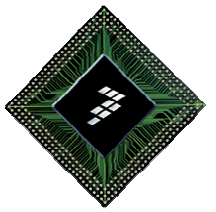
Introduction to the Plastic Ball Grid Array (PBGA)

Q1, 2008



freescale™ semiconductor

Presentation Outline

PBGA Introduction and Package Description

PC Board Design for PBGA

PBGA Assembly

PBGA Solder Joint Voids

Rework

Solder Joint Reliability

Thermal Performance

freescale ™ semiconductor

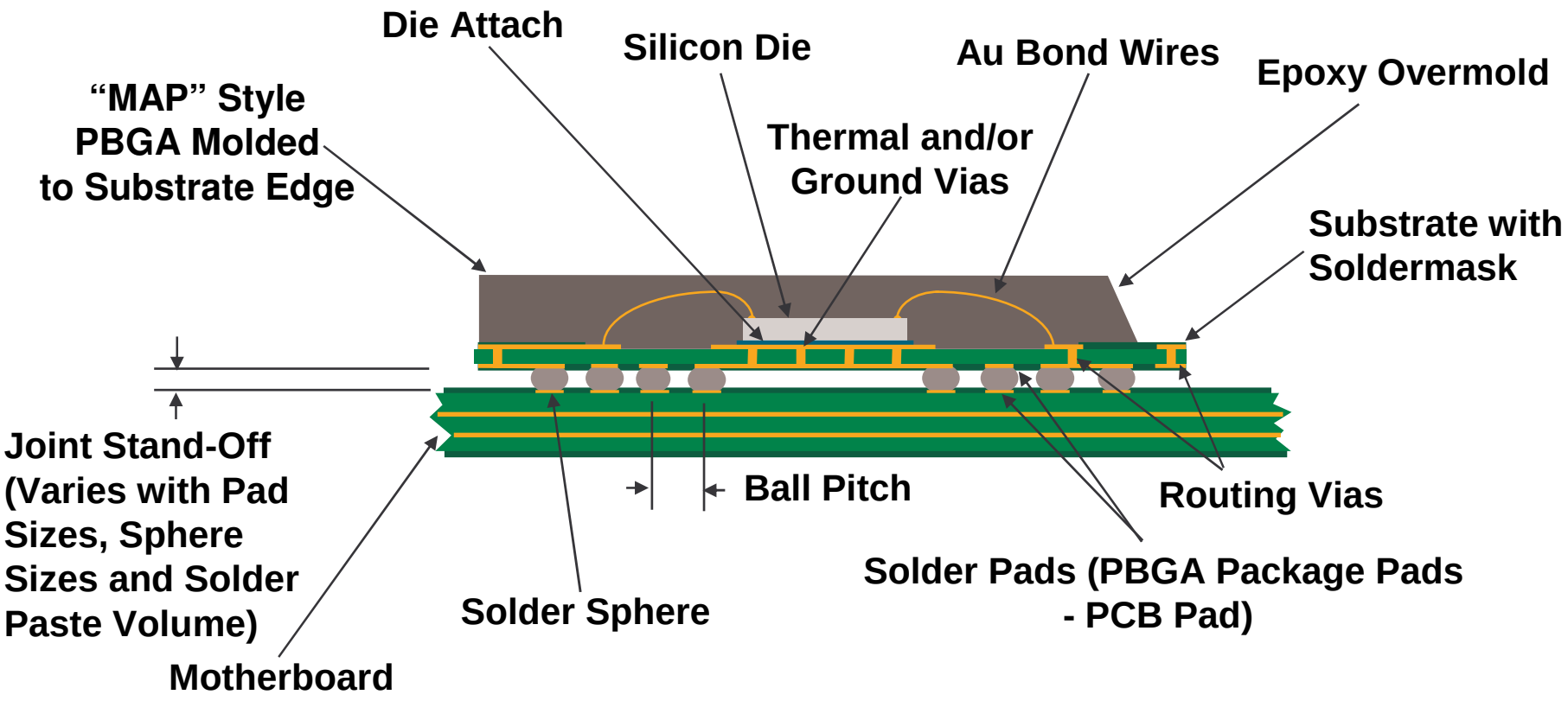
Package Terminology

PBGA >= 23x23
MAP <= 19x19

Package Terminology		
PBGA	Plastic Ball Grid Array	Can be either 2 layer or 4 layer substrates; BGA pitch range from 1.0-1.27mm; Body size >23mm x 23mm
TEPBGA	Thermally Enhanced PBGA	4 layer substrate with 2oz Cu inner layers for increased thermal performance
TEPBGA-2	Thermally Enhanced PBGA with Embedded Heatspreader	TEPBGA plus heatspreader over die
MAP	Mold Array Package	A type of PBGA, except mold cap covers entire topside surface; BGA pitch range from 0.5-1.0mm; Body size <19mm x 19mm

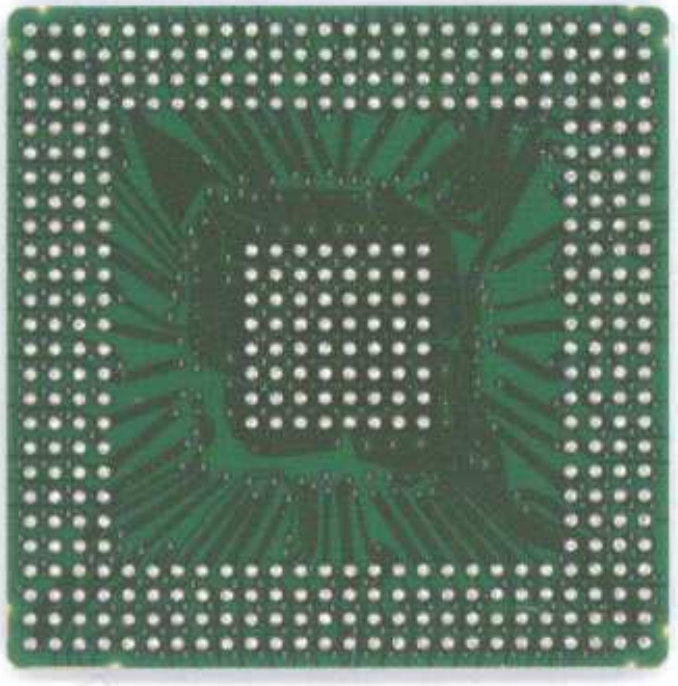
freescale™ semiconductor

PBGA BGA Construction



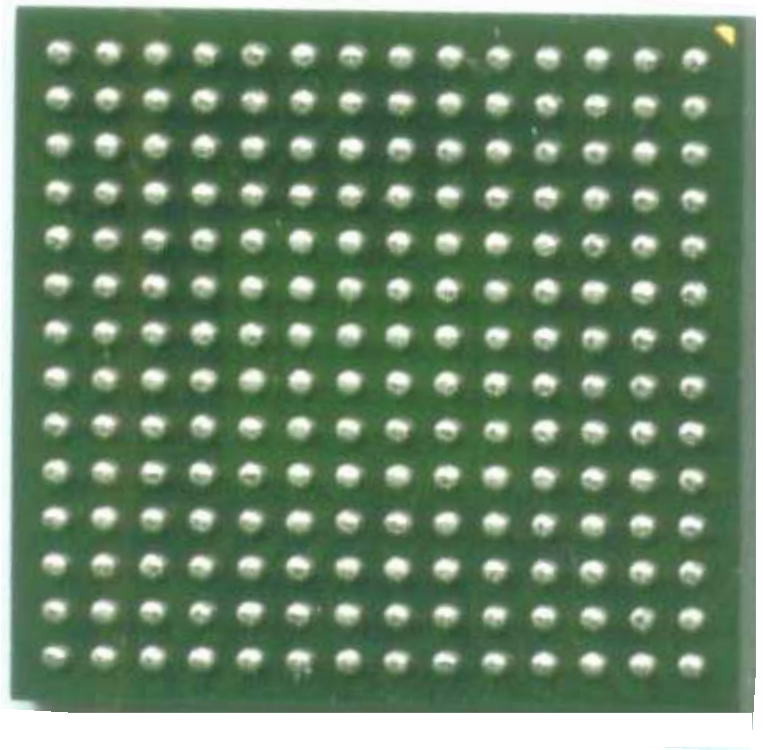
PBGA Introduction and Package Description

Typical Plastic Ball Grid Array
(416 Pin, 1.0 mm Pitch, 27x27 mm sq.)



MAP Introduction and Package Description

Typical MAP Ball Grid Array
(196 Pin, 1.0 mm Pitch, 15x15 mm sq.)



PBGA Introduction and Package Description

PBGAs ship with one of several sphere compositions depending if they are Pb-free or lead-containing

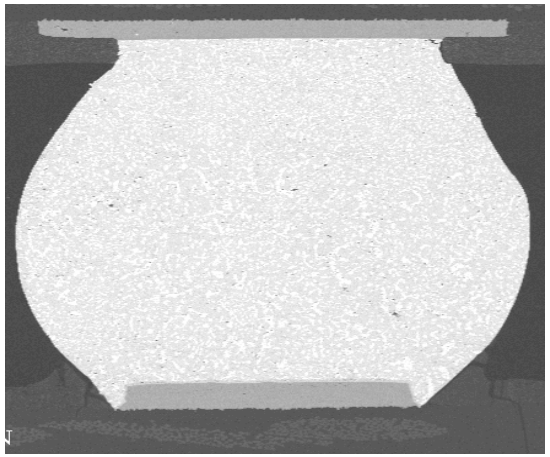
- All PBGA spheres should collapse during assembly to PCB
- Collapse is controlled by pad geometries on package and PCB and solder surface tension

Lead containing sphere compositions:

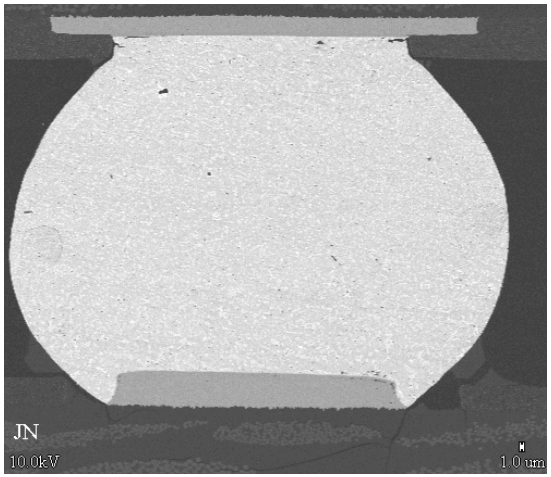
- The most common is 62Sn36Pb2Ag
- Some may contain 63Sn37Pb

Lead-free sphere compositions:

- The most common is 95.5Sn4.0Ag0.5Cu, commonly called SAC405
- Devices used in handheld applications may use SAC105 (98.5Sn1Ag0.5Cu)
- Some parts may contain 96.5Sn3.5Ag



SnPbAg PBGA Sphere



SAC405 PBGA Sphere

PBGA Introduction and Package Description

Solder alloy melting temperatures:

Composition (Weight %)	95.5Sn4.0Ag0.5Cu (SAC405)	98.5Sn1.0Ag0.5Cu (SAC105)	96.5Sn3.5Ag	62Sn36Pb2Ag	63Sn37Pb
Solidus (ºC)	**217**	**215**	**221**	**179**	**183**
Liquidus (ºC)	**224**	**227**	**221**	**189**	

References:
http://www.pbfree.com/alloys.php
http://www.indium.com/products/sorted_by_temp.pdf
http://www.boulder.nist.gov/div853/lead%20free/props01.html

PBGA Advantages

BGA: Ideal interconnect technology

- Using solder spheres as leads:
  - Provides perfect self alignment
  - Allows large tolerance in placement accuracy
  - Eliminates lead coplanarity issues
  - Allows high number of I/O connections with large pitch
    - Large savings in board real-estate
  - Offers electrical and thermal advantages
- SMT manufacturing processes simplified
- SMT manufacturing equipment demands less stringent
- SMT manufacturing yields dramatically improved versus fine pitch leaded parts

Presentation Outline

PBGA Introduction and Package Description

PC Board Design for PBGA

- Pad Diameter

PBGA Assembly

PBGA Solder Joint Voids

Rework

Solder Joint Reliability

Thermal Performance

freescale™ semiconductor

PC Board Design for PBGA

For PBGA, both PC board solder pad and stencil aperture diameters should generally be made 1:1 with the package solder pad diameter

Stencil thickness is typically dependent on other components on the PCB such as fine pitch leaded and discretes

- 0.10 to 0.20 mm thick stencils are commonly used with PBGA
- In some cases such as larger body size PBGAs, a thicker stencil may provide improved assembly robustness

Package pad diameters are generally one half the pitch, but can vary slightly on some products – check with Freescale or your distributor if there are questions

Below is a table of some typical PBGA pad diameters:

Ball Pitch	Package Pad Dia.	PCB Pad Dia. (1)	Stencil Opening Dia. (2)
1.27	0.635	0.635	0.635
1.00	0.40 to 0.50	0.40 to 0.50	0.40 to 0.50
0.80	0.40	0.40	0.40
0.65	0.3 to 0.325	0.3	0.3
0.50	0.25 to 0.3	0.25 to 0.3	0.25 to 0.3

Note:

(1) Studies have shown that PCB pad diameters up to approximately 10% smaller than the package pad still provides good board-level thermal cycling reliability. A PC board solder pad that is of greater diameter than the package pad is not recommended

(2) General industry practice for PBGA has stencil opening diameter equal to PCB pad diameter.

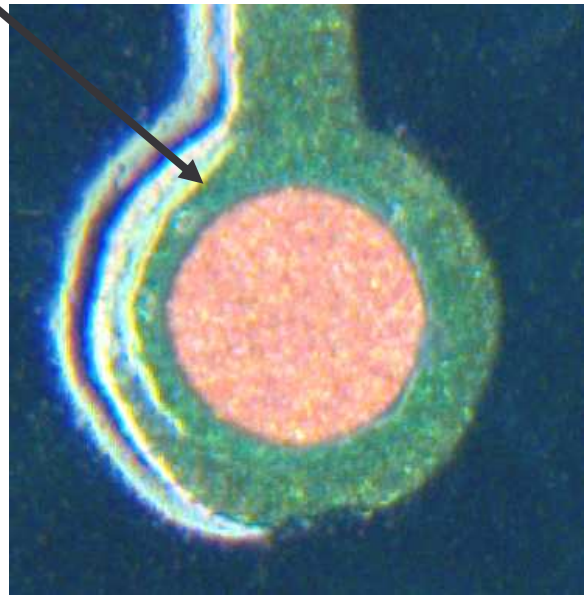
freescale semiconductor

SMD vs NSMD PCB Pads for PBGA

There are two types of PCB pad configurations for PBGA:

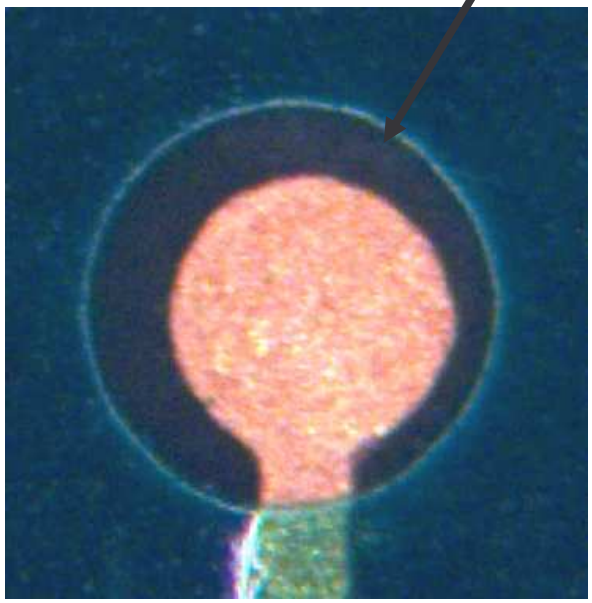
- SolderMask Defined pads (SMD) which are the type on the PBGA itself
- Non-SolderMask Defined (NSMD) or Cu defined pads are most commonly used on PC boards within the industry

Soldermask Overlap



SMD Pad

Soldermask Clearance



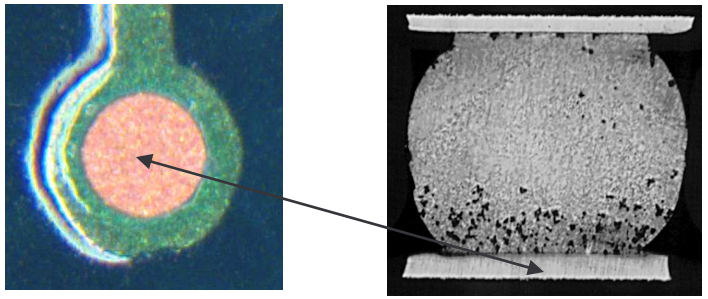
NSMD Pad

Comparison of PC board SMD (left) and NSMD (right) pads shown with OSP (OSP or organic solderability protectant) surface finish.

Besides the OSP shown above, there are many surface finishes available: HASL, Immersion Ag, ENIG, etc….

freescale™ semiconductor

Trade-Offs Between SMD and NSMD PCB Pads for PBGA



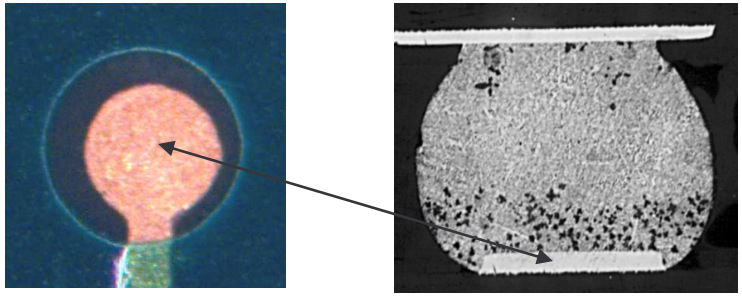
SMD = SolderMask Defined

Advantage

1. Strongest mechanical strength
2. More copper, better heat spreading to board
3. More heat resistant - multiple rework

Disadvantages

1. Potential stress point
2. Foreign matter entrapment on pad
3. Possible worse dimensional tolerance on solderable diameter versus NSMD



NSMD = Non-SolderMask Defined

Advantages

1. Most common pad configuration
2. Potentially the most solderable pad
3. No point of stress concentration at PC board
4. More copper to copper space to route traces between pads

Disadvantages

1. Reduced pad to PCB adhesion strength
2. May be weak link in other mechanical testing, i.e. board bending, mechanical shock and vibration
3. Non-symmetrical joint fails at the package interface first in thermal cycling

PCB Pad Design Guidelines

General PCB Pad Dimensions

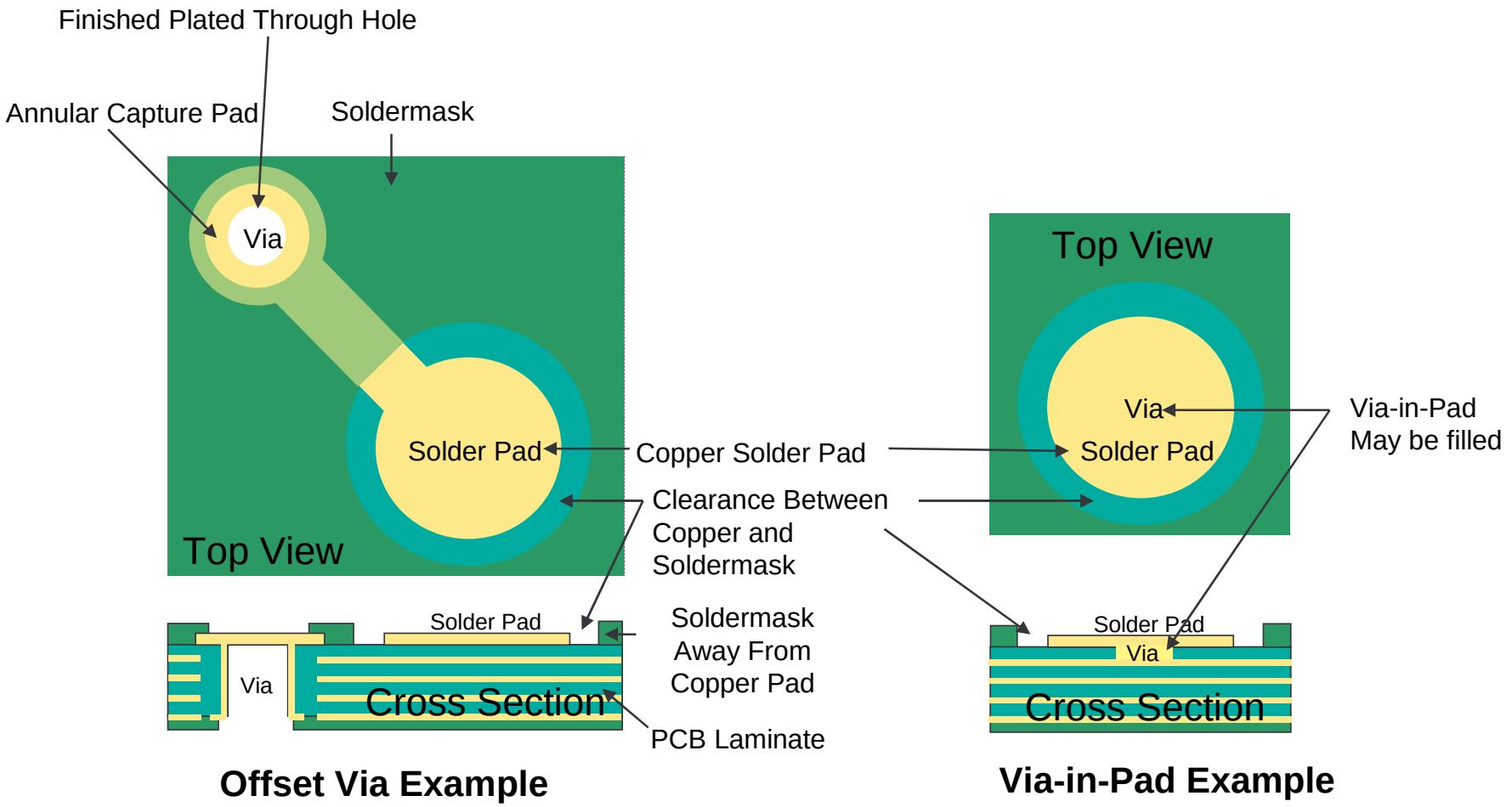
- Pad solderable diameter equal to package pad diameter
  - Smaller PC board pad diameter by approximately 10% may ease routing and still provide good reliability
- NSMD pads - soldermask opening 0.125 mm > pad diameter
- SMD pads - Cu pad area 0.125 mm > soldermask opening
- Surface finishes include: organic solderability protectant (OSP), HASL, electroless or electrolytic nickel/gold (NiAu), or immersion silver (Ag)

PCB Routing Vias

- For 1.27, 1.0 and 0.8 mm pitches, via interstitial between BGA pads
  - Via Cu capture pad set by drill size of via
  - Capture pad usually tented with solder mask
  - Trace between pad and capture pad should be covered with soldermask
    - Too small a width of soldermask can lead to soldermask lift off the trace
- For 0.8, 0.65 and 0.50 mm pitches, via-in-pad structures
  - Via is part of BGA pad
  - Filled via best for void minimization of final solder-joint
  - Contact PCB fabrication vendors for manufacturing capability
    - Laser or mechanical drilling, as appropriate
    - Size of drill hole for via
    - Through-hole via or single layer via

freescale™ semiconductor

PCB Pad Design



Presentation Outline

PBGA Introduction and Package Description
PC Board Design for PBGA
PBGA Assembly
- Pre-assembly Handling
- Placement and Reflow
- Secondary Reflow
- Cleaning and Inspection

PBGA Solder Joint Voids
Rework
Solder Joint Reliability
Thermal Performance

freescale™ semiconductor

PBGA Pre-Assembly Handling: Moisture Sensitivity

Similar to QFP and other leaded and leadless plastic packages, most PBGAs are generally moisture sensitive

- Follow moisture sensitivity guidelines printed on the package dry-pack label
- Check the label for maximum allowable reflow temperature for product

Jedec Moisture Level Table		
MSL Level	**Floor Life**	**Storage Condition**
1	Unlimited	30C/85%RH
2	1 Year	30C/60%RH
2a	4 Weeks	30C/60%RH
3	168 Hours	30C/60%RH
4	72 Hours	30C/60%RH
5	48 Hours	30C/60%RH

Source: JEDEC JESD22-A113

Floor life is the time allowed out of dry-pack before a rebake is required.

freescale™ semiconductor

PBGA Pre-Assembly Handling: Moisture Sensitivity

Easy to avoid PBGA package delamination

- Follow instructions on shipping container
- Establish strict process control and procedures to insure safe operation conditions.
  - Use dry nitrogen or dessicator cabinets to store devices after opening packages
- Rework boards within time allowed in room environment
- Failure to follow MSL guidelines can result in package failure.

Solder Paste Printing for PBGA

Solder paste stencil thicknesses generally ranges from 0.10 to 0.20 mm

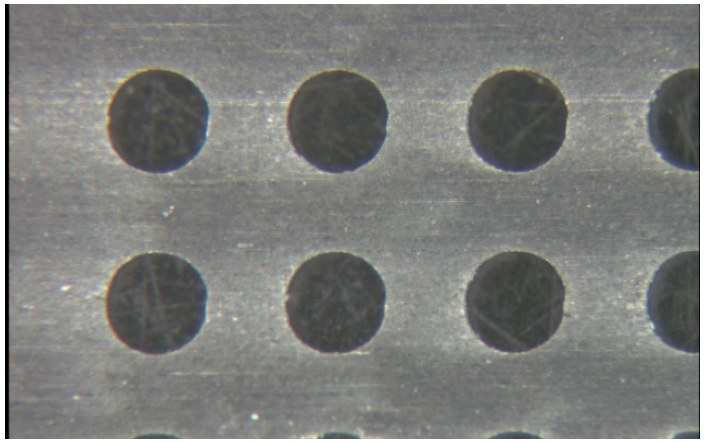
Stencil thickness typically dictated by other components on the PCB such as fine pitch leaded and small discretes

No minimum solder paste volume is typically required for PBGA since solder spheres completely melt and collapse during reflow

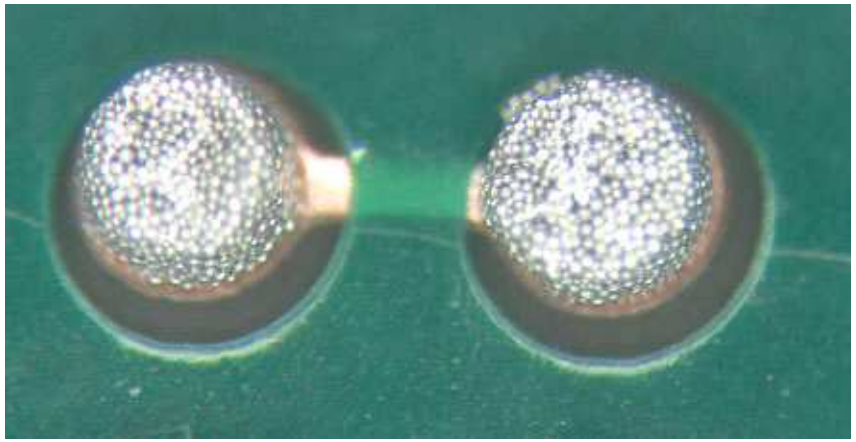
Apertures same as PCB pad diameters generally provide the best gasketing

In some cases, such as larger body size PBGAs, a thicker stencil may provide improved assembly robustness

± 0.025 mm solder paste print to PC board registration is common – paste should never touch any surrounding exposed metallization



Round Apertures for PBGA Printing



Wet Solder Paste on PBGA PCB Pads

Surface Mount Assembly of PBGA

High and low mass components should be carefully profiled with a thermocouple in a corner and inner sphere on a fully populated profile PCB

Most PBGA qualified to a maximum reflow temperature of 220℃ for SnPb and 260℃ for Pb-free

Soldering profiles are solder paste dependent so follow guidelines recommended by Solder paste vendor, but here are some guidelines that can be used:

- SnPb soldering:
  - Raise temperature of the joints to 100℃ at between 1.5 and 3.0℃/sec
  - Peak component temperature typically between 205 and 220℃
  - Desirable dwell time above 183℃ between 50 and 80 secs
- Pb-free soldering:
  - Raise temperature of the joints to 100℃ at between 1.5 and 3.0℃/sec
  - Peak component temperature typically between 235 and 245℃
  - Desirable dwell time above 217℃ between 50 and 80 secs

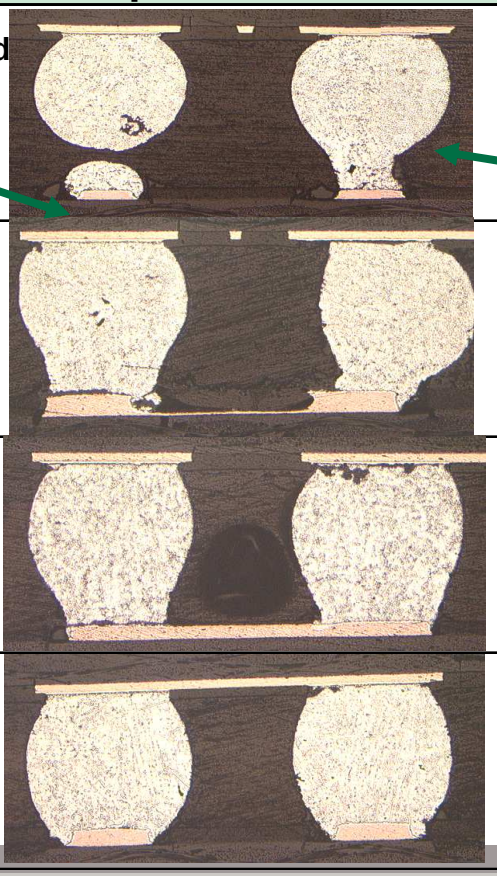
**Reference Application Note AN3300 for additional information

Soldering Pb-Free Spheres with SnPb Paste

Soldering Pb-free spheres with SnPb solder paste is not recommended by Freescale

If it is attempted, the entire Pb-free solder joint must reflow so that it collapses and mixes with the SnPb

Peak Temp (°C)	Comment	Example Solder Joints*
203	- No sphere collapse - No alloy mixing - Poor assembly yield (opens) - Early failures in temp cycling	
210	- Minimal sphere collapse - Minimal alloy mixing - Poor assembly yield (opens) - Early failures in temp cycling	
217	- Partial sphere collapse - Partial alloy mixing - 100% assembly yield on small sample - Consistent interconnect reliability	
225	- Complete sphere collapse - Complete alloy mixing - Consistent 100% assembly yield - Excellent interconnect reliability	



* Solder joints cross-sections shown are post-thermal cycling and show some evidence of typical fracturing

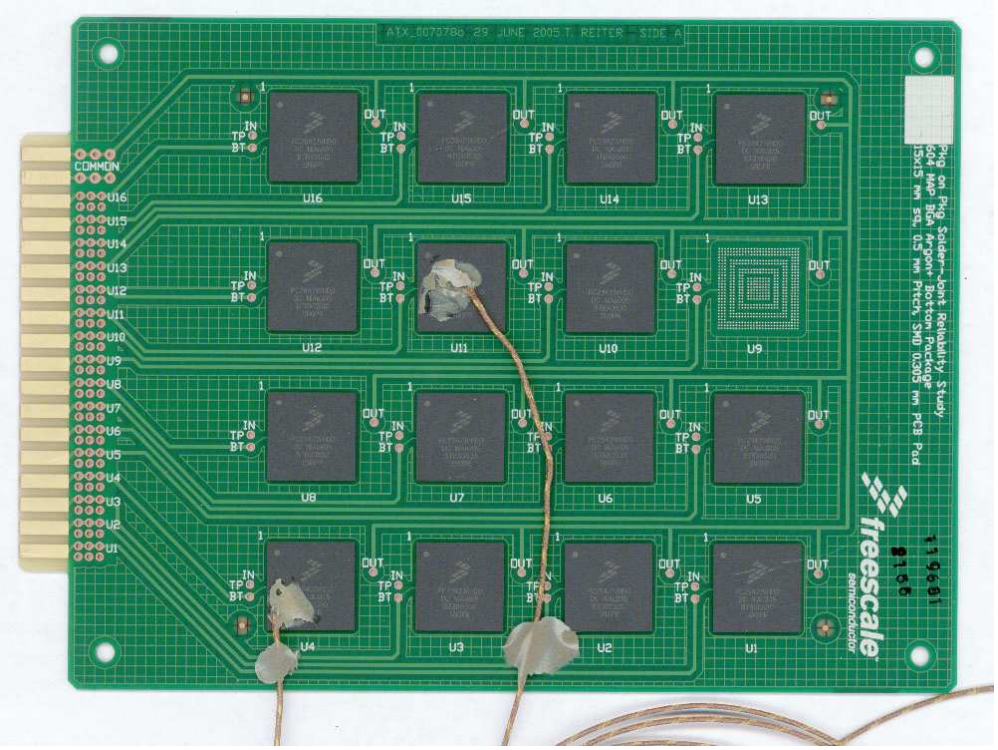
freescale™ semiconductor

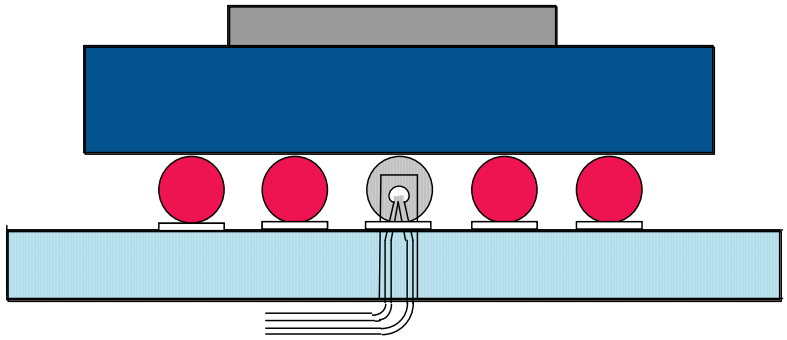
BGA Thermocouple Attachment

Profiling thermocouples should preferably be placed inside a corner and center sphere – alternately can be placed inside the package

Hole can be drilled from the top of the package or bottom of the PCB

Hole should be filled with thermally conductive high temperature epoxy





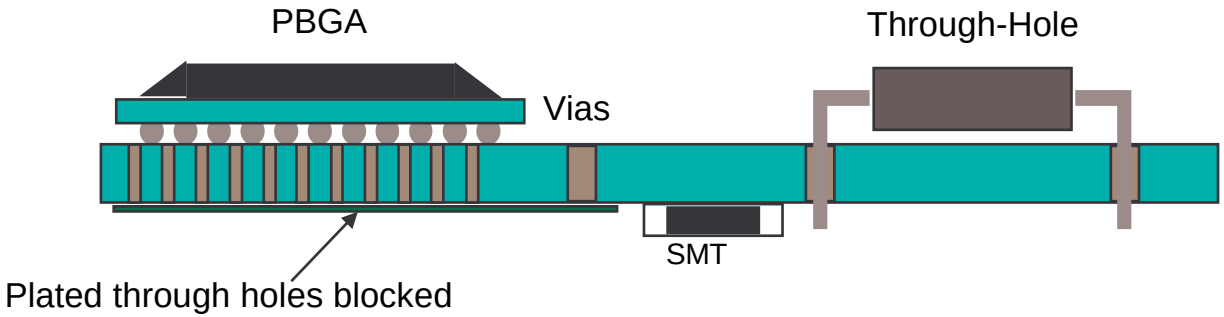
Thermocouple inserted through the top of hole drilled into a PBGA (left) and from the bottom of a PCB (bottom)

freescale™ semiconductor

Avoiding BGA Secondary Reflow

Wave soldering is often required after BGA has been soldered on the board. Secondary reflow of BGA solder joints in this operation must be avoided

- The result can be open BGA solder joints
  - PBGAs are low thermal mass packages (get hot fast)
  - The large number of plated through holes transmit heat rapidly
  - When heated from bottom only, PBGAs may warp
    - Upward bowing due to expansion of the PBGA substrate
    - Fully or partially molten solder joints separate



Mixed Technology Board

Avoiding BGA Secondary Reflow

There are many ways to avoid secondary reflow of BGA:

- Optimize the wave solder operation (temperature, PCB contact time) to minimize heat to the BGA
- Implement a wave solder pallet to block solder from BGA area
- Block heat transfer through vias around the BGA by filling or tenting with solder mask on wave side of board
  - May not be desirable from an in circuit test standpoint since it precludes using via pads as test points
- Apply polyimide tape under BGA area
  - Undesirable hand operation
- Redesign board to eliminate the wave solder operation

Presentation Outline

PBGA Introduction and Package Description

PC Board Design for PBGA

PBGA Assembly

PBGA Solder Joint Voids

Rework

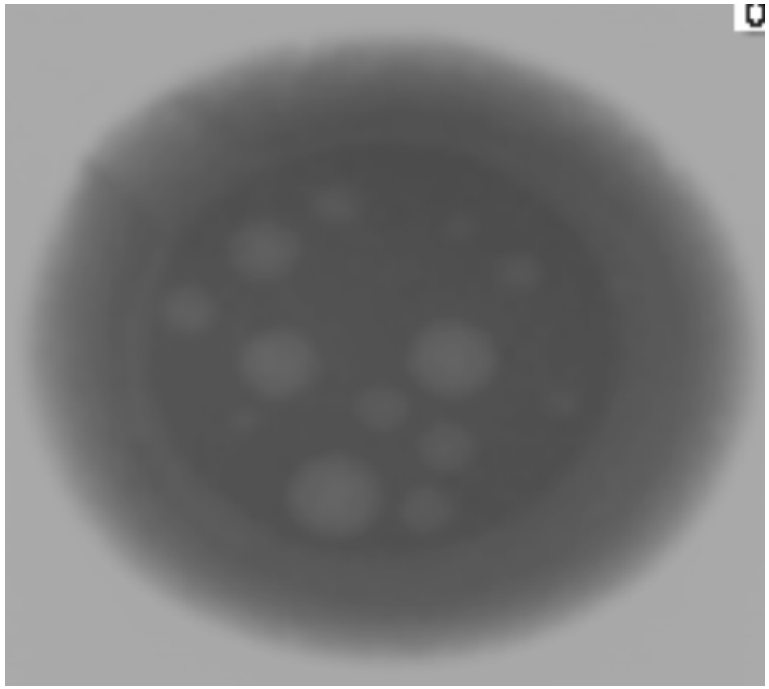
Solder Joint Reliability

Thermal Performance

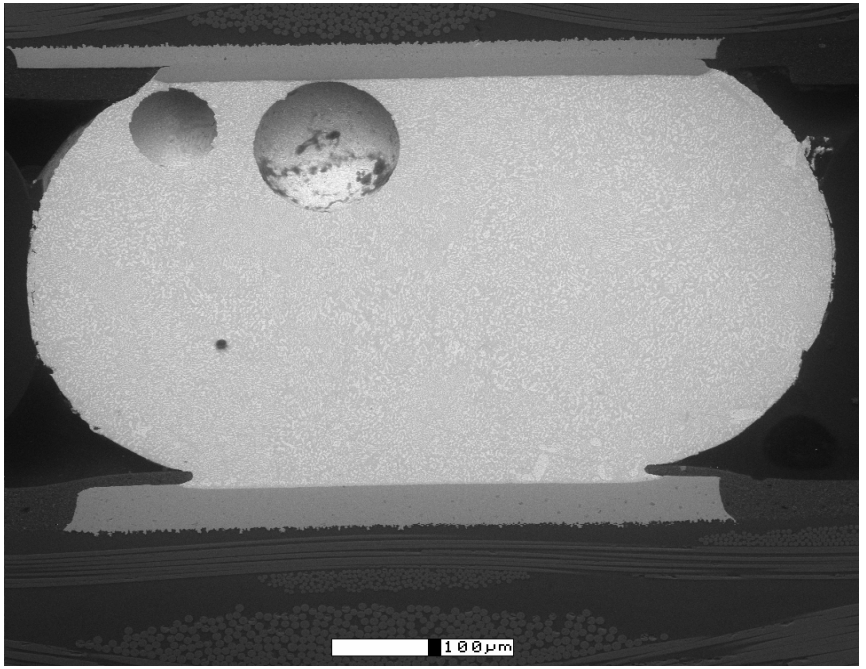
BGA Solder Joint Voiding

Voids are common in BGA solder joints after PCB assembly

- They are typically found near the package interface
- Voids are caused by entrapped flux from the solder paste volatilizing and rising in the molten solder joint due to buoyancy



Voids in X-ray



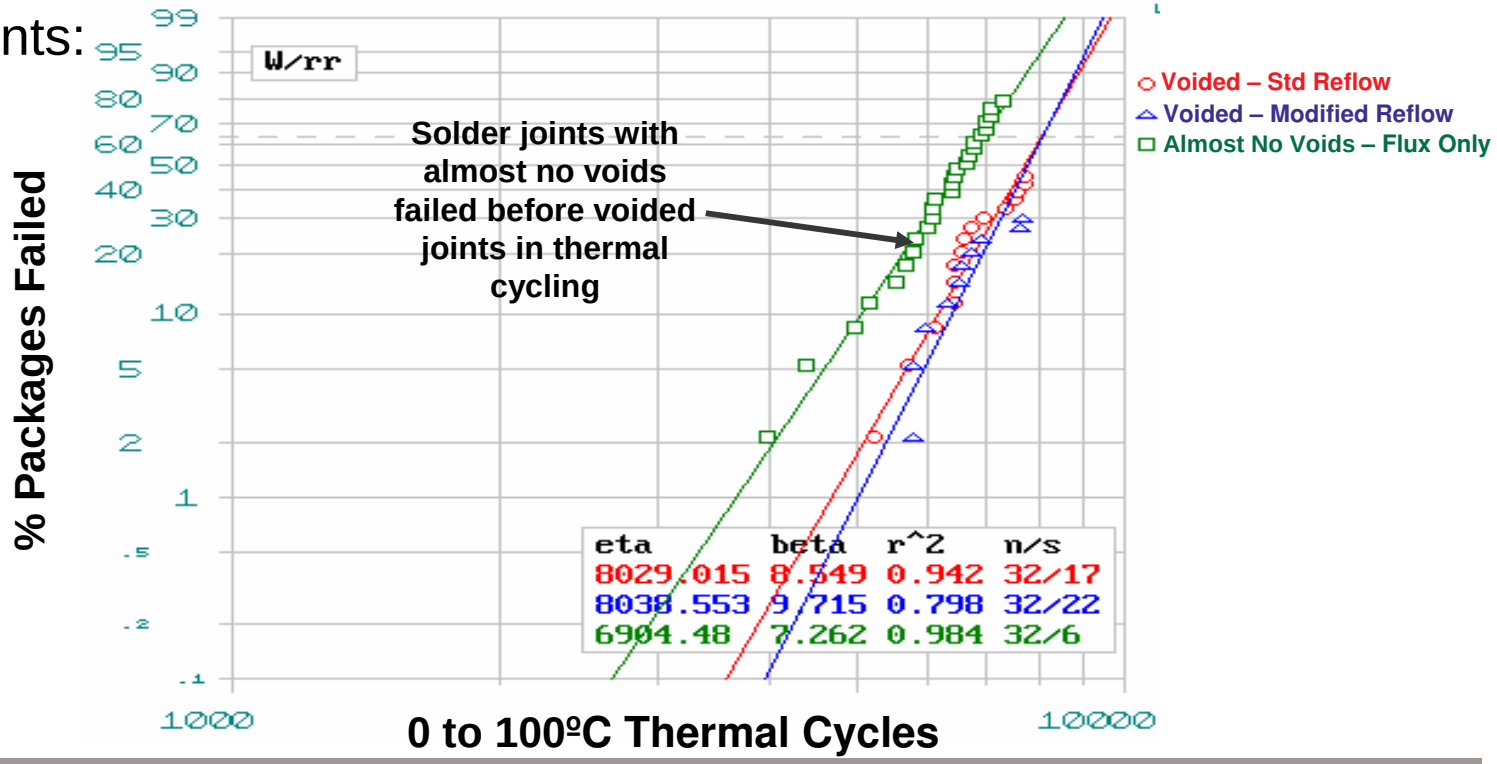
Voids in Cross-Section

BGA Solder Joint Voiding

The IPC-A-610D Standard[1] (Acceptability of Electronic Assemblies) categorizes >25% BGA joint voiding as a defect

Voiding may be reduced by many factors including selecting a solder paste with a low volatile flux and using a longer dwell time below liquidus

Some studies have shown voided joints to be just as or more reliable than non-voided joints:



1) Table of contents can be seen at: http://www.ipc.org/TOC/IPC-A-610D.pdf. Voiding is covered in section 8.2.12.5

freescale semiconductor

Presentation Outline

PBGA Introduction and Package Description

PC Board Design for PBGA

PBGA Assembly

PBGA Solder Joint Voids

Rework

Solder Joint Reliability

Thermal Performance

Avoiding PBGA Rework

Due to high assembly yields, PBGA rework should rarely be required

Key areas to focus on to maintain high assembly yield and avoid PBGA rework:

- Bare board quality - Ensure solderable pads with soldermask not encroaching onto the pads areas except the trace
- Solder paste printing – Avoid solder paste skips
  - Use 3D or visual paste inspection until the process is under control
- Solder reflow process - Carefully profile with a fully populated PCB and minimize the temperature delta between middle and corner PBGA joints
- Electrical test - Ensure that functional and in-circuit test programs do not falsely blame the PBGA as the cause for electrical failures

Key points when PBGA rework must be undertaken:

- The rework process goal is joint structure, quality and reliability that is at least as good as the initial assembly
- Removing and replacing a PBGA is easier than a QFP
- PBGA rework can be accomplished with off-the-shelf equipment
- The entire rework process must be carefully characterized

Re-use of reworked PBGAs after reballing is not recommended

freescale™ semiconductor

PBGA Detailed Rework Steps

Component Removal:

- First, the entire assembly must be baked for 4 hours minimum at the highest temperature that the most heat sensitive device on the PCB assembly can withstand (typically 80 to 125ºC)
  - This is to remove moisture from the PBGA so that it will not delaminate or popcorn during removal
- PBGA can typically be removed and replaced with topside heat only
- For thicker boards with many internal planes use additional localized bottomside heat
- To reduce cycle time and minimize board warpage, preheat entire board to at least 80℃ or higher, as permitted by the temperature rating of the other components that may be on the board
  - To minimize the number of heat cycles the PCB experiences, this pre-heat can be the bake-out described in the first step
  - Apply localized heat to PBGA package
  - Lift package immediately with a vacuum wand or tweezers when solder is molten

Excess Solder Removal from the PCB:

- Avoid an additional heat cycle by removing the excess solder from the PCB site immediately after package lift off, while board is still hot
- Can be removed quickly by a solder vacuum tool or solder wicking braid which requires heat and pressure and usually more time
- The goal is to end up with a solderable pad while avoiding damage the PCB or pads

PBGA Detailed Rework Steps - Continued

Re-Applying Flux to the PCB

- For a PBGA with collapsing spheres, no additional solder is needed - flux only
- Flux should be compatible with production cleaning strategy and applied uniformly but sparingly
- Re-applying solder paste may increase rework yield / reliability by accounting for PCB warpage

Placement and Reflow of Replacement PBGA

- Use the rework station or offline placement equipment
- As with removal, preheating of entire PCB may be advisable
- Characterize temperatures to provide uniform heat with ramp rates similar to reflow furnace
  - Non uniform heat and fast ramp rates can cause PBGA to warp or cause non uniform collapse of solder balls

Cleaning

- If cleaning is required, use the standard production cleaning method

Presentation Outline

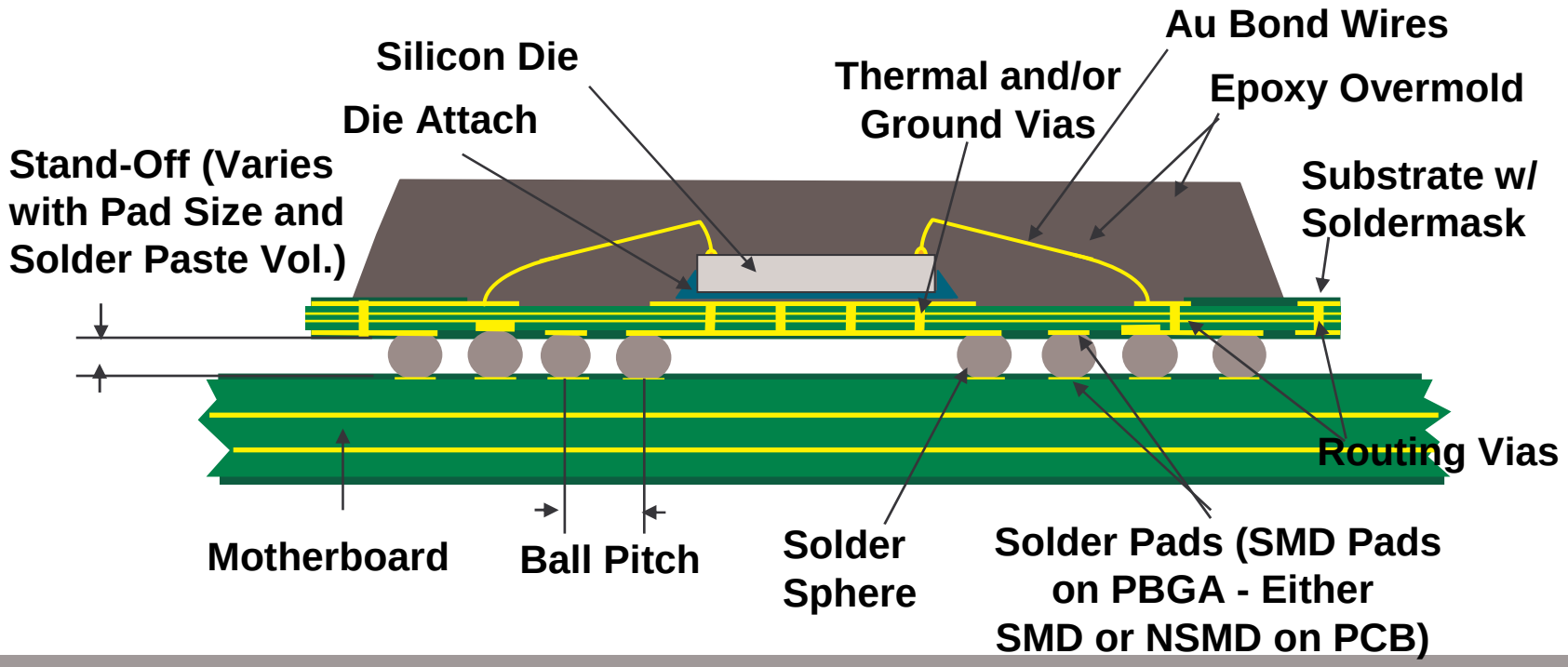
PBGA Introduction and Package Description
PC Board Design for PBGA
PBGA Assembly
PBGA Solder Joint Voids
Rework
Solder Joint Reliability
Thermal Performance

freescale semiconductor

PBGA Solder Joint Reliability

PBGA construction influences solder joint reliability

- For PBGA, silicon die mismatched to other package materials causes solder joint strain from differential package/PCB expansion during thermal excursions
- Solder joints under die edge usually fail first in accelerated thermal cycling



Solder-Joint Reliability Experiments

Purpose

- Determine life of package joined to PC Board
  - Solder-joints limit the life of final assembly

Availability

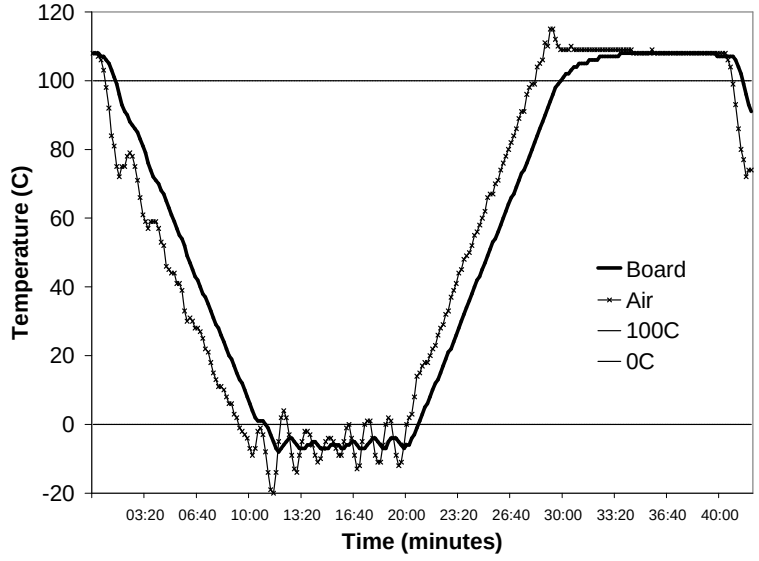
- Many new packages have data
  - Not all products will have data
    - Use existing data from similar package when available
- Experimental conditions
  - Air temperature cycling (ATC)
    - Automotive/industrial: -40/125C, 15 minute ramps and dwells
    - Telecommunications/networking/desktop: 0/100C, 10 minute ramps and dwells
    - Other conditions sometimes used for specific customer requirements
  - Daisy chain package and boards
    - Allow resistance to be monitored during cycling to detect failures
- Other Mechanical Tests
  - Monotonic Bend, IPC-9702, limited availability
  - JEDEC Shock, JESD22-B111, limited availability

Ask your customer sales representative for experimental data.

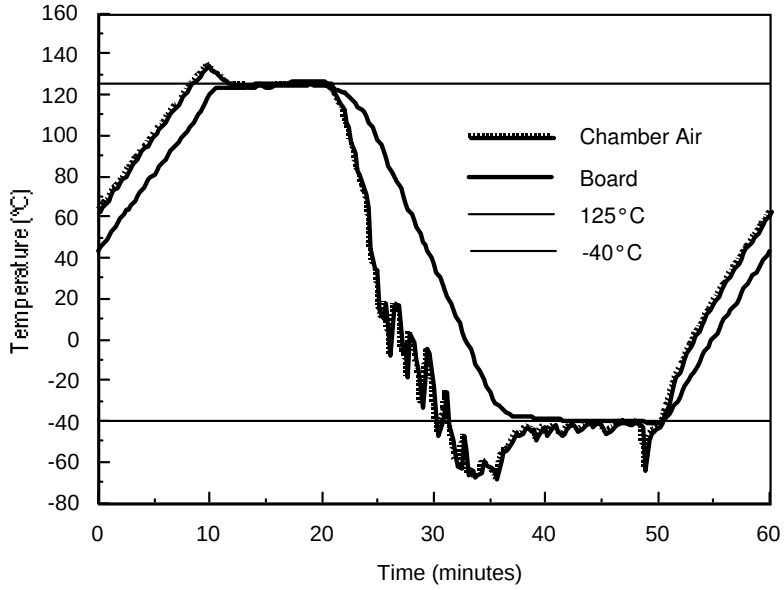
PBGA Solder Joint Reliability

Air temperature cycling is commonly performed on mounted PBGAs to assess solder joint integrity and suitability for intended applications

- Two common conditions are 0 to 100°C (networking, telecomm, computing) and -40 to 125°C (industrial, automotive)
- Testing typically continues until >50% of the packages failed to obtain the failure distribution



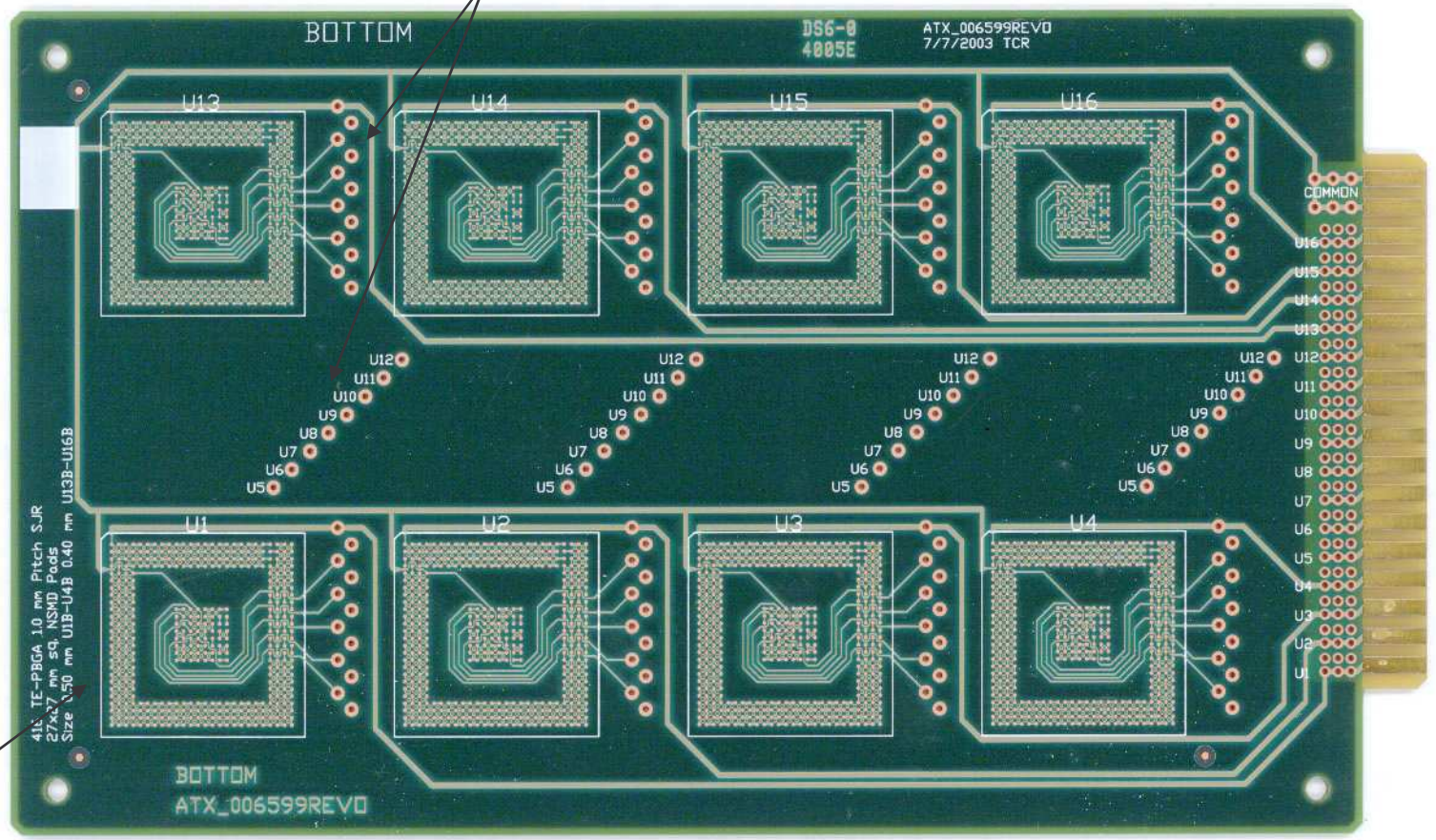
Typical 0 to 100°C Profile



Typical -40 to 125°C Profile

Example Solder-Joint Reliability Test Board

Connection points for specialized daisy chain testing

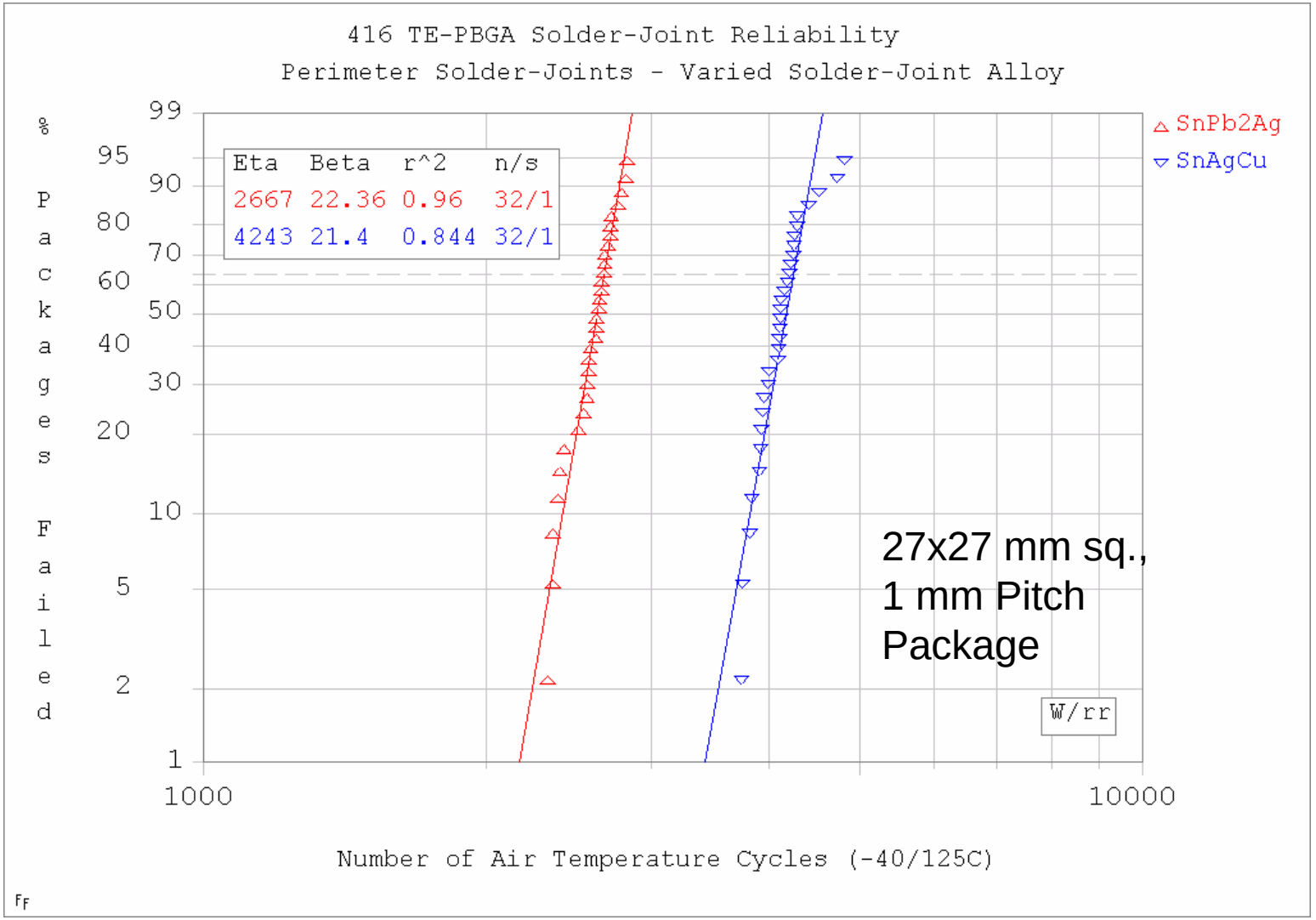


8 sites for daisy chain packages

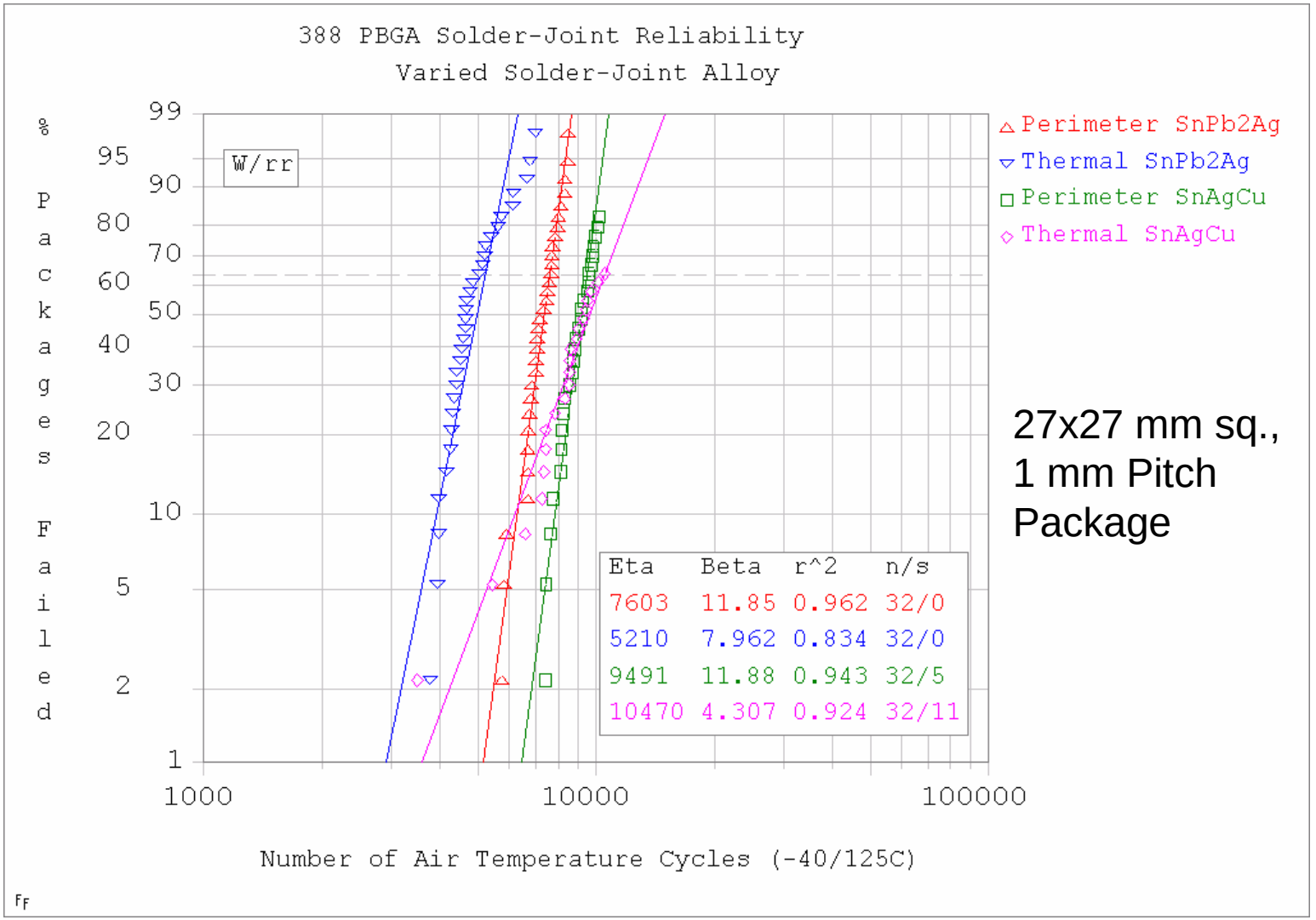
(Design follows guidelines of IPC-9701A.)

Edge fingers for monitoring

416 TEPBGA Solder Joint Reliability



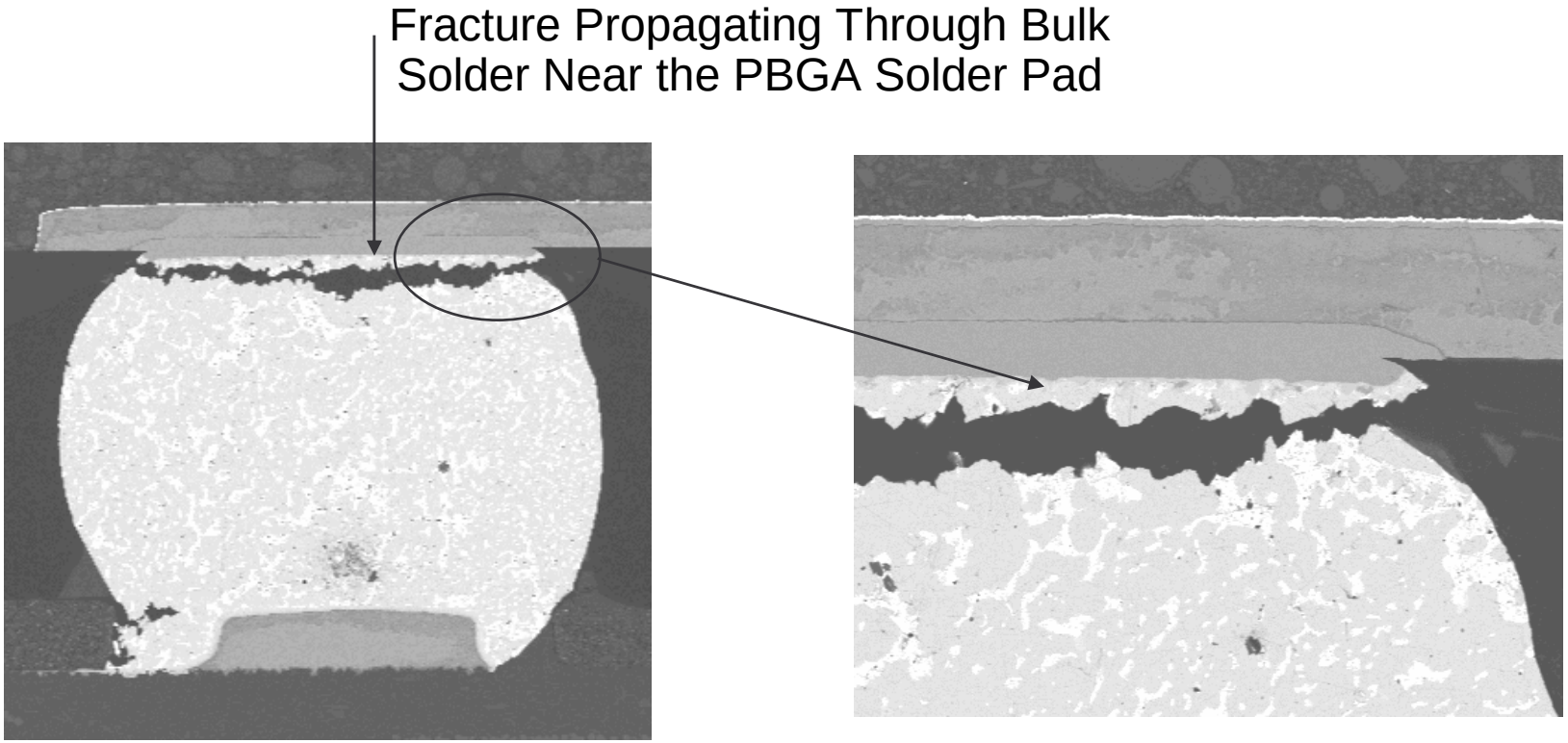
388 PBGA Solder-Joint Reliability



PBGA Solder Joint Reliability

Fractures in PBGA solder joints can initiate and propagate during extended board-mounted thermal cycling

Typical fractures are through bulk solder (not intermetallic)



Presentation Outline

PBGA Introduction and Package Description
PC Board Design for PBGA
PBGA Assembly
PBGA Solder Joint Voids
Rework
Solder Joint Reliability
Thermal Performance

PBGA Thermal Performance

Thermal performance is reported as thermal resistances determined per the JEDEC JESD51 series of specifications.

PBGA packages typically have good thermal connection to the printed circuit board. Their performance will depend on the board construction, the thermal connection to the power and ground planes in the board, and other local heat sources.

To highlight the effect of the local environment, thermal resistances are typically provided on both the single layer board and the 2s2p board (with two signal and two power planes). The planes are 1 oz copper. It is not unusual to see nearly a factor of two difference between the thermal resistances on the two boards.

A good summary description is JESD51-12, “Guidelines for Reporting and Using Electronics Package Thermal Information”

The next slide provides a cross reference to the various thermal resistances

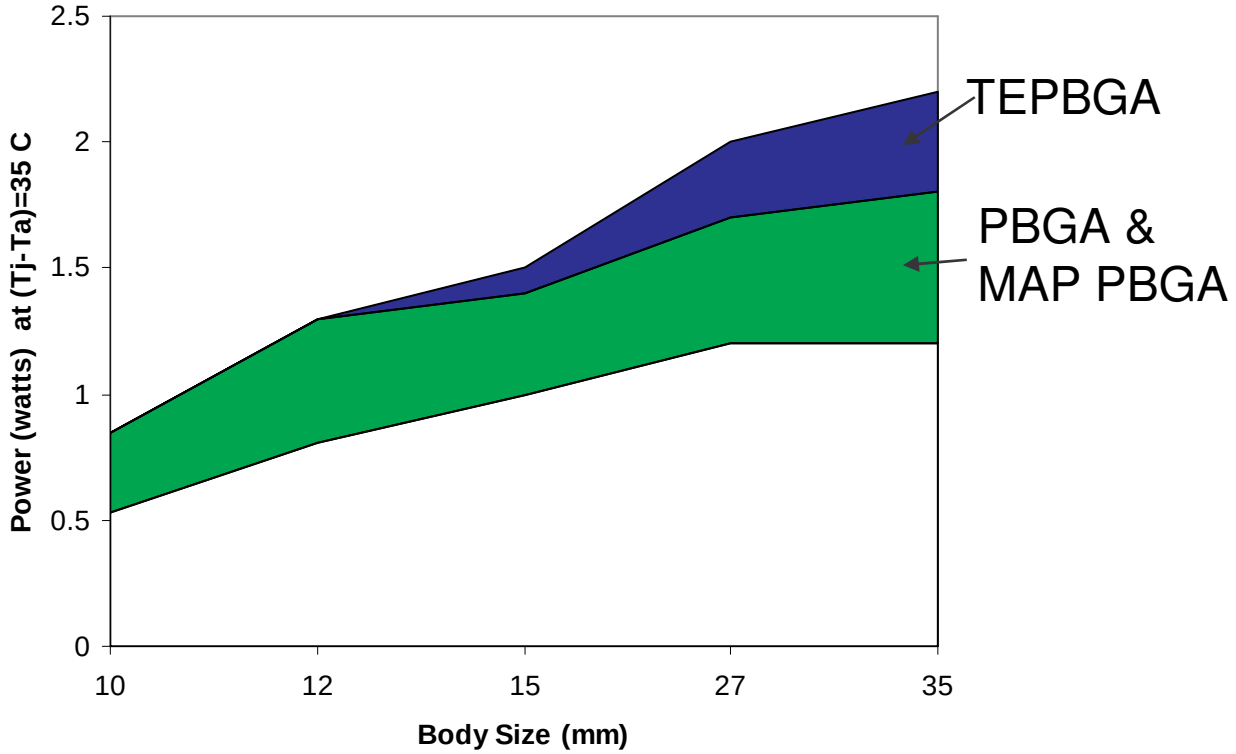
Summary of Thermal Resistances

Thermal Metric	Symbol	Condition	Usage	Specification
Junction to Ambient	$R_{\theta JA}$	Single Layer Board	Tightly packed array of devices	JESD51-2, JESD51-6
Junction to Ambient	$R_{\theta JA}$	Four layer board (2s2p)	Most commonly quoted thermal resistance	JESD51-2, JESD51-6
Junction to Board	$R_{\theta JB}$		Customer use in modeling or when board temperature known	JESD51-8
Junction to Case	$R_{\theta JC}$		Heat sinks	MIL-SPEC883, Method 1012
Junction to Package Top	Ψ_{JT}	Natural Convection	Determine junction temperature from thermocouple reading	JESD51-2

Note: PBGA packages are tested on a board specified in JESD51-9
Jedec JESD specs are available at www.jedec.org

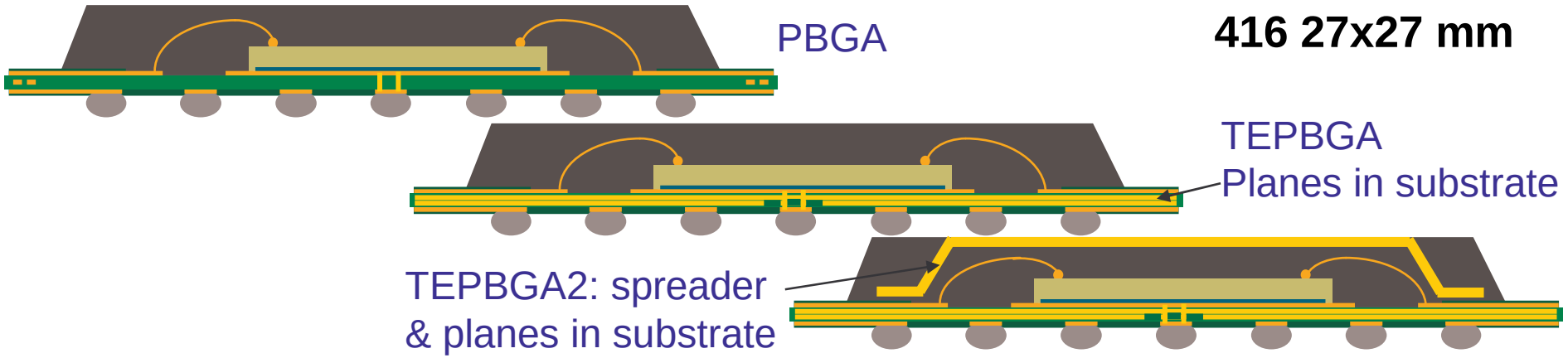
Typical PBGA Power Capabilities

$$R_{\theta JA} = (T_J - T_A)/P \Rightarrow P = (T_J - T_A)/ R_{\theta JA}$$



Assumptions: Multilayer application, lots of board area, only low power devices nearby, used junction to ambient thermal resistance on 2s2p test board in natural convection with maximum junction temperature 35 C higher than ambient temperature. Range in thermal performance caused by die size and substrate design features.

Thermal Data for Three Package Configurations



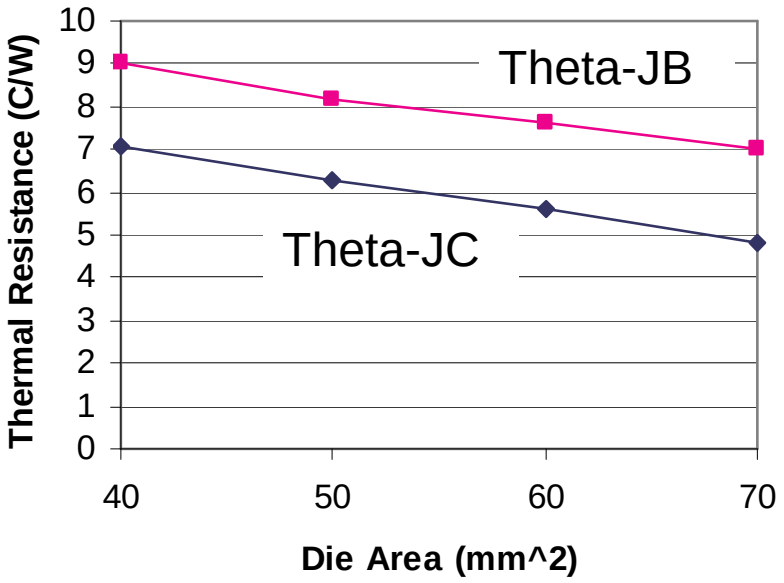
Thermal Metric	Test Board	Symbol	PBGA	TEPBGA	TEPBGA2	Unit
Junction to Ambient Natural Convection	Single layer board (1s)	$R_{\theta JA}$	57	31	25	°C/W
Junction to Ambient Natural Convection	Four layer board (2s2p)	$R_{\theta JA}$	28	21	17	°C/W
Junction to Ambient (@200 ft/min)	Single layer board (1s)	$R_{\theta JMA}$	49	25	19	°C/W
Junction to Ambient (@200 ft/min)	Four layer board (2s2p)	$R_{\theta JMA}$	25	18	14	°C/W
Junction to Board		$R_{\theta JB}$	21	13	9	°C/W
Junction to Case		$R_{\theta JC}$	13	9	7	°C/W
Junction to Package Top	Natural Convection	Ψ_{JT}	3	2	7	°C/W

Effect of Die Size

Thermal performance depends on package construction, package internal design, and die size.

Theta-JC decreased 30% as die area increased by 75% for this 416 27x27 mm TEPBGA2.

416 27x27 TEPBGA2



Summary: PBGA Thermal Performance

Thermal performance is typically available in the Data Sheets for the product

- For Networking products, it is in the “hardware specification,” for other products it will be in the product manual or datasheet. Data is usually in the “electrical specifications” section of the manual if there is not a separate thermal section.
- Thermal performance depends on die size and package design. Data should be obtained specific to the device.

Thermal performance is determined per the JESD51 series of specifications with the PBGA mounted to a board. A good summary description is JESD51-12, “Guidelines for Reporting and Using Electronics Package Thermal Information”. These specifications are available at http://www.jedec.org.

Temperature predictions using $R_{\theta JA}$ have potential errors of 2X because of uncertainty in ambient conditions and effect of other power sources.