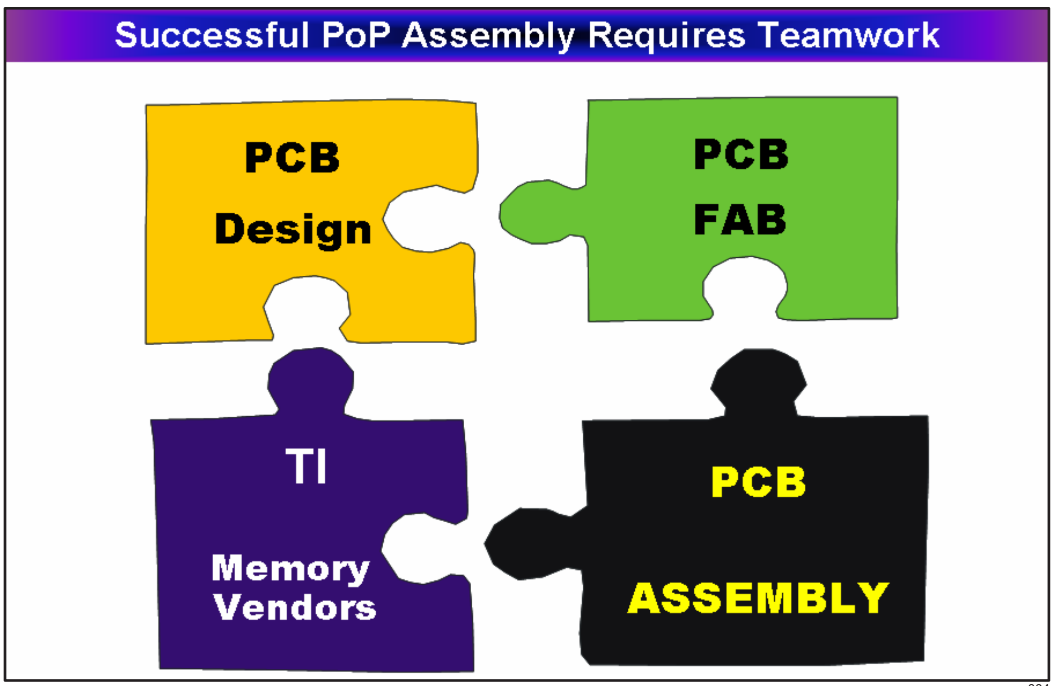


Figure 4. PoP Assembly Team

Each team member brings experience and design guidelines to bear on the task. As a result, it is not uncommon to find conflicting guidelines. These conflicts ***must*** be resolved before the start of work. To achieve the optimal yield, the philosophy of partnering in manufacture must be fostered. Constant and open communication is the key to resolving conflicts, and everyone must be in the loop.

Get to know the team members and be sure to have frequent meetings as the project proceeds from design through production. It will be money and time well spent.

4 OMAP3 and BeagleBoard

The BeagleBoard, a design based on the Texas Instruments OMAP3, was used to validate the assembly guidelines described in this document. Figure 5 is the mechanical drawing and Beagle board layout.

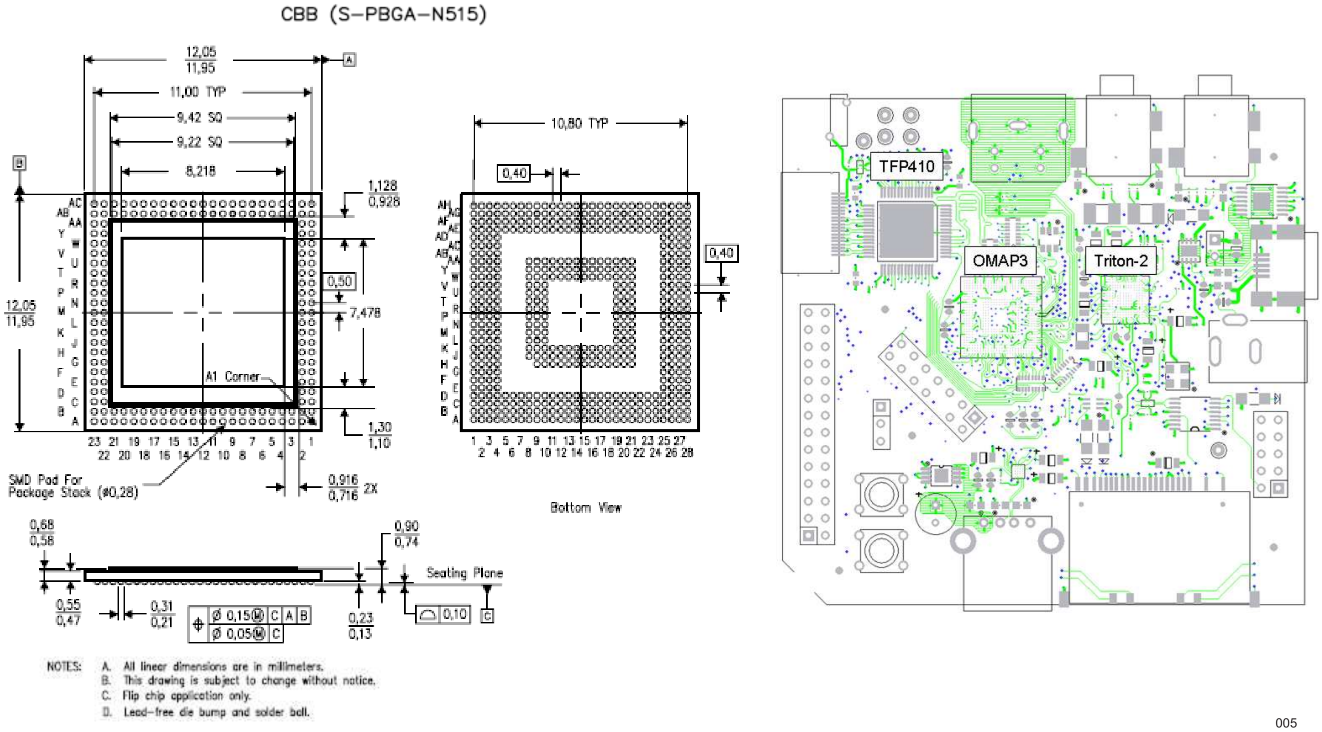


Figure 5. Mechanical Drawing and Beagle Board Layout

Figure 6 shows the PoP package-to-memory package solder mask opening (SMO) configuration.

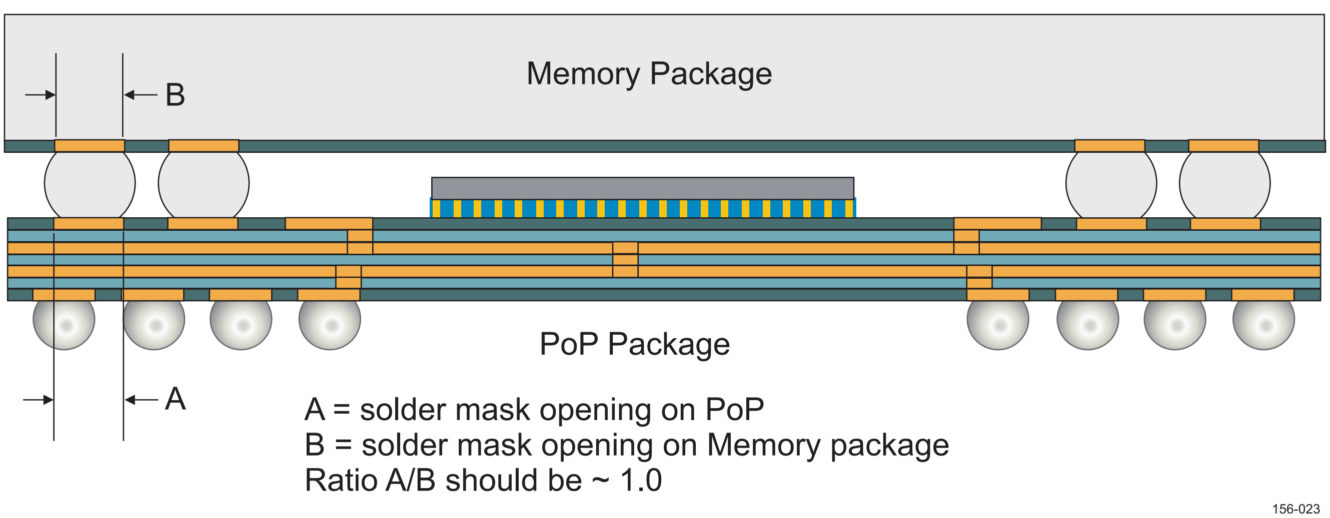


Figure 6. PoP Package-to-Memory Package SMO Configuration

4.1 Assembly Options

There are two options for building PoP assemblies, a one-pass option and a two-pass option.

In a one-pass assembly (see Figure 7), the OMAP processor is first mounted to the board, the memory is mounted to the processor, and the finished board is then run through the reflow oven in a single pass. TI has used this process successfully with several assembly shops and the yields have been good.

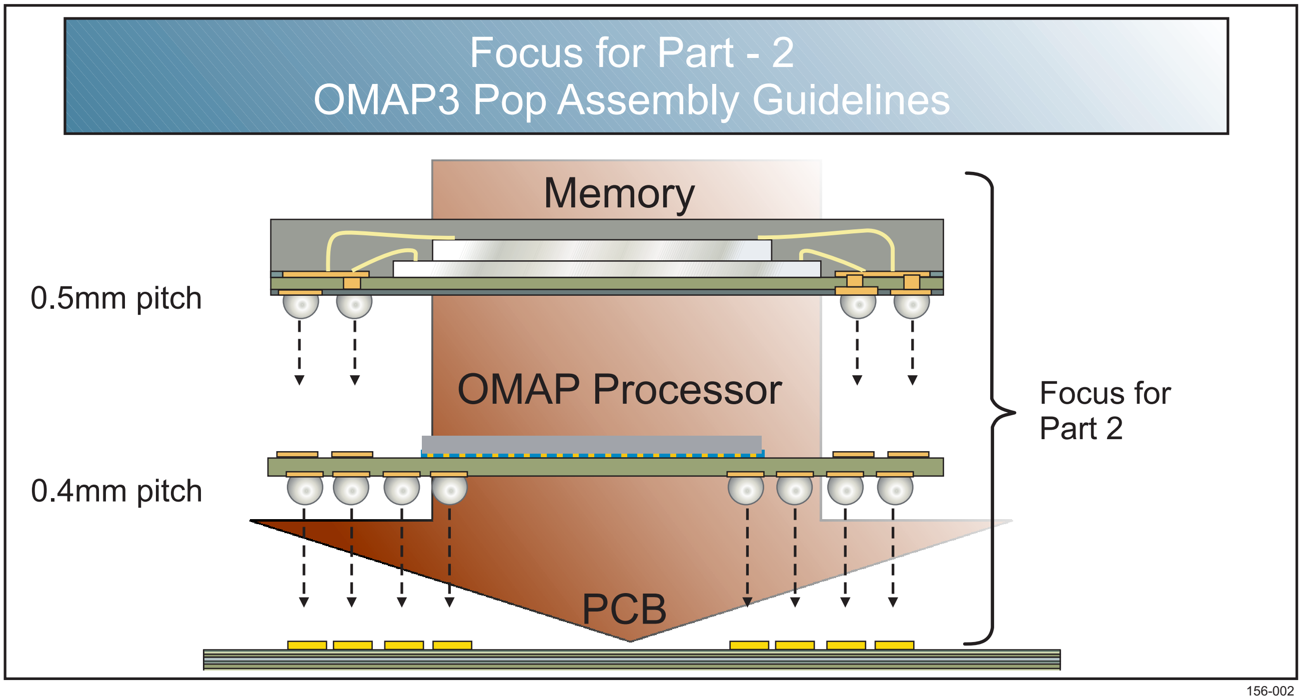


Figure 7. One-Pass Assembly Process

The two-pass assembly (see Figure 8) has an intermediate step in which the memory is first mounted onto the OMAP processor. Then these two parts are placed in a carrier tray and reflowed. These joined devices are then mounted on the circuit board and the finished board is reflowed a second time.

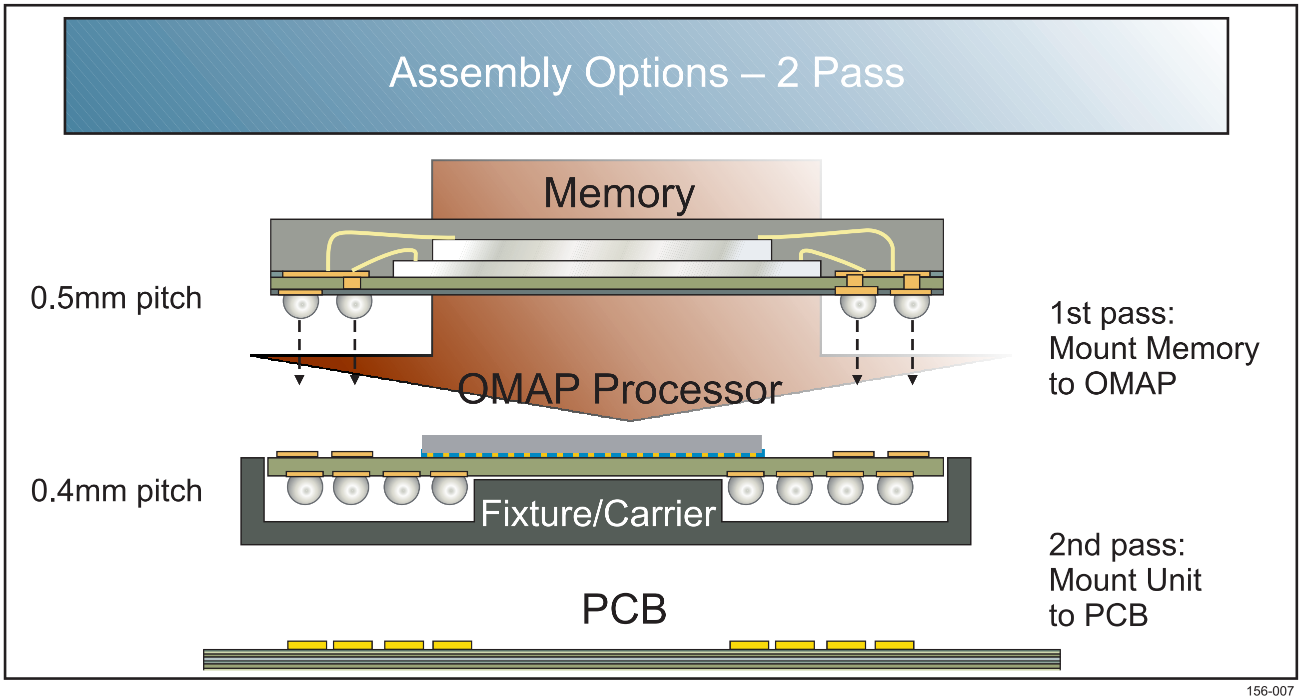


Figure 8. Two-Pass Assembly Process

TI generally recommends the one-pass process because it is more economical and provides about the same assembly yields as the two-pass technique.

4.2 Handling of Parts

Because of the exposed silicon of the OMAP3, care must be taken to avoid metal contact when handling bare parts. Impact to the silicon surface can cause damage. If the memory is not stacked on the OMAP device, the silicon is exposed to contact. In this situation, work areas must be examined for contact surfaces that can impact the edges of the OMAP device. For instance, if the PCB is dragged across the work surface, there should be no metal features that can affect the edges of the exposed silicon (see Figure 9 and Figure 10).

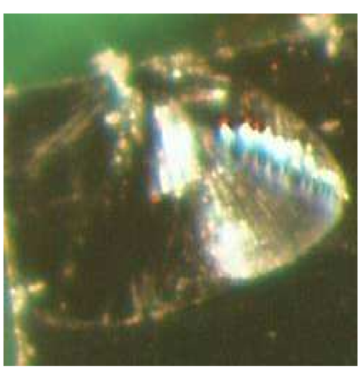


Figure 9. Chipout From Impact Damage

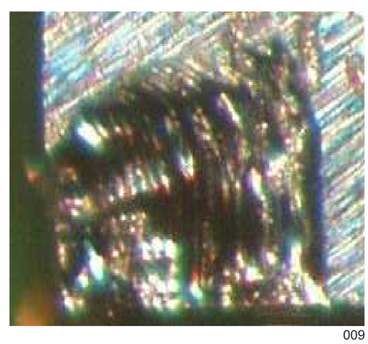
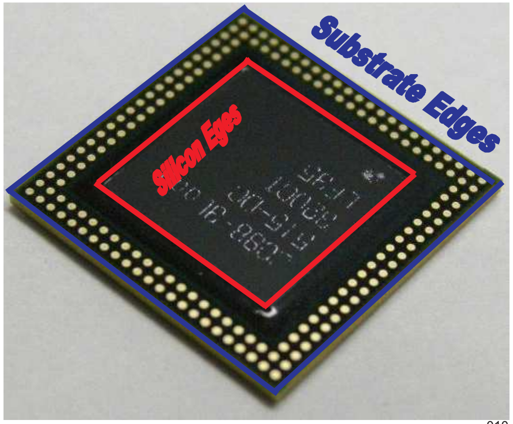


Figure 10. Corner Chipout From Impact Damage

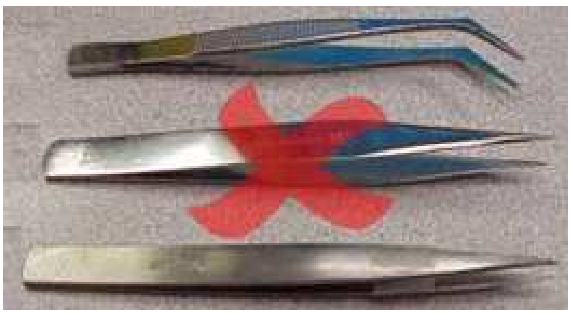
Use the substrate edges, rather than the OMAP silicon, to manipulate the device as necessary (see Figure 11).



010

Figure 11. Use the Substrate Edges to Manipulate the Device

Another best practice is to prevent metal tweezers from contacting the silicon, to prevent potential chipouts (see Figure 12



011

Figure 12. No Metal Tweezers

An additional strategy if memory is not stacked on the OMAP3 is to use higher profile components around the OMAP to act as standoffs. Placing higher profile components strategically around the OMAP3 reduces the risk of silicon damage caused by the impact of a work surface, as shown in Figure 13.

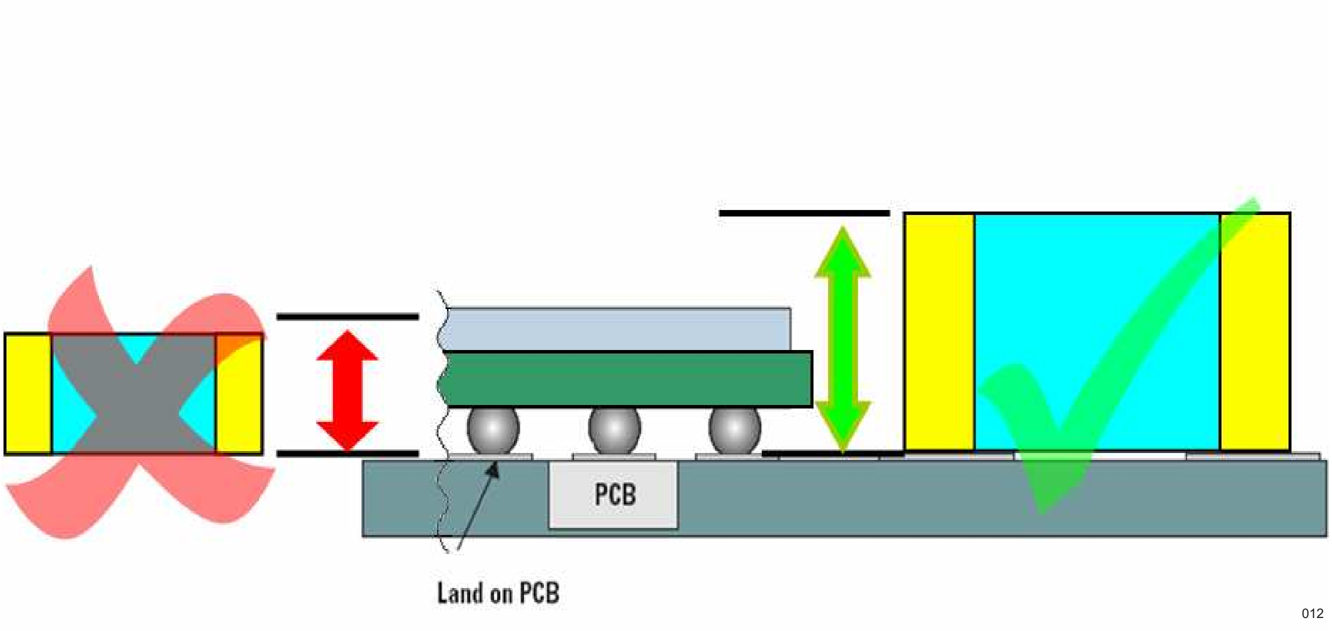


Figure 13. Passive Component Profile for Robust Handling and Assembly

4.3 Solder Paste Stencil Design

Several aspects of stencil design critically affect solder paste deposition. The recommendations here apply only to round, solder-mask-defined pads for fine-pitch BGA packages with 0.4-mm spacing. These recommendations, which assume a lead-free process, are a starting point for building design rules that fit specific manufacturing requirements and capabilities. In all cases, you must carefully monitor assembly startup to ensure that desired results are achieved.

4.3.1 Area Ratio

The area ratio (AR) is the relationship between the surface of the aperture and the inside surface of the aperture walls in the stencil. For fine-pitch BGA pads, a round aperture is recommended. The AR is more suitable than the aspect ratio for this shape.

The AR of ≥ 0.6 aperture opening to stencil wall surface area provides the best transfer efficiency and repeatability of the deposited paste. Values from 0.6 to 0.8 ensure a good paste release from the stencil.

The greatest impact on AR is stencil thickness. The equation is

$$AR = Ap/Aw = \pi r^2 / 2\pi rt$$

Where:

Ap is the aperture opening area.

Aw is the wall area.

r is the aperture opening radius.

t is the stencil thickness.

Table 1 lists the area ratio values for different aperture openings and stencil thicknesses.

Table 1. Area Ratio Values

Aperture		Stencil	AR
Diameter (um)	Radius (um)	Thickness (um)	
230	115	80	0.72
240	120	80	0.75
250	125	80	0.78
260	130	80	0.81
270	135	80	0.84
280	140	80	0.88
290	145	80	0.91
300	150	80	0.94
230	115	90	0.64
240	120	90	0.67
250	125	90	0.69
260	130	90	0.72
270	135	90	0.75
280	140	90	0.78
290	145	90	0.81
300	150	90	0.83
230	115	100	0.58
240	120	100	0.60
250	125	100	0.63
260	130	100	0.65
270	135	100	0.68
280	140	100	0.70
290	145	100	0.73
300	150	100	0.75
230	115	110	0.52
240	120	110	0.55
250	125	110	0.57
260	130	110	0.59
270	135	110	0.61
280	140	110	0.64
290	145	110	0.66
300	150	110	0.68

4.3.2 Aperture Size and Overprinting

Some assembly houses specify a slightly larger than normal opening to overprint the paste. However, for the BeagleBoard, because the pads are soldermask-defined, there is no overprinting. Overprinting is described in detail in the *OMAP25xx PCB Design Guidelines* document, where non-solder-mask defined pads are suggested.

4.3.3 Stencil Aperture Shape

A round aperture is recommended for BGA pads. This allows for an equal, uniform amount of pasted deposition on the round pad. Square corners should be avoided because they can cause unequal amounts of paste to be deposited.

4.3.4 Stencil Material and Finishing

The solder stencil must be made from stainless steel, chromium, or cobalt nickel. The apertures must be laser cut. The finish must be electro-polished to provide mirror smooth walls, which ensure good paste release.

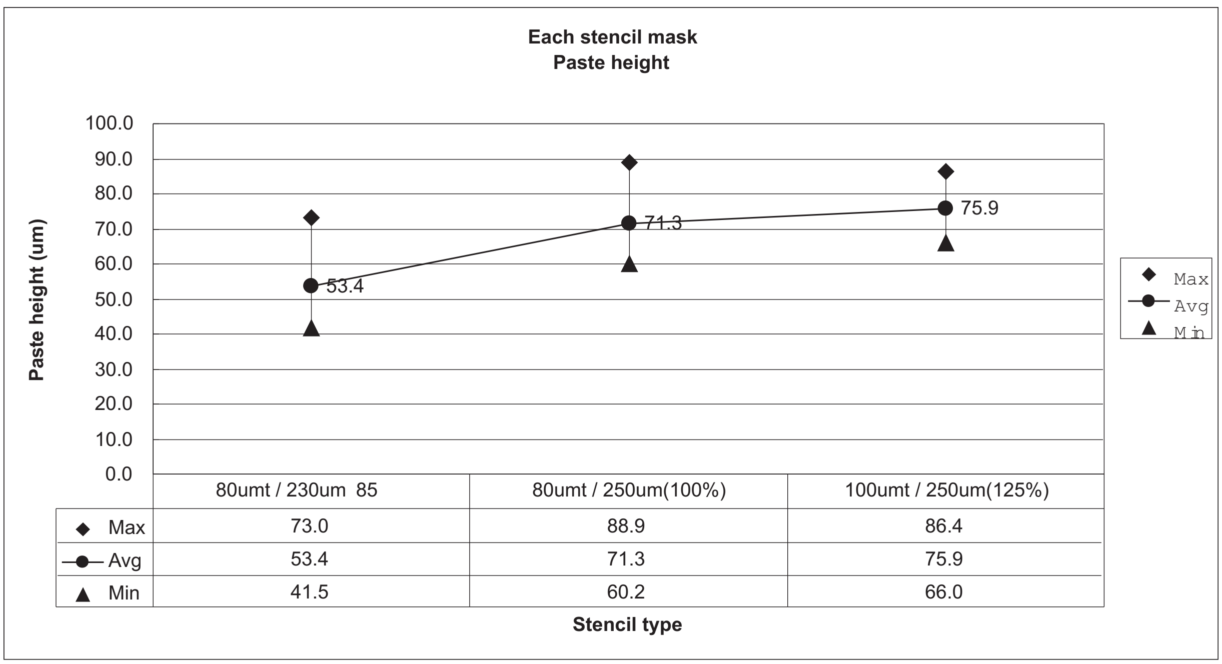
During assembly, the wider side of the aperture must be against the circuit board.

4.4 BeagleBoard Solder Stencil and Case Study

For the BeagleBoard, the solder stencil has the following parameters:

Material	Stainless steel
Thickness	4 mils (~100um)
Fabrication	Laser Cut
Aperture Shape	Round
Aperture Dimension	10 mils (~254µm)

From a subsequent evaluation, cross sections were used to characterize the solder print quality based on volumetric metal deposited. The optimal dimensions are 100um stencil thickness and 250um aperture size, as shown in Figure 14.



	80umt / 230um 85	80umt / 250um(100%)	100umt / 250um(125%)
Max	73.0	88.9	86.4
Avg	53.4	71.3	75.9
Min	41.5	60.2	66.0

Figure 14. Post Reflow Solder Height Versus Stencil Design

Figure 15 shows a cross section and measurement taken post reflow of dimension A to generate the information in Figure 14.

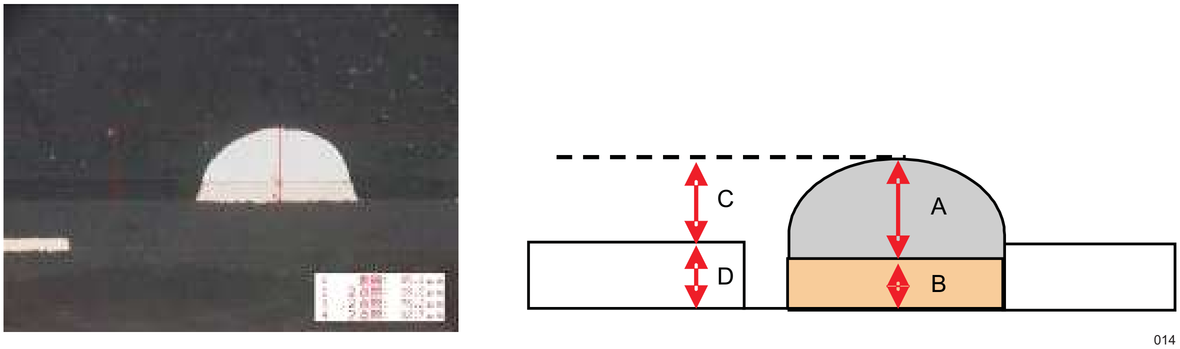


Figure 15. Sample Dimension A Measurement of Post-Reflow Solder Height

4.5 Memory Attachment Options

Memory attachment to the top of the processor is often considered one of the critical and most troublesome tasks in PoP assembly. Fortunately, this is not true. In fact, memory mounting has proven to be the least troublesome task.

The primary concern is creating a strong solder joint between the memory ball and the processor pad. Simple screening of paste cannot be used, because the die is in the way. Two processes, paste deposition and flux dipping, have been developed and used in volume production. For very high-volume production, flux dipping is preferred because of its simplicity and speed.

For the relatively low-volume BeagleBoard, a brushed-on tacky paste flux from Amtech, called LF4300 Tacky Solder Flux, was used. This material has the consistency of a gel and is simply brushed onto the OMAP3 memory pads. The BeagleBoard OMAP3 to memory attachment yield was excellent. For the data sheet for this product, see Section 7.2, *Product References*.

For high-volume automated assembly, flux and a third option, solder paste dipping for the memory device, were evaluated. Using a height gauge, various dip depths were evaluated, as shown in Figure 16, with optimal results at 66% for flux dip and 50% dip depth for solder paste. Senju 529D-1 was used for flux dip, while Senju M705-TVA was used for solder paste dipping.

Note: Insufficient/excessive deposition of flux or solder paste can sometimes be resolved with the pick and place equipment camera system.

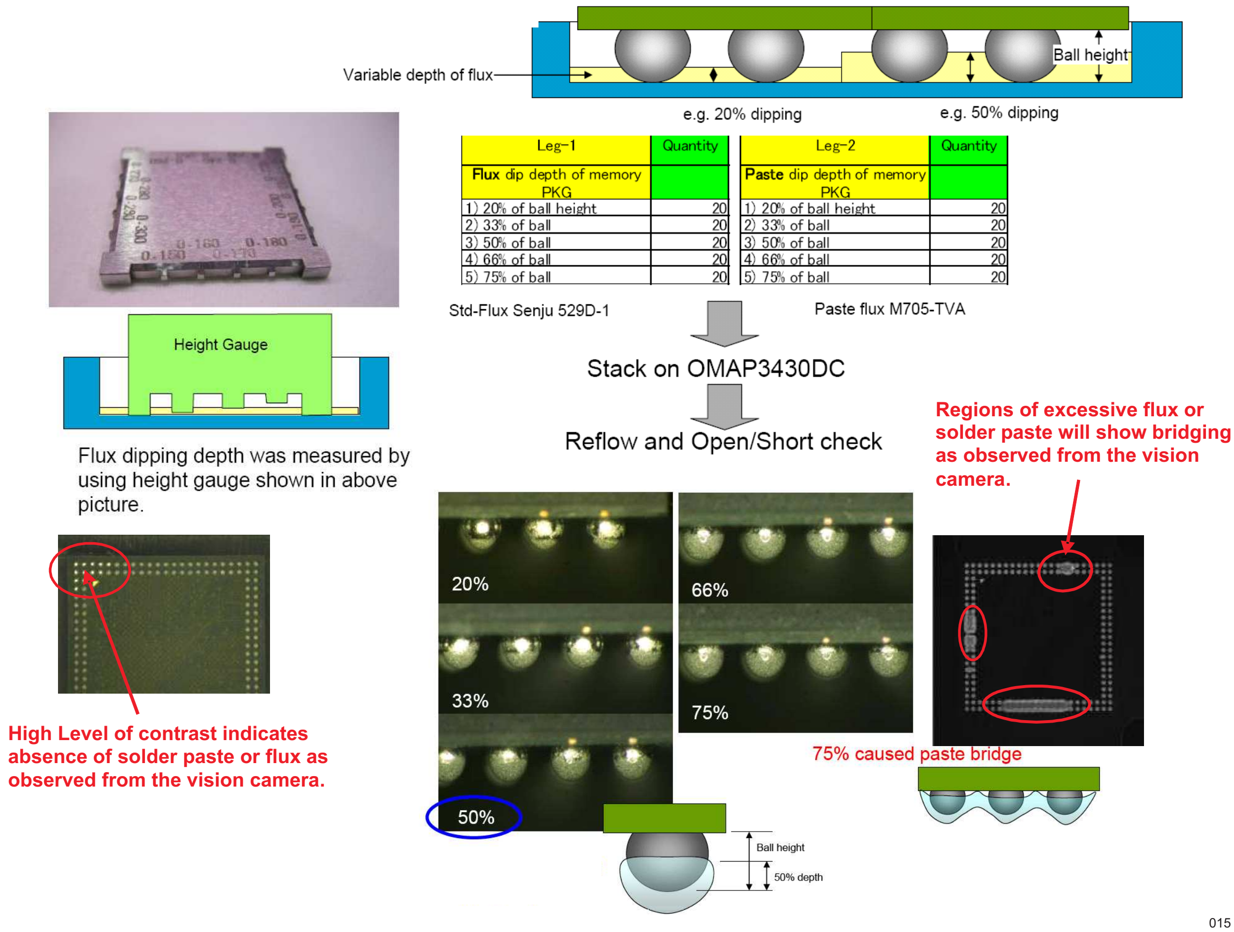


Figure 16. Factors and Techniques in Generating an Optimal Flux/paste Dip Process

4.6 Solder Paste

The most critical issues related to mounting the OMAP3 processor to the circuit board are the fluxing of the attachment surfaces and the creation of metallurgical bonds during the SMT reflow process.

From a process perspective, the following solder paste characteristics are important:

- Solder powder particle size
- Metallurgy
- Slump
- Sensitivity to temperature and humidity
- Solder content
- Type of flux residue
- Viscosity
- Propensity for solder balling

In mobile terminal PCB assembly, no-clean eutectic solder pastes with high metal content are widely used.

The choice of the particle size used in the solder powder for area array depends on the device pitch. As the device pitch decreases, the aperture size shrinks, and a smaller particle size solder paste may be required.

For the BeagleBoard, AMTECH LF4300 paste was used to attach the OMAP3 to the circuit board. It offers excellent reflow and profiling capabilities, and it is a no-clean paste that is water-soluble and lead-free. It is composed of the same alloy as the LF4300 Tacky Solder Flux, so they share the same reflow profile.

For the datasheet for LF4300 solder paste, see Section 7.2, *Product References*.

4.7 Screen Printing

Screen printing plays a key role in the assembly yield equation. The key parameters that were used for the high-yielding BeagleBoard can be used as a starting point for the screen printing process. Knowing the correct parameters allows for the solder paste to be applied in a controlled manner to prevent inconsistencies and defects.

In solder paste printing, the printer is crucial for achieving desired print quality. Screen printers available today fall into two main categories: laboratory and production. Each category has further subdivisions because companies expect different performance levels from laboratory and production printers. For example, a laboratory application that is R&D for one company can be prototype or production for another. Moreover, production requirements can vary widely depending on volume. Because a clear-cut equipment classification is not possible, the best thing to do is discuss the volume requirements with the assembly shop and ensure that the available machines match the desired application. Figure 17 shows a DEK 248 screen printer.

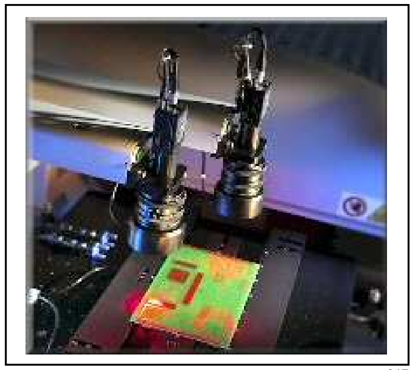
016

Figure 17. DEK 248 Screen Printer

Critical parameters for high-quality solder paste printing include print speed, print pressure, separation speed/distance (or the speed/distance at which the PCB and the stencil are separated), and printer alignment. Excellent operator training is also imperative, because solder printing is a sensitive and delicate process.

Squeegees are generally used to physically deposit and distribute solder paste evenly across the stencil. By rolling the squeegee evenly over the stencil, the solder paste passes through the stencil apertures and gets deposited on designated areas on the PCB. The stencil is then lifted, leaving behind the intended solder paste pattern on the PCB.

During solder paste printing, the PCB must be held by its support in a locked position perfectly parallel to the stencil. The squeegee angle is usually between 45 and 60 degrees. A vision system, as shown in Figure 18, is also necessary to ensure accurate printing of solder paste on the solder lands of the PCB. Modern printing equipment offers many options: computer control, vision or laser print control, environment control, automatic PCB support setup, and even stencil cleaning.



017

Figure 18. DEK 248 Camera System and Fixture

The BeagleBoard used the following screen printing parameters:

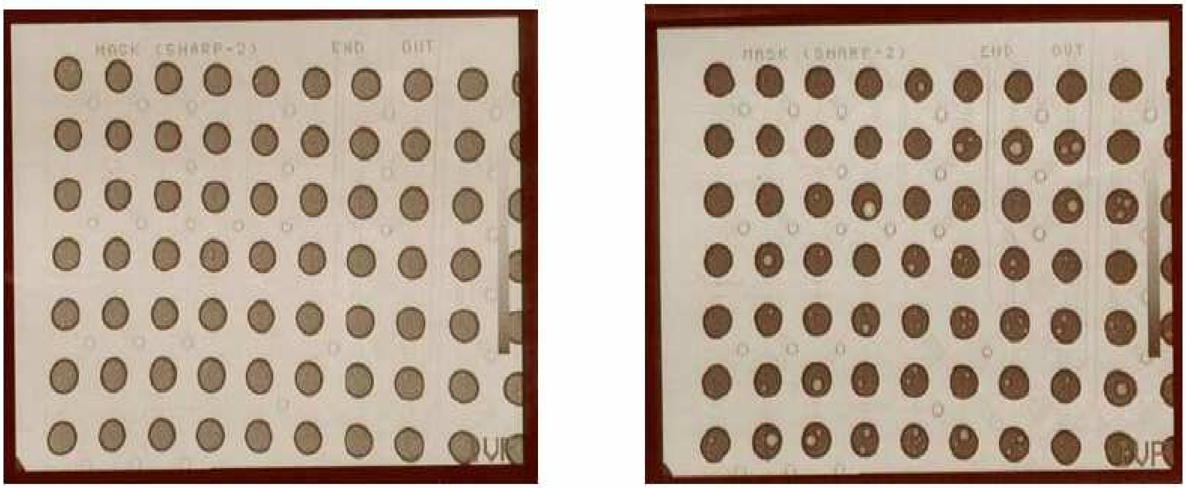
Material	DEK 248
Clamping method	Parallel
Print speed	40 mm/s/1.575 in/s
Print pressure	5.5 lbs
Separation speed	40%
PCB removal mode	Auto
Print mode	Two-pass

The environment in which solder paste printing occurs is also important. Defects such as solder bridging and poor wetting can be caused by dust particles or microscopic fibers in the air that end up on the PCB or the stencil. Quick drying of the solder paste can be caused by high ambient temperature or the presence of air draft that accelerates solvent evaporation. The viscosity of the solder paste can also be difficult to control in an environment with fluctuating ambient temperature and humidity

4.8 Reflow Atmosphere

Nitrogen used in reflow soldering decreases wetting time, provides better wetting (appearance), and allows fewer voids (inclusion of gas) in the solder joint. With SnAgCu paste, the nitrogen atmosphere decreases the wetting to about half the time needed for wetting in air. TI has repeated these tests and confirms that a nitrogen atmosphere provides the widest process window.

This means that the duration of temperatures over the melting point can be shortened by the use of nitrogen. Figure 19 shows the differences between reflow in nitrogen (left image) and air (right image) with SnAgCu solder. The number of voids is much less and the size of the voids is smaller when nitrogen is used in reflow soldering.



018

Figure 19. Reflow in Nitrogen and Air with SnAgCu Solder

For the BeagleBoard, no special atmosphere was used.

4.9 Reflow

The most popular method of reflowing solder is forced convection and/or IR radiation. other methods of solder reflow are vapor phase, laser, and hot bar. Mesh belt and edge conveyors are commonly used in reflow ovens. The following critical parameters must be controlled in the reflow profile:

- Peak reflow temperature
- Oxygen level
- Dwell time above melting temperature (liquid)
- Soak time
- Ramp rate
- Cooling rate
- Conveyor speed
- Temperature difference across the assembly (ΔT)

The ramp rate in the preheat zone must be within a reasonable range. If the rate is too low, the assembly may not reach the required soak temperature fast enough. On the other hand, if the rate is too high, components can be thermally shocked, which causes failure.

Correct soak temperature and soak times are required to evaporate solvents and to activate flux in the paste. The soak time has a significant influence on the temperature difference among components. The longer the small components are kept at a fixed temperature level, the better the chance that the large components can reach the same temperature level.

For the N2 and Air reflow development, the profile shown in Figure 20 was used, which overlaps the solder paste vendor recommendations.

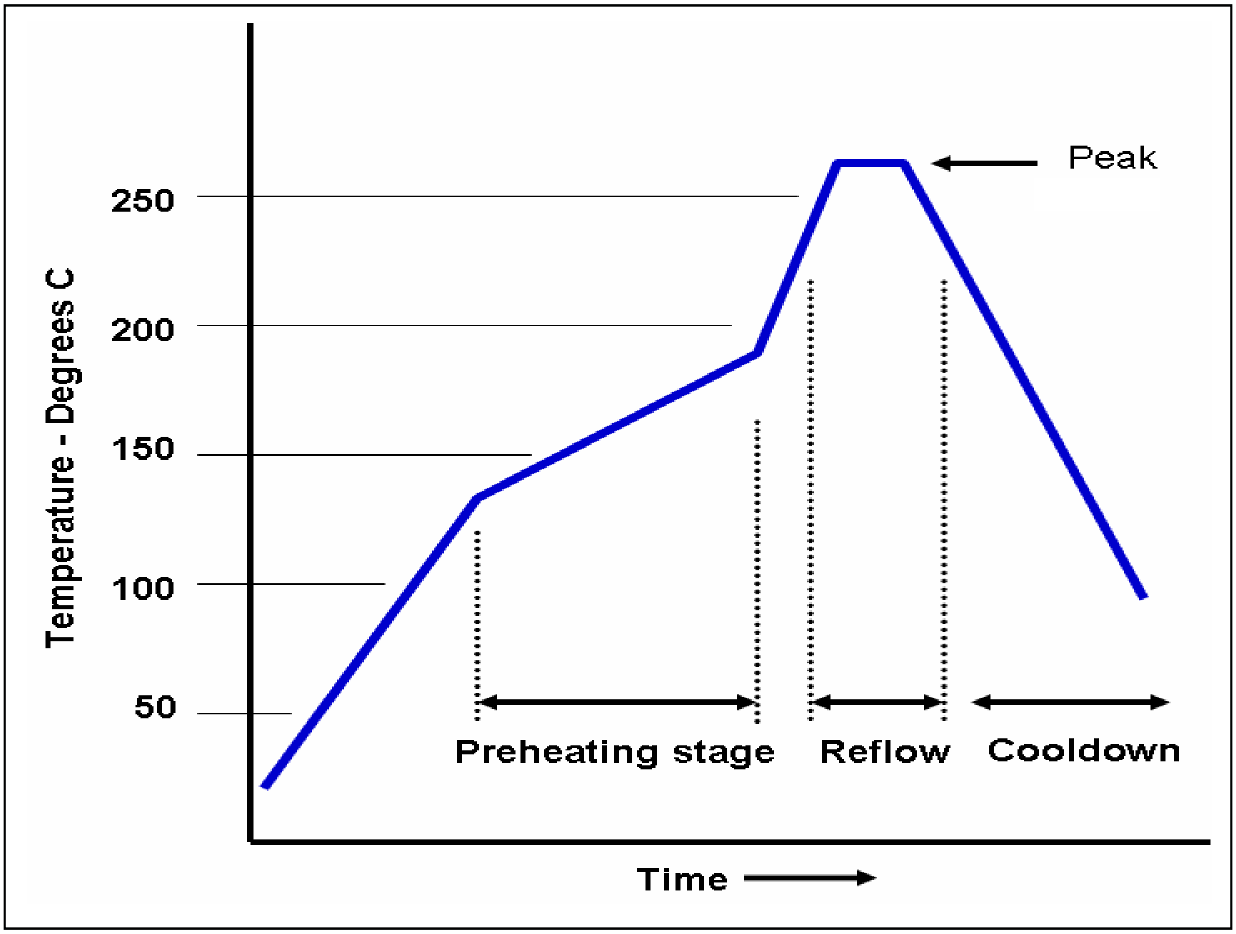


Figure 20. General Reflow Profile for Reference

CircuitCo Reflow Profile:

Number of zones in oven	12
Pre-heat temperature	165°C to 185°C
Pre-heat dwell	60 to 120 sec
Time above melting temperature (liquid state)	30 to 60 sec
Duration peak over 245C	1 to 10 sec
Peak reflow temperature	250°C
Cool down	8°C/sec
Speed	50 cm/s

For this assembly, the following paste and fluxes were used.

Paste supplier and part number	Senju M705-221
Solder paste chemistry	Sn96.5/Ag3.0/Cu.5
Solder paste dip & flux paste respectively (memory)	M705-TVA03 (Sn96.5/Ag3.0/Cu.5) & Senju 529D-1 (Flux only)

With this profile and the listed materials, yields were very good. However, this was a relatively small sample, so results can vary. Check with your solder paste supplier for recommendations.

5 Thermal Warpage

A critical aspect of PoP assembly is the type and amount of warpage that can occur during reflow of the processor and the memory. BGA package warpage during reflow soldering can cause open solder joint failure. With PoP, it is more critical at the top solder joints (memory to processor) than at the solder joints between the processor and the circuit board. As discussed in Part I, correct circuit board design prevents most common problems at that interface because with correct circuit board design, sufficient solder paste and properly sized pads between the paste and the BGA are all that are required. However, in most cases, only flux is applied to the memory device before it is reflowed. Thus, there is no additional paste volume to help fill the void as the two packages separate during reflow.

Minimizing warpage is a trade-off among materials, temperature control, and time. The assembly shop should have recommended profiles for a PoP assembly. Also, memory vendors have done extensive testing and material selection to minimize warpage. However, warpage is always present, and it may not always be repeatable from batch to batch. Be sure and talk to the memory suppliers and obtain their presentations and papers.

Locate and read the references provided in Section 7.1, *Articles, Papers, Presentations*, that discuss thermal warpage issues and test findings. Incorporating these insights into the assembly process ensures that PoP assemblies yield as well as non-PoP fine-pitch BGA assemblies.

6 Troubleshooting

This section addresses some aspects of troubleshooting assembly. There are literally thousands of variables that affect the overall success of fine-pitch assembly. This is where close cooperation with the assembly house and a willingness to try different experiments pays off.

The best way to explain many of the variables and interactions that can cause yield issues is through the use of an Ishikawa diagram, or fishbone diagram, which shows the causes of an event. Such cause-and-effect diagrams can reveal key relationships among variables and possible causes and can provide additional insight into the behavior of a process.

Typically, causes in the diagram are related to a certain category. For most assembly issues, the category is yield and the primary causes are Equipment, Process, People, Materials, Environment, and Management.

Figure 21 is a high-level overview of major causes of stress on BGA packages and solder joints.

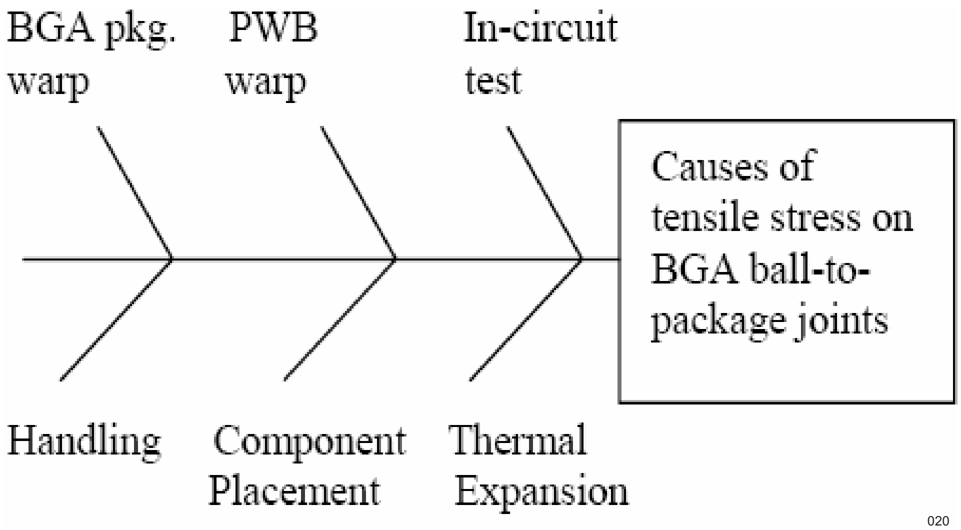


Figure 21. Causes of Stress on BGA Packages and Solder Joints

Figure 22 is typical fishbone diagram that highlights causes and effects surrounding assembly yield. Such a diagram, modified to suit specific requirements, is invaluable in creating a systematic process for evaluating problems related to fine-pitch assembly.

An excellent software package, XMID, helps build this type of diagram. XMID also has brainstorming tools for troubleshooting and trouble prevention. For the weblink, see Section 7.2, *Product References*.

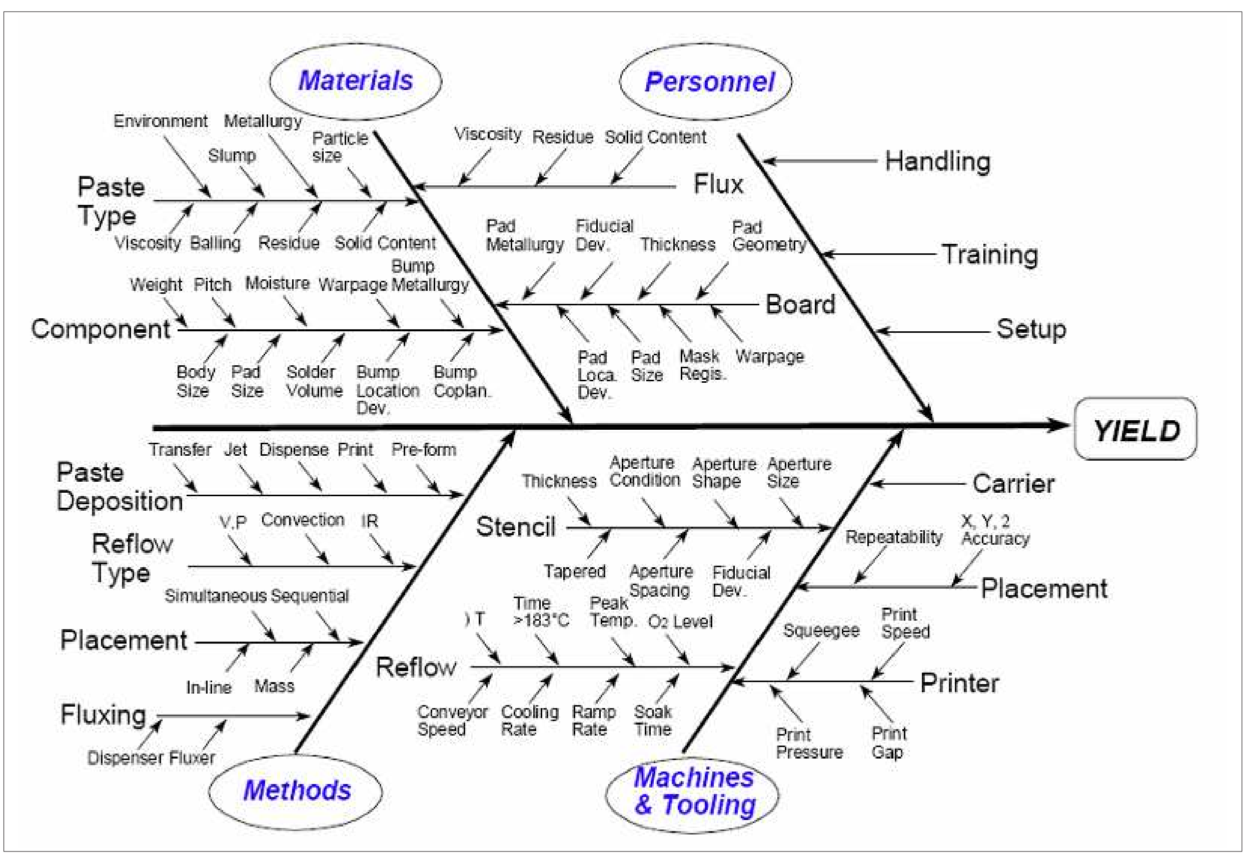


Figure 22. Factors in Assembly Yield

Finally, do not be surprised that major assembly issues are tracked down to a seemingly benign source. In other words, the cause of the problem is, in many cases, relatively simple (see Figure 23).

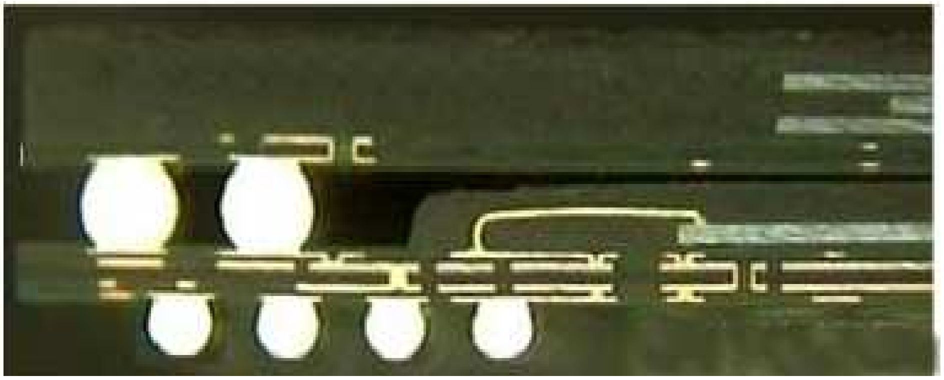


Figure 23. Cross Section of PoP Assembly

7 Acknowledgements

Jeff De Serrano, Inter-Circuit Solutions, Inc.; 400 Black Castle Drive, Lewisville, Texas 75056, Office: 972.410.0067

Ron Weindorf, AMS (Austin Division), 9715 A Burnet Road, Suite 500, Austin, Texas 78758, (512) 491-7411

Clint Cooley, CircuitCo, Richardson, TX 75081.

Franklin Troung, CircuitCo, Richardson, TX 75081

Ellen Damaso, South Central Region, Micron Technology, Inc. (972) 521-5373.

Kenji Masumoto (TIJ)

Hiroaki Ueno (TIJ)

Michael Chen (TITL)

7.1 Articles, Papers, Presentations

PoP (Package-on-Package) Stacking Yield Loss Study, Kazuo Ishibashi, Nokia Japan Co.,Ltd., ARCO Tower 4F, 1-8-1 Shimomeguro, Meguro-ku, Tokyo 153-0064, Japan. Published in the IEEE 2007 Electronic Components and Technology Conference Proceedings, 1-4244-0985-3/07.

Influence Of PCB Parameters On Chip Scale Package Assembly And Reliability, Anthony A. Primavera, Surface Mount Technology Laboratory, Universal Instruments Corporation, Binghamton, N.Y.

ECO Solder Lineup, Senju Metal Industry Co, Ltd. (SMIC), Senju Hashido-cho 23, Adachi-ki, Tokyo 120-8555, Japan.

Fan-Out Interposers Solve Fine-Pitch Micro BGA Dilemma, Mark Gilliam, Interconnect Systems, Inc. 759 Flynn Road, Camarillo, CA 93012, USA, (805) 482-2870. wwwlisipkg.com

Package on Package (PoP) Stacking and Board Level Reliability in *Printed Circuits Handbook - 6th Edition*, Clyde Coombs, Jr., McGraw-Hill, January 2008. Results of Joint Industry Study, Dreiza, Smith, Dunn, Vijayaragavan, Werner.
www.amkor.com/products/notes_papers/PoP_Stacking_IMAPS_0306.pdf

7.2 Product References

XMIND – 2008, PC-based software for fishbone diagrams, mindmaps and brainstorming sessions, www.xmind.org/us

Lead-Free Solder Products: AMTECH: Advanced SMT Solder Products Systems, 75 School Ground Road, Branford, CT 06405. Toll free: 800-435-0317, www.amtechsolder.com

DEK48 Screen Printer, DEK USA Inc., 2225 Ringwood Avenue, San Jose, CA. 95131, www.dek.com

8 Trademarks and Disclaimers

Technology for Innovators, the black/red banner, DaVinci, CCStudio, C55x, C62x and C64x are trademarks of Texas Instruments. DLP is a registered trademark of Texas Instruments. All other trademarks are the property of their respective owners.

All referenced material has been obtained from public domain papers, websites and publications and is described as part of the resource appendix.