PCB Assembly Guidelines for 0.4mm Package-On-Package (PoP) Packages, Part II

Keith Gutierrez and Gerald Coley

ABSTRACT

Once the bottom circuit board has been designed, the assembly guidelines for package-on-package (PoP) versions of the OMAP35xx processor and the memory device must be considered.

PoP packages have an enormous number of variables associated with assembly. The following factors have a major effect on the quality and reliability of final assembly: the circuit board design, the solder paste characteristics, solder paste deposition, the reflow profile, and the fluxing or solder paste deposition onto the PoP memory prior to assembly.

There are two major techniques for mounting the PoP memory: one pass and two pass. Although both are discussed, the one pass process is recommended as it is the most economical and provides about the same assembly yields as the two pass.

The BeagleBoard, discussed in the companion article to this document, *PCB Design Guidelines for 0.4mm Package-On-Package (PoP) Packages* (SPRAAV1) - which will be referred to as Part I throughout the reminder of this document - forms the basis for the understanding of 0.4mm assembly guidelines and process parameters. This application report describes only one of many processes. Your assembly facility will likely have others; it cannot cover all of the possible variations. It is strongly recommended that you perform your own trial runs and studies in conjunction with your assembly house to optimize the PoP assembly process.

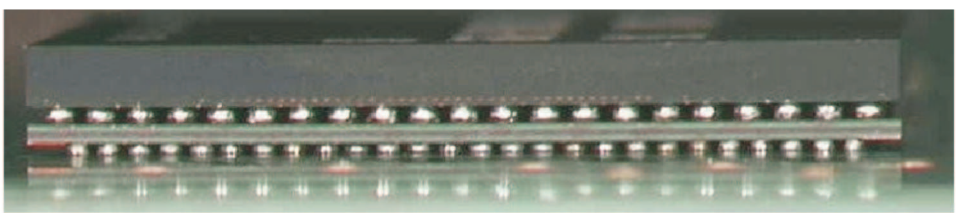

Figure 1. OMAP35xx Processor (bottom device) And PoP Memory (top device) Stack Up On PCB

Contents

List of Figures

List of Tables

1 Introduction

This application report is the second of two parts. Part I covers the main PCB design guidelines. The design of the bottom board to which the OMAP35xx 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, Part II, covers the guidelines for the assembly of printed circuit boards that use the PoP package. Included are assembly options as well as suggestions to use when qualifying and working with your assembly sites, either internal or contract.

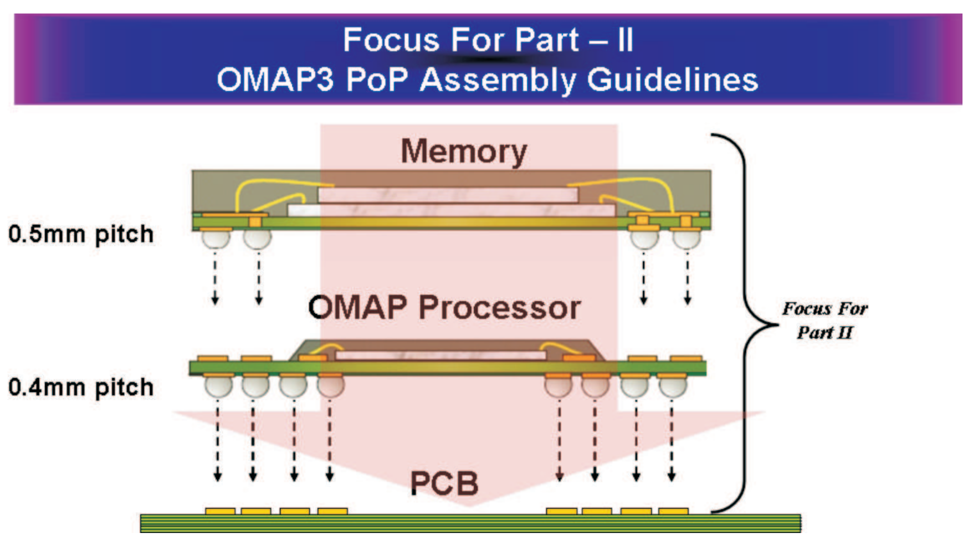

Figure 2. Part 2 - OMAP35xx PoP Assembly Guidelines

Amtech is a trademark of AMTECH Inc.
All other trademarks are the property of their respective owners.

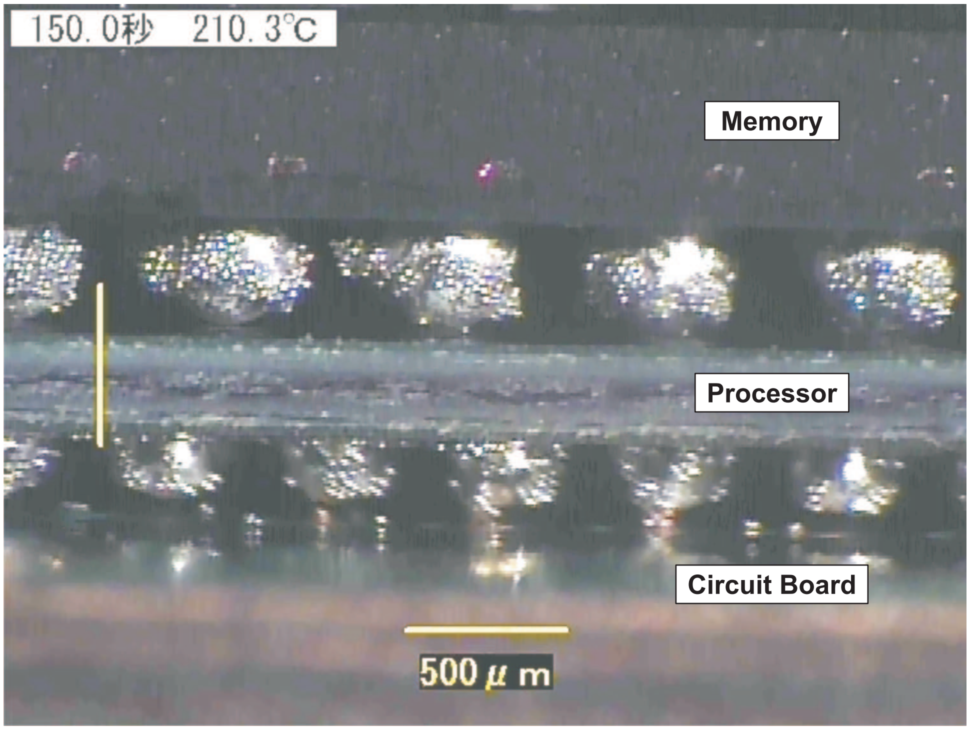

Figure 3. Close-up Of The OMAP35xx Processor And PoP Memory Stack Up

2 A Word Of Caution

This section discusses the assembly of the OMAP35xx 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. Therefore, the material in this document will age and go out of date quickly.

Since this is a rapidly evolving technology, spend some time reading the huge number of articles, papers, and company presentations on all aspects of fine-pitch board design. Spend a lot of time with your assembly house. Find out what they know, what experience they have and their documentation of fine pitch package assembly. Time spent pre-planning and discussing is time well spent.

A common theme emerged as this application report was developed and after an extensive literature search was conducted. Fine pitch BGA assembly, especially PoP assembly, is considered a differentiator among assembly houses. Also, several assembly shops indicated they could or would not handle the fine pitch packages and many did not have the appropriate equipment.

If your assembly house has never done PoP, be prepared to allow time for several learning cycles before committing to volume production. The use of the BeagleBoard gerbers will be a big help. Use it to help your assembly shop work through the issues of bringing up a new process. Based on your findings, you'll discover that your suppliers can handle the device or that you must locate a more experienced assembly shop with better equipment.

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 originated with these companies. Always take advantage of any recommended changes from a vendor's DFM studies and don't be afraid 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 cubical wall to the board designer who then tosses them to the circuit board fabrication 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 different members representing 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. There may be more members or some members may do more than one job.

Each of the team members brings their own experiences and design guidelines to bear on the task. As a result, it is not uncommon to find conflicting guidelines. These conflicts must be resolved prior to the start of work. Unresolved conflicts will result in poor assembly yields at best or 100% failure at worst. Constant and frequent communication is the key to resolving conflicts and everyone must be in the loop.

Get to know your 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.

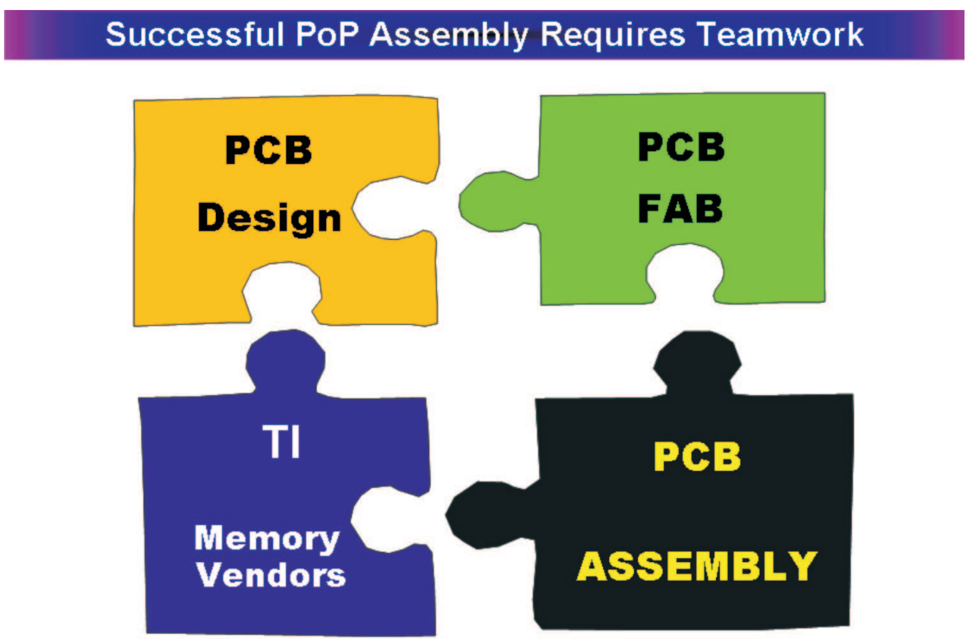

Figure 4. PoP Assembly Teamwork

4 BeagleBoard Schematics and Gerbers Available

The BeagleBoard is an open-source hardware platform based on the Texas Instruments' OMAP35xx. The platform is a complete single-board computer suitable for software development and debug. Use the following link for additional information, including the hardware schematics and gerber files: http://beagleboard.org. Also consider subscribing to the BeagleBoard RSS feed.

The BeagleBoard was used to validate the assembly guidelines discussed in this application report.

Figure 5. beagleboard.org

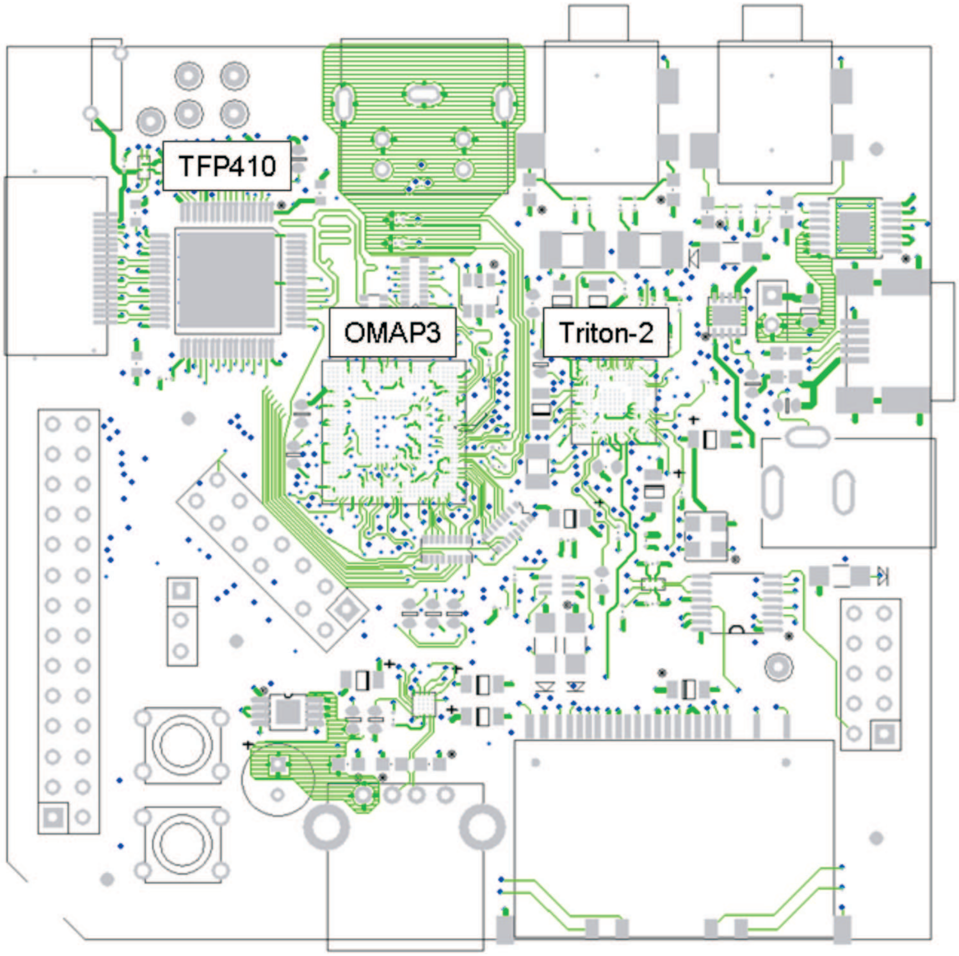

Figure 6. BeagleBoard Top Side View

5 Assembly Options

There are two options to consider when deciding to build PoP assemblies: a one pass option and a two pass option.

In a one-pass assembly the OMAP35xx 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 uses this process with several assembly shops; none have reported any issues and the yields have been good.

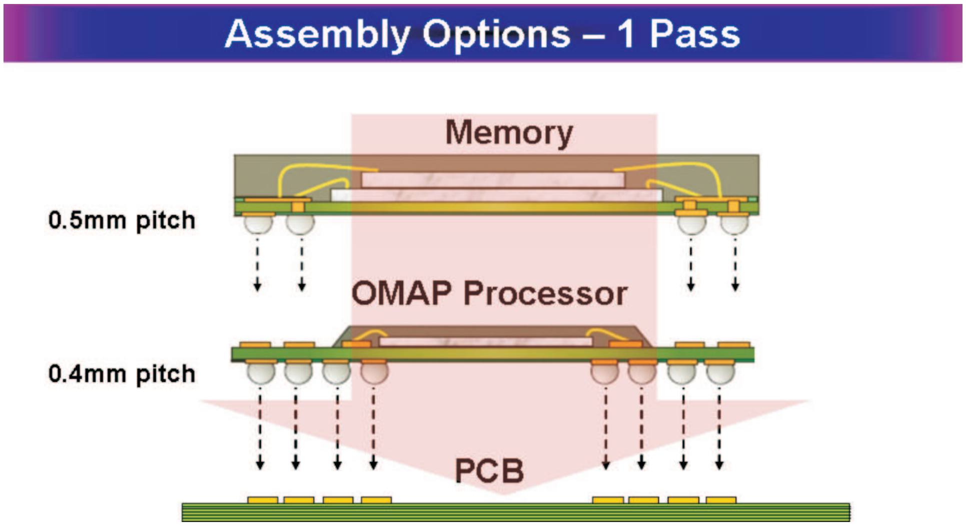

Figure 7. Assembly Options - 1 Pass

The two-pass assembly has an intermediate step in which the memory is first mounted onto the OMAP45xx 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.

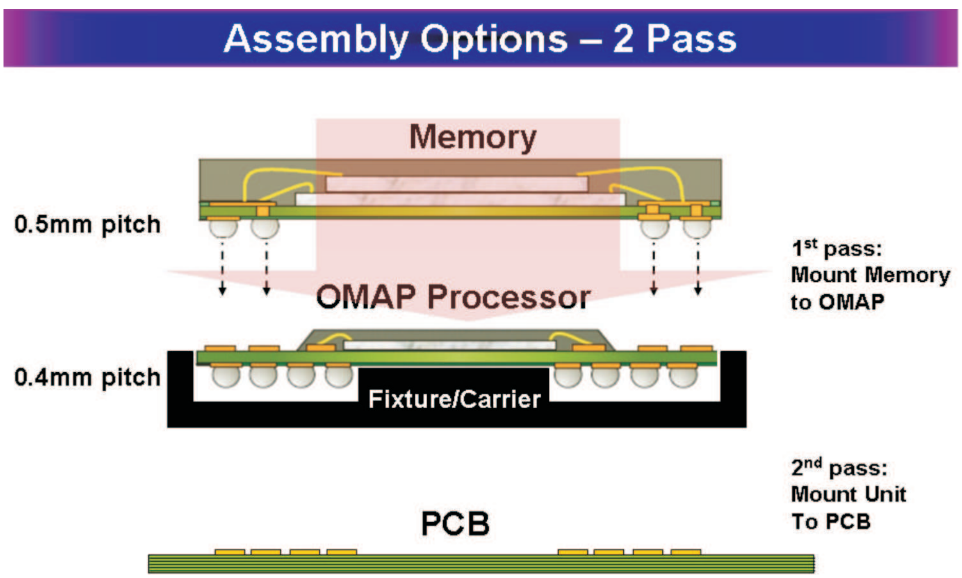

Figure 8. Assembly Options - 2 Pass

6 Solder Paste Stencil Design

Stencil design has several critical factors that affect solder paste deposition. For this discussion, the recommendations apply only to round, solder-mask-defined pads for fine-pitch BGA packages with 0.4mm spacing. These recommendations can be considered as a starting point for building design rules that fit your manufacturing requirements and capabilities, assuming a lead-free process is used. In all cases, you must carefully monitor assembly startup so as to insure the desired results are being achieved.

6.1 Area Ratio

The relationship between the surface of the aperture and the inside surface of the aperture walls in the stencil is referred to as the area ratio (AR). The area ratio is much more suitable for shapes such as circles than the aspect ratio. For fine pitch BGA pads, a round aperture is recommended.

The area ratio of ≥ 0.66 aperture opening to the stencil wall has been shown to provide the best transfer efficiency and repeatability of the deposited paste. Values from 0.66 to 0.8 insure a good paste release from the stencil.

The biggest impact on area ratio is the stencil thickness. The equation is:

$$AR = Ap / Aw \geq 0.66$$

Where, Ap is the aperture opening area and Aw is the wall area:

$$r / 2T \geq 0.66$$

Where, T is the stencil thickness and r is the aperture radius. Table 1 shows the various values of AR for different aperture openings and stencil thicknesses.

Table 1. Different Aperture Openings and Stencil Thicknesses

	Aperture Opening (mils)	Aperture Radius (mils)	Stencil Thickness (µm)	Stencil Thickness (mils)	AR
100 µm Stencil	10	5	100	3.94	0.64
	11	5.5	100	3.94	0.70
	12	6	100	3.94	0.76
	13	6.5	100	3.94	0.83
	14	7	100	3.94	0.89
	15	7.5	100	3.94	0.95
	16	8	100	3.94	1.02
125 µm Stencil	10	5	125	4.92	0.51
	11	5.5	125	4.92	0.56
	12	6	125	4.92	0.61
	13	6.5	125	4.92	0.66
	14	7	125	4.92	0.71
	15	7.5	125	4.92	0.76
	16	8	125	4.92	0.81
5 mil Stencil	10	5	127	5	0.50
	11	5.5	127	5	0.55
	12	6	127	5	0.60
	13	6.5	127	5	0.65
	14	7	127	5	0.70
	15	7.5	127	5	0.75
	16	8	127	5	0.80
6 mil Stencil	10	5	152.4	6	0.42
	11	5.5	152.4	6	0.46
	12	6	152.4	6	0.50
	13	6.5	152.4	6	0.54
	14	7	152.4	6	0.58
	15	7.5	152.4	6	0.63
	16	8	152.4	6	0.67

6.2 Aperture Size and Overprinting

Some assembly houses specify a slightly larger than normal opening to overprint the paste. However, for the BeagleBoard, there is no overprinting since the pads are soldermask defined.

6.3 Stencil Aperture Shape

A round aperture shape is recommended for BGA pads. This allows for an equal, uniform amount of pasted deposition on the round pad. Square corners can result in unequal amounts of paste being deposited and should be avoided.

6.4 Stencil Material and Finishing

The solder stencil should be made from stainless steel, chromium or Cobalt Nickel. The apertures should be laser cut. The finish should be electro-polished to provide mirror smooth walls. The mirror smooth aperture walls insure good paste release.

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

7 BeagleBoard Solder Stencil

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

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

8 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 has proven to be untrue. 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. The two processes to be described have been developed and used in volume production: paste deposition and flux dipping. For very high volume production, the flux dipping is preferred due to its simplicity and speed. A third option is solder paste dipping which has emerged as an alternative.

For the relatively low volume BeagleBoard, a brushed on tacky paste flux was used. This material, called LF4300 Tacky Solder Flux from Amtech™, has the consistency of a gel and is simply brushed onto the OMAP35xx memory pads. The BeagleBoard OMAP35xx to memory attachment yield was excellent.

9 Solder Paste

The most critical issue related to mounting the OMAP35xx processor to the circuit board is the proper fluxing of the attachment surfaces and the creation of proper metallurgical bonds during the SMT reflow process.

Solder paste characteristics that are important from a process perspective include the solder powder particle size, metallurgy, slump, temperature and humidity sensitivity, solder content, type of flux residue, viscosity, and the propensity for solder balling. No-clean eutectic solder pastes with high-metal content are used widely in mobile terminal PCB assembly.

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

For the BeagleBoard, AMTECH LF4300 paste was used to attach the OMAP35xx to the circuit board. It offers excellent reflow and profiling capabilities, plus it is a no clean paste that is water soluble and it is lead free. It is also composed of the same alloy as the LF4300 Tacky Solder Flux so they share the same reflow profile.

10 Screen Printing

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

Figure 9. DEK 248 Screen Printer

In the solder paste printing process, 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 could 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 your volume requirements with your assembly shop and be sure they offer machines to match the desired application.

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. It goes without saying that excellent operator training is also imperative since solder printing is a very sensitive and delicate process.

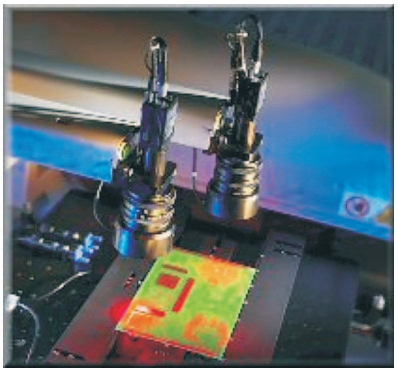

Figure 10. DEK 248 Optical Inspection Station

Squeegees are generally used to physically deposit and distribute the solder paste evenly across the stencil. By properly rolling the squeegee 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 that's perfectly parallel to the stencil. The squeegee angle is usually between 45-60 degrees. A vision system 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 set-up, and even stencil cleaning.

The BeagleBoard used the following screen printing parameters:

Machine	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 is done 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 stencil. Quick drying of the solder paste, on the other hand, can be caused by high ambient temperature or the presence of air draft that accelerates solvent evaporation. The viscosity of the solder paste will also be difficult to keep under control in an environment with fluctuating ambient temperature and humidity.

11 Reflow Atmosphere

Nitrogen gives shorter wetting time, better wetting (appearance) and 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, based on several published application reports. TI has repeated these tests and confirms that a nitrogen atmosphere provides the widest process window.

This means that the time for temperatures over melting point can be shortened by the use of nitrogen. Figure 11 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 has been used in reflow soldering. For the BeagleBoard, no special atmosphere was used.

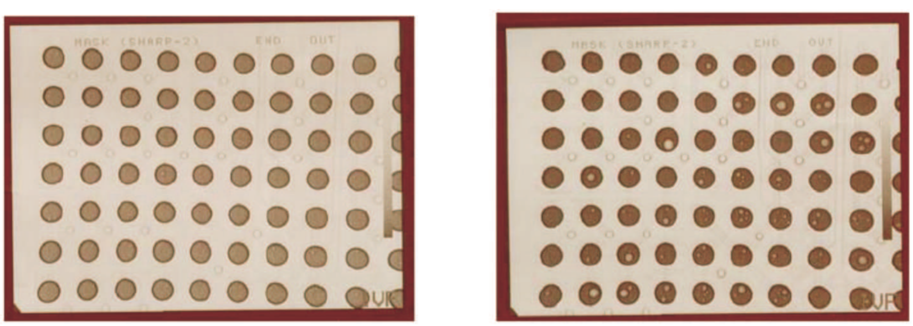

Figure 11. Differences Between Reflow in Nitrogen (left image) and Air (right image) With SnAgCu Solder

12 Reflow

The most popular method of reflowing solder is based on forced convection and/or IR radiation. Some other methods of solder reflow are vapor phase, laser, and hot bar. Mesh belt and edge conveyors are commonly used in reflow ovens. Critical parameters that need to be controlled in the reflow profile are the peak reflow temperature, oxygen level, dwell time above liquidous, soak time, ramp rate, cooling rate, conveyor speed, and the temperature difference across the assembly (ΔT).

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

Proper 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 BeagleBoard, Figure 12 shows the profile that was developed by CircuitCo in Richardson, Texas who is using the profile for all BeagleBoards.

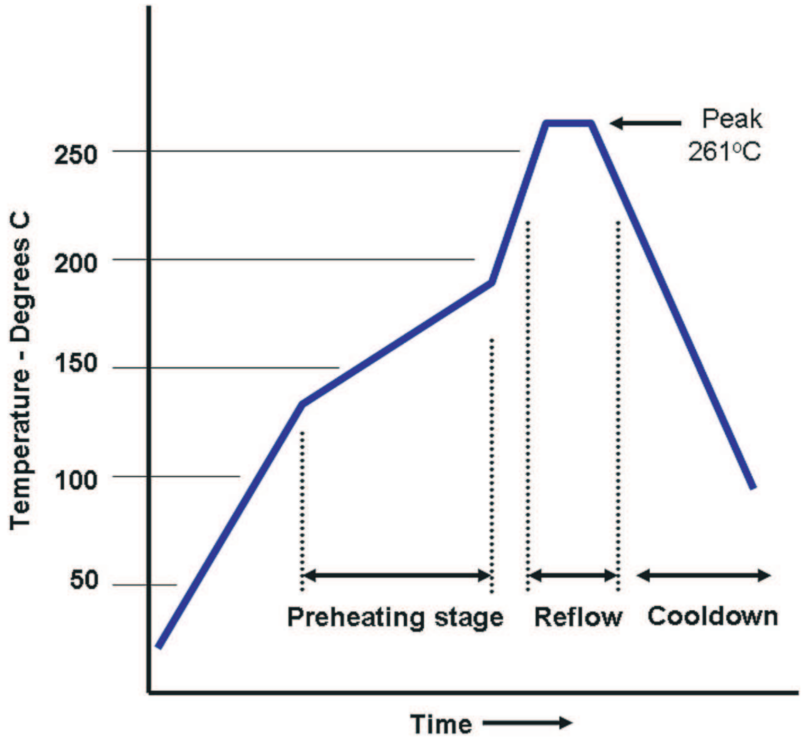

Figure 12. CircuitCo Reflow Profile

CircuitCo Reflow Profile:	
# Zones In Oven	18
Pre-Heat Temp	150°C
Pre-Heat Dwell	100 sec
Time Above Liquid	120 sec
Duration Peak	20 sec
Peak Reflow Temp	261°C
Cool Down	8°C/ sec
Speed	50cm/s

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

Paste supplier and part number	Amtech LF4300 Lead Free Solder Paste
Solder paste chemistry	Sn96.5/Ag3.0/Cu.5
Tacky paste flux (memory)	Amtech Tacky Solder Flux

With this profile and the listed materials, yields were very good. However, this was a relatively small sample so your mileage may vary.

13 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 and in the PoP case it's 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, proper circuit board design stops almost all of the common problems at that interface since it is more a case of providing sufficient solder paste and properly sized pads between the paste and the BGA. However, in most cases, there is only flux applied to the memory device before it is reflowed. Therefore, there is no additional paste volume to help with filling the void as the two packages move apart during reflow.

Minimizing warpage is a trade off between materials, temperature control and time. Your assembly shop should have recommended profiles for a PoP assembly. Also, memory vendors have done extensive testing and material selection to minimize warpage. However, it 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 application reports.

Locate and read the references provided at the end of this application report that specifically discuss thermal warpage issues and test findings. Incorporate these insights into your own assembly process and you will find that your PoP assemblies yield as well as non-PoP fine pitch BGA assemblies.

14 Troubleshooting

This section briefly touches on some of the aspects of troubleshooting assembly issues. There are literally thousands of variables that affect the overall success of fine pitch assembly. This is where close cooperation with your 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 which is also called a fishbone diagram. It is a cause-and-effect diagram that can reveal key relationships among various variables and possible causes, and perhaps, provide additional insight into a process's behavior.

Typically, causes in the diagram are often based around a certain category. For most assembly issues, the category is yield and the primary causes are equipment, process, people, materials, environment, and management.

Figure 13 shows a simple diagram that provides an overview of major causes of stress on BGA packages and solder joints.

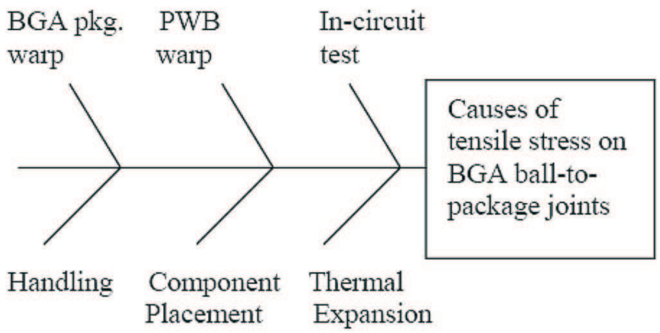

Figure 13. Fishbone Diagram of Major Causes of Stress on BGA Packages and Solder Joints

Figure 14 shows a typical fishbone diagram that highlights the various causes and effects surrounding assembly yield. Such a diagram, modified to suit your requirements, is invaluable in creating a systematic process for evaluating various problems related to fine pitch assembly.

There is an excellent software package, XMID, that helps build this type of diagram. It also has brain-storming tools suitable for use in both troubleshooting and trouble prevention. See Section 16 for the weblink.

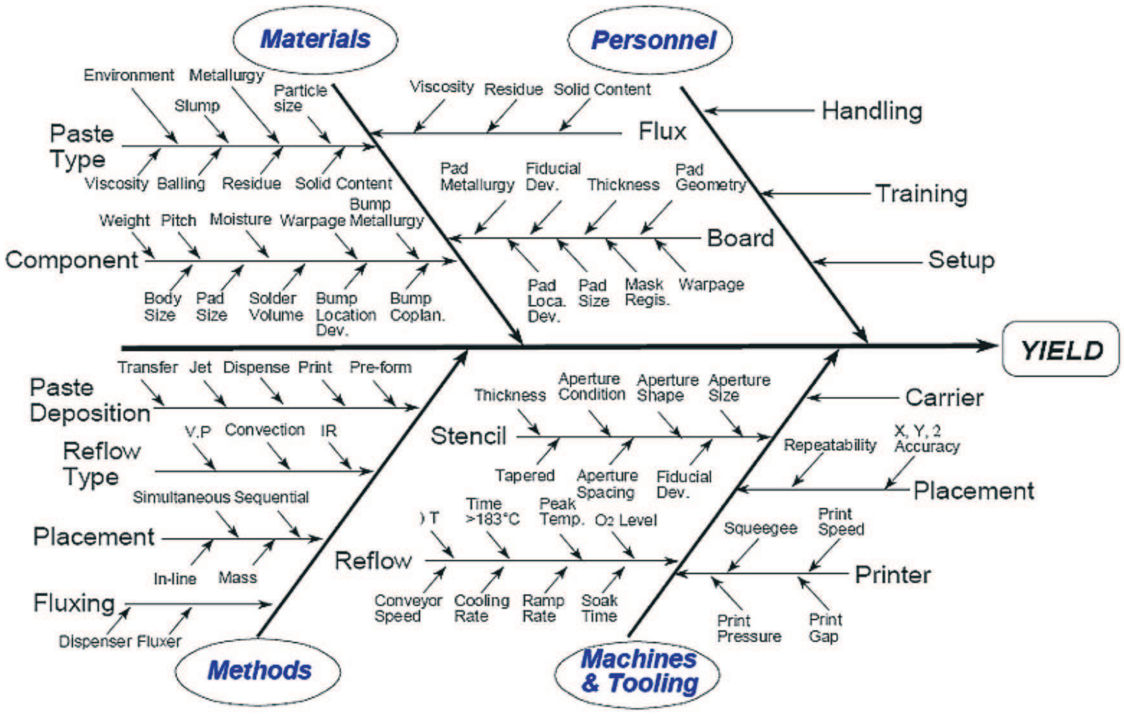

Figure 14. Fishbone Diagram Highlighting Various Causes and Effects Surrounding Assembly Yield

In many cases, the cause of major assembly issues are tracked down to something relatively simple.

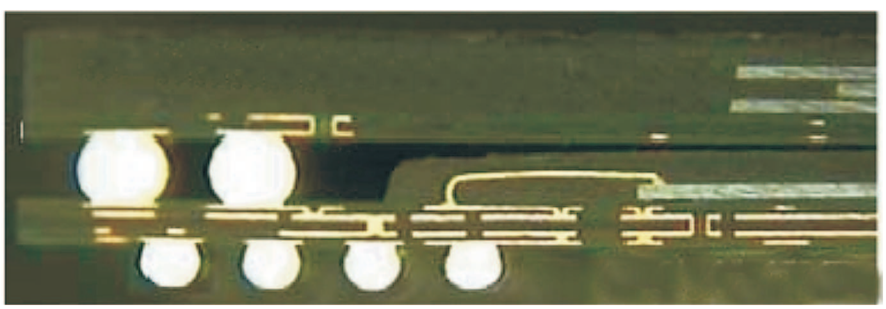

Figure 15. Cutaway View Of OMAP35xx Processor And PoP Memory

15 Acknowledgments

- 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 and Franklin Troung, CircuitCo, 675 N. Glenville #195, Richardson, TX 75081, 214-466-6690, www.CircuitCo.com
- Darren Larson, CEO and Founder / Sr. Designer ionDesign, Inc., 4410 Shoalwood, Austin, Texas 78756. (512)260-5778 Ext. 300, Cell: (512)630-9814, dlarson@iondsn.com, Fax (512)260-0640, http://www.iondsn.com.
- Ellen Damaso, South Central Region, Micron Technology, Inc. (972) 521-5373

16 References

- *PCB Design Guidlines for 0.4mm Package-On-Package (PoP) Packages, Part I* (SPRAAV1)
- *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. www.isipkg.com
- *Printed Circuits Handbook - 6th Edition*, Clyde Coombs, Jr., McGraw-Hill, January 2008.
- *Package on Package (PoP) Stacking and Board Level Reliability, Results of Joint Industry Study*, Dreiza, Smith, Dunn, Vijayaragavan, Werner. www.amkor.com/products/notes_papers/PoP_Stacking_IMAPS_0306.pdf
- *XMIND – 2008*, PC based software for fishbone diagrams, mindmaps and brainstorming sessions, www.xmind.org/us
- Lead-Free Solder Products: LF4300 by AMTECH: Advanced SMT Solder Products Systems, 75 School Ground Road, Branford, CT 06405, Toll free: 800-435-0317, www.amtechsolder.com/pdf/LF-4300.pdf
- DEK48 Screen Printer, DEK USA Inc., 2225 Ringwood Avenue, San Jose, CA. 95131, www.dek.com

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, enhancements, improvements and other changes to its semiconductor products and services per JESD46, latest issue, and to discontinue any product or service per JESD48, latest issue. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All semiconductor products (also referred to herein as "components") are sold subject to TI's terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its components to the specifications applicable at the time of sale, in accordance with the warranty in TI's terms and conditions of sale of semiconductor products. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by applicable law, testing of all parameters of each component is not necessarily performed.

TI assumes no liability for applications assistance or the design of Buyers' products. Buyers are responsible for their products and applications using TI components. To minimize the risks associated with Buyers' products and applications, Buyers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI components or services are used. Information published by TI regarding third-party products or services does not constitute a license to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of significant portions of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI components or services with statements different from or beyond the parameters stated by TI for that component or service voids all express and any implied warranties for the associated TI component or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Buyer acknowledges and agrees that it is solely responsible for compliance with all legal, regulatory and safety-related requirements concerning its products, and any use of TI components in its applications, notwithstanding any applications-related information or support that may be provided by TI. Buyer represents and agrees that it has all the necessary expertise to create and implement safeguards which anticipate dangerous consequences of failures, monitor failures and their consequences, lessen the likelihood of failures that might cause harm and take appropriate remedial actions. Buyer will fully indemnify TI and its representatives against any damages arising out of the use of any TI components in safety-critical applications.

In some cases, TI components may be promoted specifically to facilitate safety-related applications. With such components, TI's goal is to help enable customers to design and create their own end-product solutions that meet applicable functional safety standards and requirements. Nonetheless, such components are subject to these terms.

No TI components are authorized for use in FDA Class III (or similar life-critical medical equipment) unless authorized officers of the parties have executed a special agreement specifically governing such use.

Only those TI components which TI has specifically designated as military grade or "enhanced plastic" are designed and intended for use in military/aerospace applications or environments. Buyer acknowledges and agrees that any military or aerospace use of TI components which have ***not*** been so designated is solely at the Buyer's risk, and that Buyer is solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI has specifically designated certain components as meeting ISO/TS16949 requirements, mainly for automotive use. In any case of use of non-designated products, TI will not be responsible for any failure to meet ISO/TS16949.

Products		**Applications**	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Applications Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com
Wireless Connectivity	www.ti.com/wirelessconnectivity		

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265