Advanced Rework Technology and Processes for Next Generation Large Area Arrays, 01005, PoP and QFN Devices

Brian Czaplicki
Director, Technical Marketing Programs
Air Vac Engineering Co., Inc
Seymour, Connecticut

Abstract
BGA Rework is now largely mature, although new supplemental processes that provide improved process control such as Solder Paste Dipping and Non-Contact Site Cleaning can now be integrated into existing processes if the rework technology that is used allows.

So what are the next set of challenges that will need to be addressed in regard to Area Array and SMT Rework? The International Electronics Manufacturing Initiative or iNEMI has recently published its 2013 Technology Roadmap for the global electronics industry which includes a section dedicated specifically to rework and repair. Of particular interest and importance is iNEMI's gap analysis which identifies future specific gaps and challenges that will result from such factors as government regulations, disruptive technologies and new product requirements.

This paper will review five of the key rework gaps and challenges identified by iNEMI including:

1) Reworking very large, next generation area arrays on large high thermal mass assemblies.
2) Development of hand soldering processes for reworking 01005 components.
3) Development of industry-standardized processes for reworking Package-on-Package (PoP) devices.
4) Development of industry-standardized processes for reworking Quad Flat, No Lead (QFN) devices.
5) Development of site redressing processes that prevent lifted pads, solder mask damage and copper dissolution [(1)]

The objective of this paper is to discuss the five iNEMI rework gaps and challenges including identification of the key technical/process challenges, outlining in detail the efforts-to-date aimed at addressing these new challenges as well as the next steps required for complete resolution of these challenges.

Introduction
The five major rework gaps and challenges identified in the rework and repair section of the 2013 iNEMI Technology Roadmap requires a combination of new thinking, new and innovative technology and a lot of good old fashioned hard work.

BGA Rework Systems, which have successfully handled the requirement to rework lead-free BGA's for ten plus years will now be required to handle a number of new technical challenges including the ability to align next generation extra large (70-100mm), high I/O (>1000) components as well as having the thermal capacity to reflow these large components while meeting the current stringent standards for both maximum package temperature and the maximum joint temperature Delta which are based on much smaller, current generation components. Other machine/process challenges related to next generation large devices will include such "simple" things as vacuum-holding capability as well as more complicated issues such as component warpage and site preparation. In addition, these XXL components will most often be found on very large, high thermal mass assemblies for server, telecommunications and networking applications which means that the BGA Rework System must also have the capability to hold and preheat these very large, high thermal mass assemblies.

BGA Rework Systems will also be required to handle new challenges associated with PoP Rework including controlled solder paste/flux dipping, the accuracy to place 0.3-0.4mm pitch devices and the force control necessary to place a dipped top device on an unreflowed bottom device so both packages can be reflowed together. QFN rework requires the integration of pasted or pre-pasted and reflowed devices as well as the process-know how to control voiding in the center ground pad area.

The rework industry's reluctance to replace wick-based site redressing with non-contact methods is beginning to change. The industry's view of solder wick as a fast and easy solution for preparing a site must be tempered by the fact that wick-based cleaning lacks process control, thereby subjecting the PCB to a higher potential incidence of pad or solder mask damage.

The pros and cons of various flux/paste application methods including dipping, paste-on-device, paste-on-board, stay-in-place stencils and multi-up stencils are reviewed and summarized.

Finally, the topic of 01005 rework is analyzed in detail including the potential use of hand tools, BGA Rework Systems and combined (man/machine) technology systems.

Next Generation Large Area Array Rework

Some of the largest Area Arrays in use today include CCGA's, as well as BGA sockets and connectors with lead patterns approaching 50mm. iNEMI forecasts component sizes of 60-75mm, however future rework requirements for 80-90mm components have already begun to surface.

These next generation large area arrays may have high thermal mass issues such as metal BGA sockets. Further complicating this issue is the fact that these large, thermally challenging devices will typically be found on large, high thermal mass PCB's.

iNEMI's major concern is the ability to reflow these next generation large devices on large, high thermal mass PCB's based on the current stringent specifications as shown in Table 1.

Table 1: Large Area Array Rework [2]

Soldering Process	Parameter	Units	2011	2013	2015	2017	2023
Pb-free	Maximum package sizes	mm	50	50	55	60	75
	Maximum package temperature (dependent on component body size)	°C	245-260	245-260	245-260	245-260	245-260
	Target solder joint temperature	°C	235	235	235	235	235
	Target delta T across solder joints	°C	<10	<10	<10	<10	<10
	Time Above Liquidus (TAL)	Sec	60-90	60-90	60-90	60-90	60-90

The challenge will be that next generation area arrays will increase in size by 50-100% and will typically be found on large, high thermal mass PCB's, however the reflow specifications are the same as for today's standard BGA's.

Large Area Array Thermal Profiling

A 114mm (4.5") square BGA with 10,000 I/O was thermally profiled. Seven (7) thermocouples (TC's) were used to instrument the test vehicle. Six (6) TC's measured joint temperature and one measured package temperature. The PCB was the same size as the device so preheating the joints to 150°C represented the board preheat stage.

Significant development effort was done in regard to optimizing the thermal distribution of the nozzle required for this very large device. The nozzle design is proprietary at the time of this writing and therefore cannot be photographed or illustrated in detail.

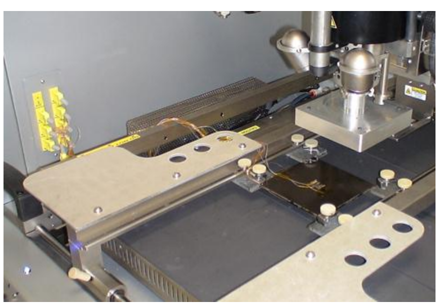


Figure 1. Instrumented 114mm BGA on BGA Rework System

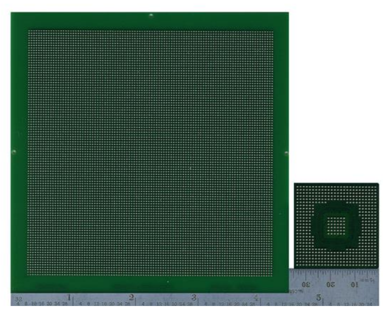


Figure 2. 114mm BGA (10,000 I/O) with 35mm BGA shown for scale

This extremely large device required significant thermal energy and time to reflow it. The PCB was preheated to 150°C with a high power, indirect IR bottom heater. The top heater setting after preheat was 495°C at 2.5 scfm flow for 150 seconds. The top heater setting of 495° is very high compared to the 300-325°C setting typically used for BGA's. However the large size of the component and the nozzle required significantly more thermal energy. A Board Cooling System was used to cool the part down after reflow which is critical to achieving targeted Time Above Liquidus (TAL). The results are shown in Table 2.

Table 2: 114mm BGA (Ph 150°C, 495C @ 2.5 scfm, 150 sec)

		Max Temp (°)		Delta T (°)		TAL (sec)	
TC #	Location	Actual	Spec	Actual	Spec	Actual	Spec
1	Top of Package	260	260		-	-	-
2	Joint (corner)	235	235	6	< 10	52	60-90
3	Joint (corner)	235				55	
4	Joint (corner)	239				72	
5	Joint (corner)	238				68	
6	Joint (corner)	237				69	
7	Joint (corner)	241				75	

All specifications were met except the TAL for two joints fell short of the target. The volume of air used to cool down the component can be easily reduced to correct this. However the bigger issue is that the maximum package temperature was 260°C, which leaves no margin for error.

A modification was made to the rework system that allowed significantly higher flow rates to be used. The thermal energy transferred to the component is a combination of temperature and flow so the higher the flow rate, the lower the temperature required. A low temperature/high flow approach typically reduces package temperature, however it can negatively impact the delta T of the joints if the flow is too high, so the optimum balance of temperature and flow must be found for each application.

Increasing the gas flow from 2.5 scfm (current maximum) to 3.0 scfm (25% increase) allowed the nozzle heater temperature to be reduced from 495°C to 350°C (30% reduction). The results of the temperature and flow modification are shown in Table 3.

Table 3: 114mm BGA (Ph 150°C, 350C @ 3.0 scfm, 167 sec)

		Max Temp (°)		Delta T (°)		TAL (sec)	
TC #	Location	Actual	Spec	Actual	Spec	Actual	Spec
1	Top of Package	250	260		-	-	-
2	Joint (corner)	240	235	6	< 10	75	60-90
3	Joint (corner)	240				78	
4	Joint (corner)	241				92	
5	Joint (corner)	241				88	
6	Joint (corner)	235				76	
7	Joint (corner)	239				86	

The lower temperature/higher flow approach reduced the maximum package temperature from 260°C to 250°C, well below the current guidelines with no negative impact on the delta T of the joints. All joints except one met the TAL target of 60-90 seconds using this approach.

It is important to note that the 114mm BGA is attached to a 114mm test board with no layers. The thermal settings required to reflow a device of this size on a large, high thermal mass assembly will be significantly higher. It will be critical for the BGA Rework System to have the nozzle temperature/flow and board heating/cooling/handling capability required to rework these next generation large devices on high thermal mass assemblies (Figure 3).

Next generation large component technology will also provide additional challenges not cited by iNEMI including alignment capability, warpage issues, component/site preparation, large board handling capability and nozzle vacuum requirements. It is anticipated that the vision system on rework machines of the future must be able to align components at least up to 75mm and perhaps as large as 100mm or more (Figure 4). Warpage/coplanarity issues will be a major issue for next generation large devices, especially as the pitch decreases over time. Non-Contact Site Cleaning technology must include large size nozzles that provide safe, effective and fast cleaning of the residual site solder. Finally, the vacuum capability of the nozzle must be sufficient not only to hold large devices in place, but also to lift and remove them out of reflowed solder.

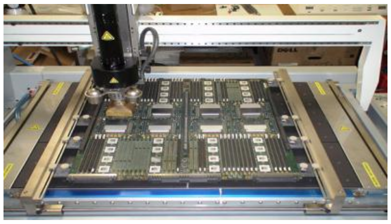


Figure 3. 22x24 high thermal mass PCB on large board rework system (board courtesy of IBM)

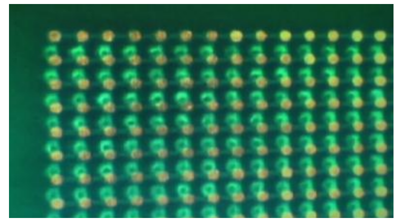


Figure 4. 114mm BGA Corner Viewing in Vision

One major roadblock to the development of effective rework technology and processes for next generation large devices on thermally challenging assemblies is access to and the cost of these devices and assemblies. Cooperative effort is needed between OEM's/CM's and rework equipment suppliers in this area to address the critical issues outlined above.

01005 Rework

Small enough to pass through the eye of a needle and multiple times smaller than a pepper flake, 01005's are nearly invisible to the human eye. These microscopic devices pose significant rework challenges including component handling, site preparation and reflow.

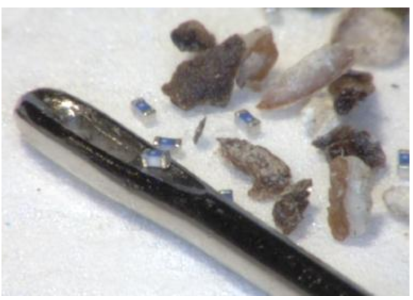


Figure 5. 01005 in eye of Needle and Next to pepper flakes

01005's are not yet used in widespread high volume production, however it is believed that current users are performing hand soldering rework when required. iNEMI's view is that hand soldering rework of 01005's is difficult but possible for a skilled operator.

These are conflicting industry views regarding 01005 rework with the majority of views indicating that 01005's can not be reworked. On the other hand, manufacturers of both hand tools and BGA rework equipment indicate that they have equipment capable of reworking 01005's. Web based research found several hand tools said to be capable of 01005 rework, however, no hand tool-based video of 01005 rework was found. On the other hand, several 01005 rework videos were found using BGA rework machines, but many have limitations or issues including high capital cost, low throughput, lacking ease of use, and the ability to rework 01005's with adjacent spacing of 0.2mm or less.

Another issue is the inability of most machine-based systems to handle almost half of the cases where the residual site solder can not be reused due to paste printing defects such as insufficient solder (Figure 6). Some rework machines do provide paste dispensing capability, however the complexity and efficiency of dispensing microscopic dots in a repeatable fashion in a hot rework environment is a major concern.

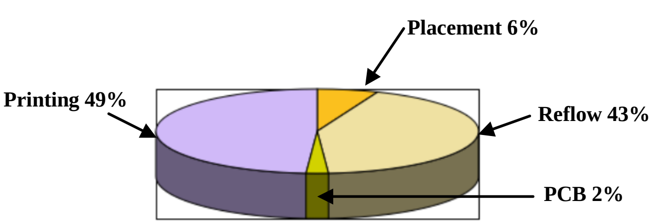


Figure 6. Root Cause of Defect [3]

One other alternative for 01005 rework is to combine the best features of hand soldering and BGA-type machine rework into a "Man" (ie: man or woman)/Machine Rework Interface (MMRI). In this approach, the operator has manual control of all processes which are done directly at board level without using slower, more expensive and often complicated beamsplitter-based systems. The board level rework approach also eliminates "Z" axis accuracy issues that exist with most beamsplitter-based systems. If the beamsplitter is not calibrated accurately and often, the nozzle will not properly contact the 01005 during removal, and the positioning will be off during placement. In addition, some BGA Rework Systems do not have the accuracy necessary for placing micro-discretes regardless of how often the vision system is calibrated. Remember that we are talking about microscopic devices, so minor placement errors which had no impact in the past on BGA rework, now become significant.

Figure 7. “Man”/Machine 01005 Rework Interface (MMRI)

In addition to manual control, the MMRI approach also provides the operator with numerous machine-related advantages, including inspection quality microscope-based optics, integrated top and bottom heating technology and perhaps most importantly, elimination of the precise manual dexterity required for hand soldering rework.

The 01005 removal throughput with the MMRI approach is multiple times faster than beamsplitter-based systems, and as fast as hand tools.

The 01005 replacement process is greatly simplified when the residual solder on the pads can be reused. A stereo microscope with high magnification zoom lens assists the operator in picking the replacement device from the tape holder, eliminating any manual handling of the part. The site is fluxed with a micro syringe to minimize over application of flux which is a key iNEMI concern. Indium 30B halide-free flux is used as it does not contribute any ionic species which can create a conductive pathway for dendritic growth that can cause electrical failure if not properly heat activated. Unlike hand soldering, the MMRI process includes full board preheat which should eliminate any flux non-activation concerns however the use of halogen-free flux is recommended as an additional safeguard.

Alignment is done easily and quickly at board level using the inspection-quality stereo microscope with high magnification zoom lens, fine adjust X/Y Table and Theta Rotational Adjustment. Another advantage of the MMRI process is that a misaligned 01005 can be removed, aligned and replaced in a single step.

The 01005 replacement process becomes far more complicated when the residual solder can not be used. First, any solder remaining on the pads must be removed with a micro-site cleaning nozzle, which provides both heat and vacuum. The Board Cooling System and the nozzle cool air bypass are activated and continue until the board temperature drops to 70°C. The Board Cooling System is powerful and the PCB’s that 01005’s will be used on will have low thermal mass so the cooldown will occur quickly. A proprietary micro-dipping tool is used in conjunction with a precision depth dip tray to transfer solder paste to the pads. The stereo microscope with zoom lens allows the operator to view the paste application process and to inspect the paste prior to proceeding. 01005 rework throughput can be increased by using a batch approach where all 01005 defects are removed and site cleaned and then all sites are pasted and replaced separately.

The nozzle is heated at low temperature and the replacement component is picked from the tape pocket. Static electricity and paper dust are two tape-related concerns. The replacement device is aligned with the pasted pads, placed and reflowed. Force feedback is provided to the operator during manual placement, which is important as assembly-based placement testing indicates that placement force in excess of 2 newtons (200 grams) can cause component cracking. Another significant advantage of the MMRI System compared to both hand soldering and BGA Rework Systems is that after the replacement component is placed on the pasted pads, the operator can retract the nozzle slightly and reflow the device with convective heating. This allows the 01005 to self center which is not possible when the device is held in place with a conductive heating tip. Using the IR bottom heater in conjunction with the convective heating nozzle provides the gradual ramp up desired for replacing ceramic capacitors which cannot be accomplished with conductive heating tips. Nitrogen is recommended for the convective reflow process to improve wetting and reduce oxidation of the fine grain solder particles.

- Tip Contact Time : 7 seconds
- Heating Slope : 13° / second
- Max 01005 Temp : 242°C
- Heating Cycle : 7 seconds

01005 Removal (Combined Conductive / Convective Heating)

- Tip Contact Time : 0 seconds
- Heating Slope : 2° / second
- Cooling Slope : -5° / seconds
- Max 01005 Temp : 242°C
- Heating/Cooling Cycle : 60 seconds

01005 Replacement (Convective-only Heating)

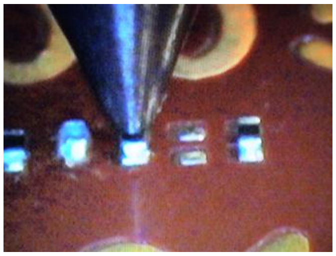


Figure 8. 01005 Micro-Tip Nozzle

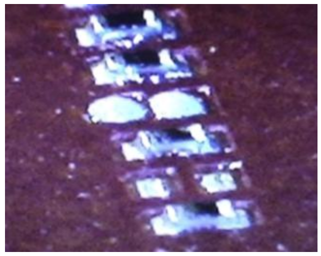


Figure 9. Paste Transfer onto 01005 Pads

Figure 10. 01005 Replacement (left) Site Cleaned Pads (right)

The cleanliness of the micro-tip is critical to insure consistent vacuum removal of these microscopic devices on an ongoing basis. The operator periodically uses an ultra-fine gauge cleaning tool to keep the micro-vacuum tip clean.

Package on Package (PoP) Rework

PoP is two or more fine pitch components stacked on top of one another in an effort to save board space. The bottom package is typically a high performance logic device and the top package is typically a high capacity memory device. Warpage of the bottom package caused by the CTE mismatch between the die, molding compound and substrate is by far the most common PoP issue. Warpage is a key issue for PoP due to the fact that the packages are extremely thin and typically fine pitch. PoP warpage typically manifests itself as a Head-in-Pillow (HiP) defect, which is defined as the incomplete coalescence of the solder joint between the PoP sphere and the printed solder paste. Other contributors to HiP defects include flux exhaustion, poor wetting and incorrect solder paste chemistry.

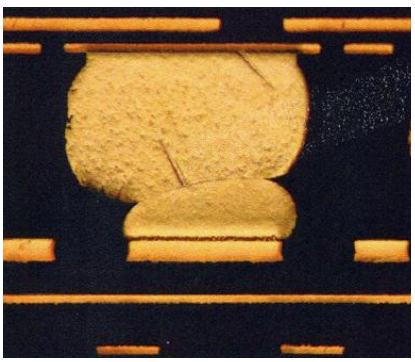


Figure 11. Head in Pillow Defect [4]

PoP Rework mimics standard BGA Rework with a few modifications. PoP devices can either be removed separately with a standard nozzle that requires zero clearance (Figure 12) or together with a vacuum-activated tweezer nozzle, which requires some adjacent clearance (Figure 13). Some companies glue the two packages with adhesive so they can be removed together with a standard nozzle. PoP removal should not include any downward nozzle pressure during the removal process as this can result in solder ball migration to adjacent components. Vacuum-based component removal using a vacuum sensor accomplishes this task.

Figure 12. Standard Nozzle (Removes 1 Package at a time)

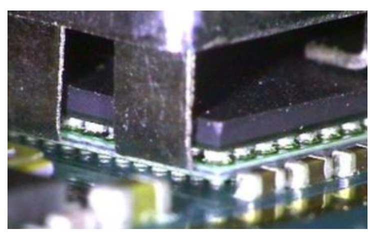


Figure 13. Tweezer Nozzle (Finetech) (Removes both Packages together)

After the components are removed, the residual site solder must be removed. The "stone age" practice of removing the residual solder with a soldering iron and wick should be replaced by non-contact solder removal to eliminate the potential for lifted pads and solder mask damage. The cautionary verbage associated with using solder wick (ie manual site dressing is very dependent on operator skill, damage often occurs when operators do not tin the bit, do not apply pressure to the pads, the speed of the wicking process is critical, etc. etc.) should serve as a wake-up call to companies who continue to allow wick-based site cleaning to be used because their operators prefer it or because it is fast. Higher lead-free reflow temperature and the continued drive toward finer pitch devices should signal the end of wick-based cleaning, however iNEMI estimates that wick is still currently used 80% of the time for site preparation. Every operator can detect a damaged pad and knows how to fix it, however determining if the solder mask has been damaged by the solder wicking process is far more difficult.

Some BGA rework equipment manufacturers' claim to have a non-contact site cleaning system, however they really have a heated nozzle with a metal vacuum tip that is moved across the site with manual "x", "y" and "z" controls. A true non-contact site cleaning tool incorporates a vacuum sensor that automatically and continuously adjusts the vacuum tip height. The most advanced systems go one step further by using a high temperature composite vacuum tip rather than a metal tip to completely eliminate the possibility of a heated metal tip contacting the board or the pads (Figure 14).

Users are demanding faster non-contact site cleaning solutions in order to switch from solder wick. Some BGA rework manufacturers' have already developed larger, site specific cleaning tools designed to clean the site in a single pass (Figure 15).

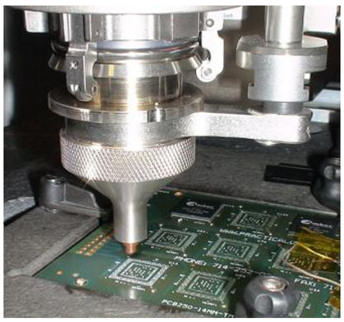


Figure 14. Non-Contact Site Cleaning System

Figure 15. Large Site Cleaning Nozzle (Zevac AG)

PoP replacement is more complicated than replacing a BGA. First, the bottom package is picked and is typically dipped in a controlled volume of flux or dippable solder paste, using force control to insure repeatability. Ideally, the dipping process can have multiple programmable dip locations so that the dip tray does not have to be re-prepared after each process (Figure 17). Dipping the bottom package in solder paste helps to address the warpage/HiP issue discussed earlier. Using a dippable solder paste chemistry with a higher temperature activation will help reduce flux exhaustion and improve wetting. If the beamsplitter has independent top and bottom lighting, camera zoom and the ability to move to all areas of the component, the solder paste on the spheres can be inspected during alignment in vision (Figure 18). This allows the operator to terminate the process if the paste has bridged or is missing from any spheres.

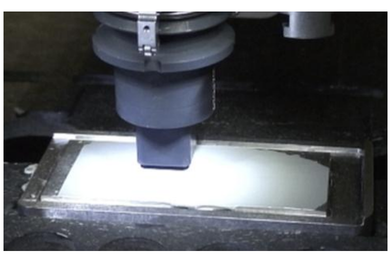


Figure 16. Force-Controlled Solder Paste Dipping

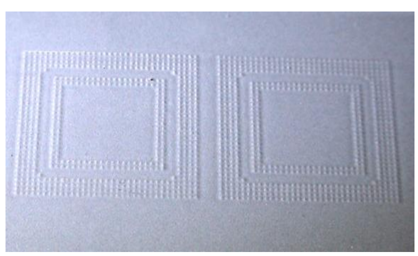


Figure 17. Multiple Imprints in Solder Paste

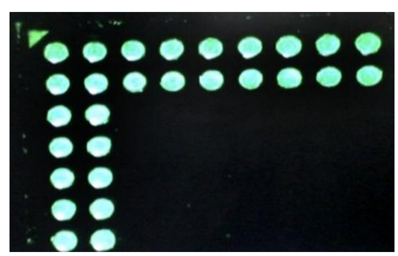


Figure 18. Inspecting Paste on Spheres in Vision

The top package can be dipped in either solder paste or tacky flux as the top package will have a lower CTE mismatch and therefore less warpage. If the top package is dipped in solder paste, it can be inspected in vision. If the package is dipped in tacky flux, inspection in vision will not be possible, however, the imprint of the spheres into the flux tray can be inspected to insure that all spheres have flux on them. Some flux manufacturers are now adding a color dye to the tacky flux so inspection is possible.

The top package is aligned with the top side of the bottom package and placed onto the bottom package. Force control (1-2 newtons) is recommended. Both packages are then reflowed together. Some BGA rework manufacturers' recommend reflowing the packages separately. It is not clear why this approach is recommended as it results in an additional complete reflow cycle for the bottom package and the PCB. Nitrogen is recommended for the reflow process to improve wetting and reduce oxidation of the fine grain solder particles.

Through-Mold Via (TMV™) PoP (Figure 19) is Amkor's next generation 3D packaging solution which uses a laser ablation process to create recesses within the dielectric material as opposed to current photolithographic techniques where the signals are formed on the surface of the dielectric [5]. TMV PoP utilizes a balanced, fully-molded structure which improves warpage control and allows bottom package thickness reductions. Thermal Shadow Moiré testing demonstrates that the TMV PoP exhibited a dramatic improvement in warpage compared to the conventional PoP package as shown in Table 4.

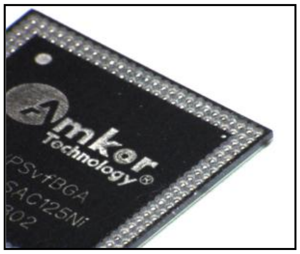


Figure 19. Amkor TMV™ PoP

Table 4: Thermal Shadow Moiré Results [6]

Package/Substrate	TMV/0.30	FC PoP/0.30
Max Warpage (μM)	-51.8	-136.8

TMV™ PoP Solderability Testing

A 77x132mm test board (8 layers, 1.0mm thick) and 14mm TMV™ PoP devices were used for solderability testing (Figure 20). The bottom package is 0.65mm pitch with 620 I/O while the top package is 0.5mm pitch with 200 I/O. A typical preheat/soak/ramp/reflow/cool soldering profile was developed on a BGA Rework System by instrumenting one TMV PoP.

No thermal changes were made to the reflow process throughout the study. The only thing that was changed was the component/site preparation method and material as summarized in Table 5.

Figure 20. TMV PoP Test Vehicle

Table 5: TMV™ PoP Solderability Testing

	Bottom Package				Top Package		
Placement #	**Paste Dip**	**Flux Dip**	**Flux Site**	**Flux Top of Pkg**	**Paste Dip**	**Flux Dip**	**Nitrogen**
1-2		✓				✓	
3-4			✓	✓			
5-6			✓			✓	
7-8	✓				✓		
9-10	✓		✓	✓	✓		
11-12			✓		✓		
13-14		✓				✓	✓
15-16	✓				✓		✓
17-18			✓	✓			✓

The objective of the study was to see if the component/site preparation method/material had any significant impact on the TMV™ PoP in regard to warpage, HiP or any other defects.

PoP Solderability Testing Results

Cross sectioning results of the PoP solderability testing indicates that the biggest factor influencing good joint formation is the liberal use of flux. Best results for both top and bottom joints resulted when flux was brushed on both the site and top side pad surface on the bottom PoP. Flux dipping produced reasonable results but joints were not always as well formed as when flux was brushed on the pads. Solder paste dipping showed the poorest results with some cases of complete opens and at best only partial wetting of solder sphere to pad. Nitrogen also appears to produce better overall results than air and appears to help in marginal soldering situations.

One thought is that the small size of the PoP solder spheres limits the volume of flux or solder paste that can be applied during the dipping process to levels that are not sufficient to produce good solderability results. However, the sample size in the study was relatively small and should be verified by follow up testing using a larger sample size.

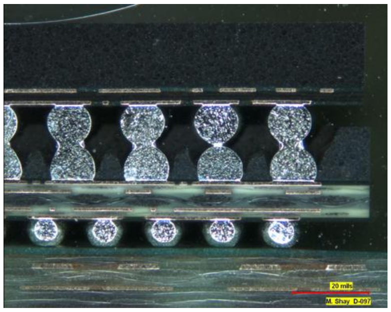


Figure 21. Cross Section showing poor results (solder paste dipping)

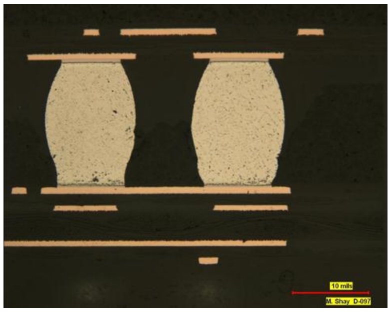


Figure 22. Cross Section showing good results (flux brushed on)

QFN/MLF Rework

Amkor, calls them MLF's (micro lead-frame) while many others refer to these components as QFN's (quad flat, no leads). IPC refers to these devices as BTC's (bottom terminated components).

The MLF is a plastic encapsulated package with a copper lead frame substrate. The package uses perimeter lands on the bottom of the package to provide electrical contact to the PCB. The package also has a large center pad on the bottom of the package to provide an efficient heat path to the PCB.

The two major issues associated with QFN technology are excessive voiding in the thermal pad area and out gassing which may cause solder balling and/or splatter. Both issues are caused by flux entrapment due to the low component standoff height. IPC-A-610 allows voiding levels of up to 25%.

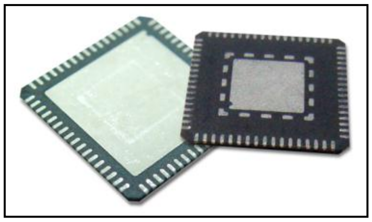


Figure 23. QFN / MLF Devices

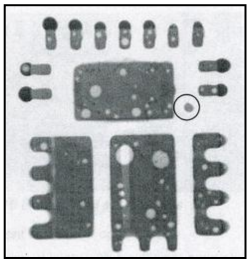


Figure 24. Example of Voiding and Solder Balling under a Power QFN (7)

The QFN/MLF rework process mimics the PoP process explained earlier except that solder paste dipping is not possible with QFN's due to package flatness. Solder paste must be applied to either the component or the pads prior to replacement. There are a number of processes for applying solder paste, including solder preforms, polyimide or metal site stencils, polyimide or metal component stencils, stay-in-place stencils or multi-up stencils.

The stay-in-place stencil approach applies a tacky polyimide stencil to the device (Figure 25a). Paste is applied and the excess removed with a doctor blade. The device is then reflowed and the stencil is removed leaving a bumped device (Figure 25b). A second stencil is applied to the pads and paste or flux is applied (Figure 25c). The bumped QFN/MLF is placed in the site stencil and reflowed. The site stencil remains on the board permanently. Reliability testing by NASA/DOD indicates that the stay-in-place stencil performed at the same level as traditional board paste printing.

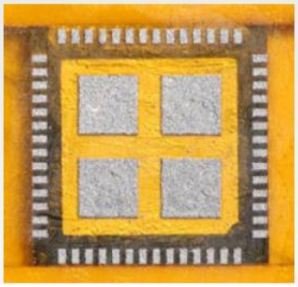


Figure 25a
QFN with Stencil [8]
And Solder Paste

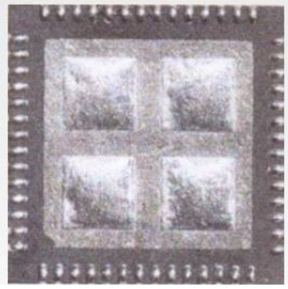


Figure 25b
Bumped QFN after [8]
Reflow and Removal
of Stencil

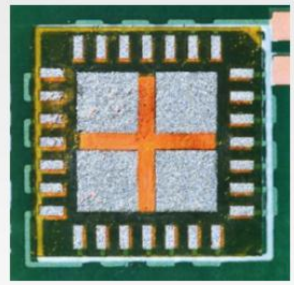


Figure 25c
Stay-in-Place Site [8]
Stencil

Another pasting methodology is the multi-up method where a multi-up stretched metal stencil is used instead of a single component stencil (Figure 26). Typically twenty devices can be stenciled together depending on size. After the components are stenciled, a vacuum table is activated to hold the components in place as the stencil is lifted. The pasted components are reflowed and stored as bumped components for later use. Tacky flux is applied to the site prior to placement.

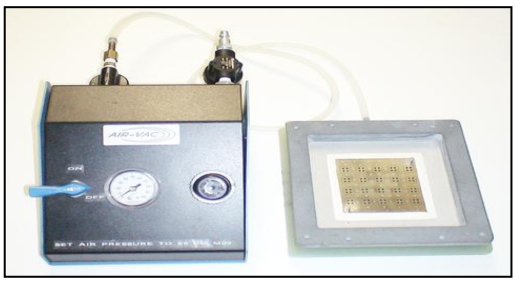


Figure 26. Multi-Up Stenciling System

The design of the solder paste stencil for the thermal pad on a QFN is critical to help minimize voiding and out gassing. Amkor recommends that the stencil have multiple smaller openings instead of one large opening which will typically result in 50% to 80% solder paste coverage. Amkor also recommends a stencil thickness of 0.125mm for 0.4 and 0.5mm pitch parts and 0.15-0.2mm thickness for coarser pitch parts [9] .

QFN Solderability Testing

A 203x140mm (8"x5.5") 0.062" thick test vehicle along with 10mm square MLF components with 0.5mm pitch were used for solderability testing. Two (2) thermal profiles were developed; a short soak/ramp profile and a longer soak/ramp profile.

Three paste-on-device metal stencils with different center pad designs and paste coverage were used as shown below. All stencils used were 0.125mm (.005") thick.

Center Pad #1

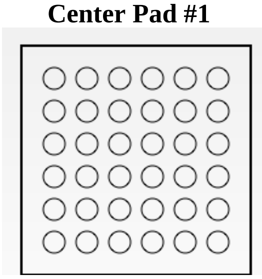


Figure 27a
1.00mm diameter [10]
circles @ 1.2mm pitch
– 50% coverage

Center Pad #2

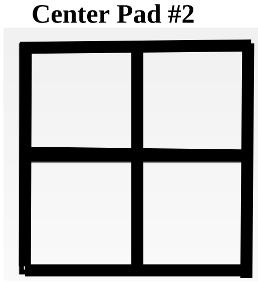


Figure 27b
3.10mm squares [10]
@ 3.95mm pitch
– 60% coverage

Center Pad #3

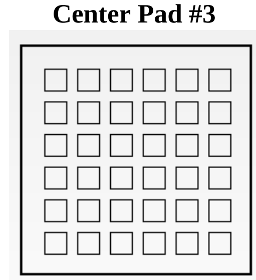


Figure 27c
1.35x1.35mm squares [10]
@ 1.5mm pitch
– 81% coverage

In addition, the Multi-Up Stencil System described previously was used in conjunction with the center pad #2 stencil design. The MLF devices were stenciled and reflowed prior to use in the test. Indium 9.0A No Clean solder paste (type 4 mesh) was used with all stencils.

The objective of the solderability study was to determine which stencil design provided the best results in regard to minimum voiding in the center pad region. A second objective was to determine if the pre-reflowed QFN's from the multi-up stencil performed as well as pasted QFN's with the same stencil design.

Results

Voiding results using the various center pad stencil designs and short versus long soak/ramp cycles are summarized in Table 6.

Table 6 : QFN Voiding Summary

				Voiding (Center Pad)	
Stencil	**Location**	**% Coverage**	**Profile**	**Total**	**Largest Single**
36 Circles	1A	50%	Short Soak/Ramp	11.5%	1.6%
36 Circles	1B	50%	Long Soak/Ramp	6.5%	0.6%
4 Windows	2A	60%	Short Soak/Ramp	17.9%	2.0%
4 Windows	2B	60%	Long Soak/Ramp	10.5%	1.9%
25 Windows	3A	81%	Short Soak/Ramp	13.4%	2.4%
25 Windows	3B	81%	Long Soak/Ramp	11.3%	1.8%
4 Windows (1)	4A	60%	Short Soak/Ramp	13.3%	1.1%
4 Windows (1)	4B	60%	Long Soak/Ramp	16.1%	1.4%

(1) Using Multi-Up Stencil, Reflowed in Advance of Use, Flux Added to Site

All of the QFN's had total percent voiding well below the IPC specification of 25%. The long soak stage provided less voiding than the short soak stage with all three stencils. The long soak stage with the lowest percent solder paste coverage yielded the best results. Finally, the QFN's that were pre-bumped with the multi-up stencil approach yielded similar results to the standard pasting approach.

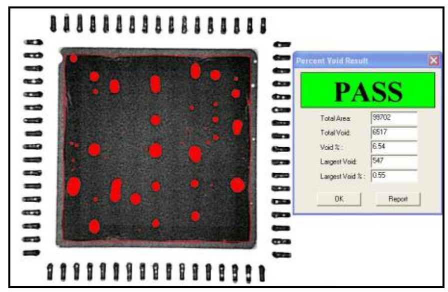


Figure 28. QFN with acceptable void levels

Conclusion

Large area arrays (>50mm) on large, high thermal mass PCB's will create rework challenges including meeting the current strict reflow and package temperature guidelines that were established for much smaller BGA's. Other challenges that will need to be addressed include alignment capability, warpage, large board handling and safe/fast/effective site cleaning. Access to next generation large area arrays on high thermal mass PCB's is required to develop effective rework solutions.

Methodologies must be developed for reworking microscopic 01005's in a cost effective, practical manner than provides high throughput capability. The ability to clean the site and prepare it with solder paste will be important as 50% of 01005 defects are created by the paste printing process. A "Man"/Machine Rework Interface (MMRI) approach was proposed as a possible solution.

Packaging innovations such as Amkor's TMV™ PoP will help resolve Head in Pillow (HiP) issues that are found frequently with current PoP packaging technology. A PoP solderability study showed that manual flux application and the use of nitrogen yielded the best results.

Several MLF/QFN solder paste application methods were discussed including two new and innovative methods: stay-in-place stencils and the multi-up stencil. The design of the stencil for the center pad area is critical to minimize voiding caused by flux entrapment from the low package stand-off height.

Modifications to existing BGA rework equipment as well as new equipment and processes will be required to meet the challenges associated with reworking next generation SMT applications.

Acknowledgements
I would like to acknowledge Mario Scalzo, Senior Technical Support Engineer at Indium Corporation for providing all of the solder pastes and fluxes used in this paper as well as for the insight he provided regarding the various solder alloys and flux chemistries.

I would also like to thank Chuck Richardson at iNEMI for allowing me to utilize information from the Rework and Repair section of the 2013 iNEMI Technology Roadmap in this presentation.

Thanks also to Don Morgenstern and Ron Wachter at Air-Vac for performing all of the PoP and MLF solderability testing. I would also like to thank all of my other co-workers at Air-Vac who helped on this project.

Finally, I wish to thank MPI for providing X-Ray analysis for the PoP and QFN/MLF solderability tests as well as 01005 production assemblies and Endicott Interconnect Technologies for performing the PoP cross sectioning.

References

1. "iNEMI 2013 Technology Roadmap; Rework and Repair Section"
2. "iNEMI 2013 Technology Roadmap; Rework and Repair Section"
3. Combet and Chang, Vi Technology "01005 Assembly, The AOI Route to Optimizing Yield", Page 4
4. Scalzo, Indium "Addressing the Challenge of Head-in-Pillow Defects in Electronic Assemblies, Page 1
5. Zwenger, et al, "Next Generation PoP Platform with TMV™ Interconnection Technology", Page 7
6. Zwenger, et al, "Next Generation PoP Platform with TMV™ Interconnection Technology", Page 7
7. International Rectifier. Application Note AN-1137. June, 2008, Page 7
8. BEST, Inc Web Site "Stencil Mate™ Leadless Device Rework Stencils"
9. Amkor Technology®. Application Notes for Surface Mount Assembly of Amkor's MLF® Packages". September, 2008, Rev G, Page 11
10. Amkor Technology®. Application Notes for Surface Mount Assembly of Amkor's MLF® Packages". September, 2008, Rev G, Page 11

Contacts

Brian.czaplicki@air-vac-eng.com

- Based on only a few test samples subjected to thermal cycling and visual inspection, it appears that re-columned CGA60 package with no interposer is a viable rework solution from an assembly perspective only. Further work is required to substantiate these test results for an active-die version.
- All CGA1144 assemblies with re-columned packages passed 200 severe thermal cycles (–120°/85°C) with no apparent visual damage or daisy chain failures.
- Based on limited thermal cycle test results and visual inspection during thermal cycling, it appears that re-columning of CGA1144 is a viable option from a solder attachment perspective only.

Acknowledgments

The research described in this publication is being conducted at the Jet Propulsion Laboratory, California Institute of Technology, under a contract with the National Aeronautics and Space Administration. Copyright 2012, California Institute of Technology. Government sponsorship acknowledged.

The author would like to acknowledge A. Mehta, N. Neverida, R. Ruiz at JPL for their support in test vehicle assembly, thermal cycling, failure analysis. Thanks also to column attachment manufacture's personnel for providing service and support. The author extends his appreciation to program managers of the National Aeronautics and Space Administration Electronics Parts and Packaging (NEPP) Program, including Michael Sampson, Ken LaBel, and Dr. Charles Barnes and Douglas Sheldon for their continuous support and encouragement.

References

[1] R. Ghaffarian, "Thermal Cycle Reliability and Failure Mechanisms of CCGA and PBGA Assemblies With and Without Corner Staking," IEEE Transactions on Components and Packaging Technologies, Vol. 31, Issue 2, June 2008, Pages 285–296.

[2] R. Ghaffarian, "CCGA Packages for Space Applications," Microelectronics and Reliability, Volume 46, Issue 12, December 2006, Pages 2006–2004.

[3] R. Ghaffarian, "Thermal Cycle and Vibration/Drop Reliability of Area Array Package Assemblies," Chapter 22 in *Structural Dynamic and Photonic of Electronic Systems*, ed. E. Suhir, T.X. Yu, D.S. Steinberg (Wiley, 2011).

[4] White Paper, "Solder Column Qualification for Ceramic Column Grid Array (CCGA)," Aeroflex Colorado Springs, July 2008: http://ams.aeroflex.com/ProductFiles/AppNotes/AeroflexCCGAQualWhitePaper.pdf

[5] R. Kuang, L. Zhao,, "Thermal Cycling Test Report for Ceramic Column Grid Array Packages (CCGA)," Actel Corp.: http://www.actel.com/documents/CCGA_board_level_testing_report.pdf

[6] R. Kuang, R., L. Zhao, "Thermal Cycling Test Report for Ceramic Column Grid Array Packages (CCGA)," Actel Corp.: http://www.actel.com/documents/CCGA_board_level_testing_report.pdf

[7] Column Grid Array and Rework, IBM User's Guideline (July 22, 2002).

[8] R.N. Master, R.N. and Ong, O.T., "Ceramic Grid Array Technologies for ACPI Applications" *The Proceedings of Surface Mount International*, Chicago, IL (Sept. 25–28, 2000).

[9] IPC-9701A, "Performance Test Methods and Qualification Requirements for Surface Mount Solder Attachments," IPC, Association Connecting Electronics Industries.

Advanced Rework Technology & Processes

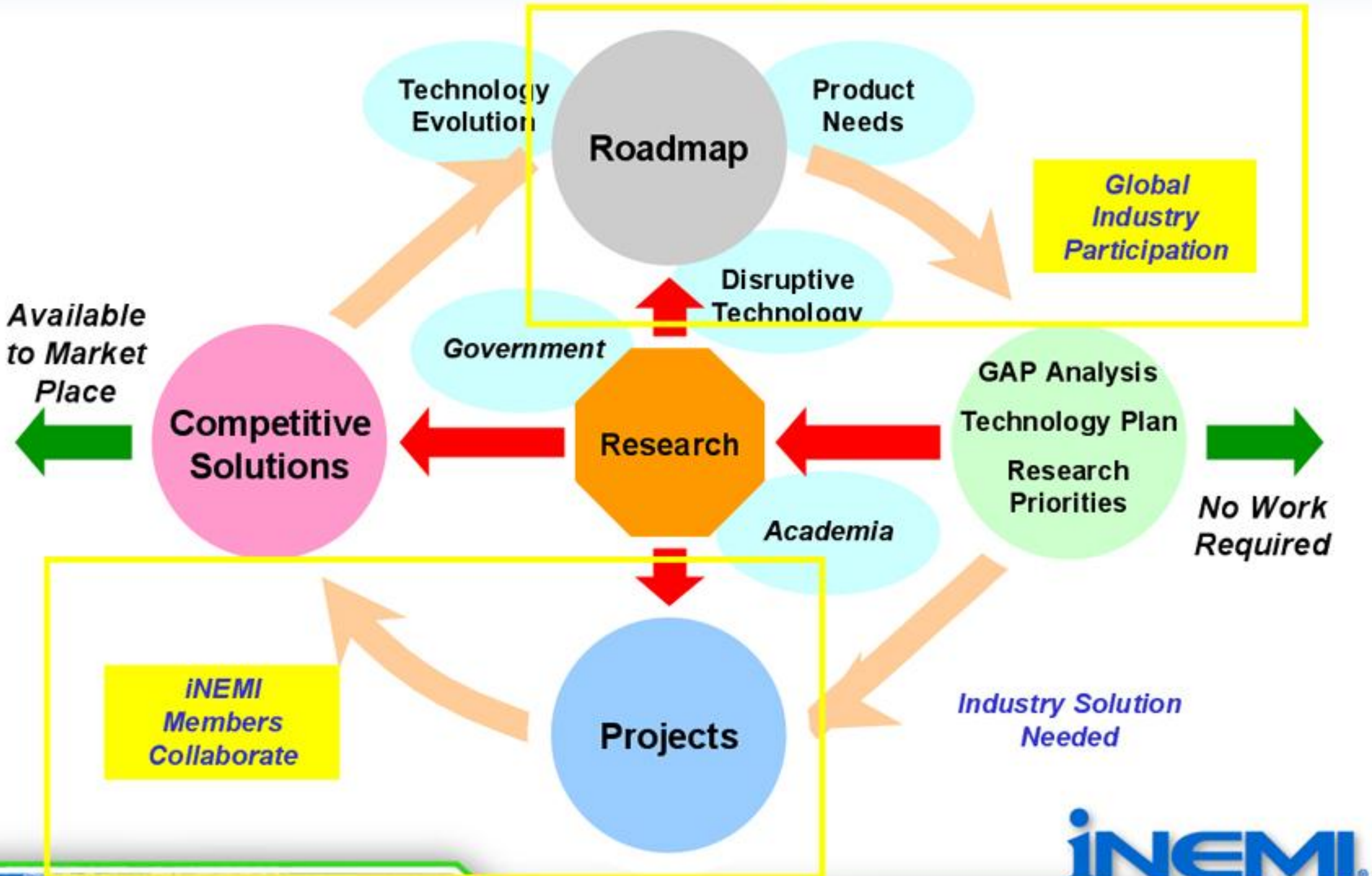
For Next Generation SMT Applications

IPC APEX Expo Technical Conference

February 19-21, 2013

Brian P. Czaplicki
Air-Vac Engineering Company, Inc.
Brian.Czaplicki@Air-Vac-Eng.com

Methodology



2013 iNEMI Roadmap

Rework & Repair Section: Challenges & Gaps [1]

Establishing a process for reworking large (>50mm) area arrays on thermally challenging PCB's while conforming to the current J-STD-020 spec for maximum package body temperature of 260˚C (BGA connector bodies often blister, melt or discolor at temperature required to reflow the joints).

Process for hand soldering 01005 [0402 metric] components.

- Development of fluxes for hand soldering which do not cause a reliability concern if not properly heat activated (dendritic growth > electronic failure).
- Guidelines for external benchtop preheaters to aid in hand soldering of Lead-Free assemblies.

2013 iNEMI Roadmap

Rework & Repair Section: Challenges & Gaps [(1)]

Industry Standardized Rework Processes for QFN/MLF and PoP Components

- Site redressing processes that prevent lifted pads, solder mask damage and copper pad dissolution.

- Development needed to understand the advantages/disadvantages of:
    - *Paste printing on board*
    - *Paste printing on component*
    - *Paste dispensing on board*
    - *Flux application techniques*

2013 iNEMI Roadmap

Rework & Repair Section: Challenges & Gaps (1)

Establishing a process for reworking large (>50mm) area arrays on thermally challenging PCB's while conforming to the current J-STD-020 spec for maximum package body temperature of 260˚C (BGA connector bodies often blister, melt or discolor at temperature required to reflow the joints).

Process for hand soldering 01005 [0402 metric] component.

- Development of fluxes for hand soldering which do not cause a reliability concern if not properly heat activated.
- Guidelines for external benchtop preheaters to aid in hand soldering of Lead-Free assemblies.

2013 iNEMI Roadmap

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB's

Table 1 [(2)]

Soldering Process	Parameter	Units	2011	2013	2015	2017	2023
Pb-free	Maximum package size *(1)*	mm	50	50	55	60	75
	Maximum package temperature (dependent on component body size)	°C	245-260	245-260	245-260	245-260	245-260
	Target solder joints temperature	°C	235	235	235	235	235
	Target delta T across solder joints	°C	<10	<10	<10	<10	<10
	Typical rework profile length (time)	min	8	8	8	8	8
	Time Above Liquidus (TAL)	sec	60-90	60 - 90	60 - 90	60 - 90	60 - 90

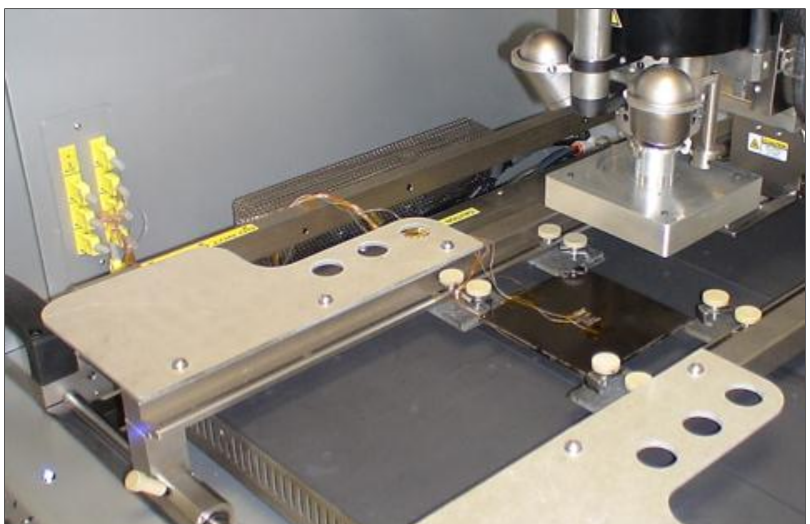
(1) Cited as a significant issue

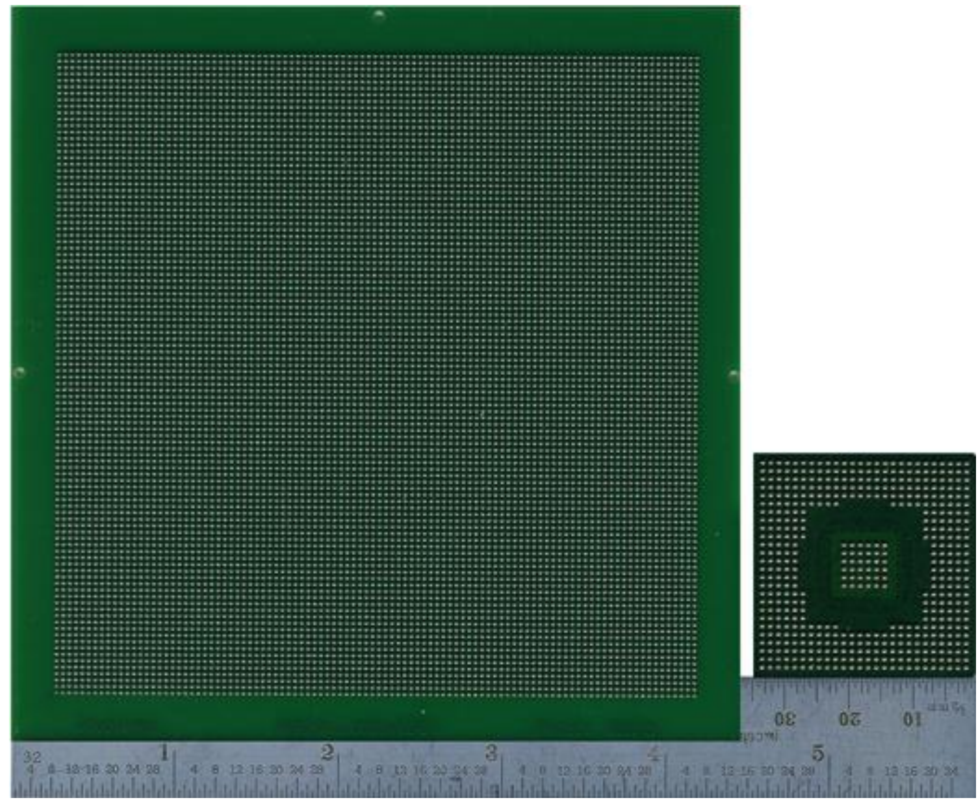
- *Maximum component size will increase by 50-100% and will be used on large, high thermal mass PCB's, however the rework guidelines are currently unchanged.*
- *Developments needed by component manufacturers to increase maximum temperature specs on package body (connector bodies rated to 240˚C).*
- *Rework process solutions being developed/need to be developed.*
- *May be equipment developments such as vapor phase rework.*

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB’s

Large Component Challenges

- Reflow within iNEMI specifications
    - *<10˚ Delta T*
    - *Max package temperature: 260˚C*
    - *Required joint temperature: 235˚C*
    - *60-90 seconds TAL*





114mm x 114mm (4.5 x 4.5) BGA (10,000 I/O)
(35mm BGA as size reference)

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB's

Table 2: 114mm BGA (PH 150°C, 495C @ 2.5 scfm. 150 sec)

		Max Temp (°)		Delta T (°)		TAL (sec)	
TC #	Location	Actual	Spec	Actual	Spec	Actual	Spec
1	Top of Package	260	260		-	-	-
2	Joint (corner)	235	235	6	< 10	52	60-90
3	Joint (corner)	235	↓	↓	↓	55	↓
4	Joint (corner)	239				72	
5	Joint (corner)	238				68	
6	Joint (corner)	237				69	
7	Joint (corner)	241				75	

- Development effort for nozzle design
- Significant thermal energy required to reflow the 114mm BGA
- Board cooling system to control TAL
- **Max package temperature reached the J-STD-020 spec**

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB’s

Table 3: 114mm BGA (PH 150°C, 350C @ 3.0 scfm. 167 sec)

		Max Temp (°)		Delta T (°)		TAL (sec)	
TC #	Location	Actual	Spec	Actual	Spec	Actual	Spec
1	Top of Package	250	260		-	-	-
2	Joint (corner)	240	235	6	< 10	75	60-90
3	Joint (corner)	240				78	
4	Joint (corner)	241				92	
5	Joint (corner)	241				88	
6	Joint (corner)	235				76	
7	Joint (corner)	239				86	

- Machine modification (max flow increase from 2.5 to 4.0 scfm)
- Increased process flow from 2.5 to 3.0 scfm (25% increase)
- Reduced temperature from 495˚ to 350˚C (30% reduction)
- **High flow/low temperature approach:10˚C reduction in package temperature**

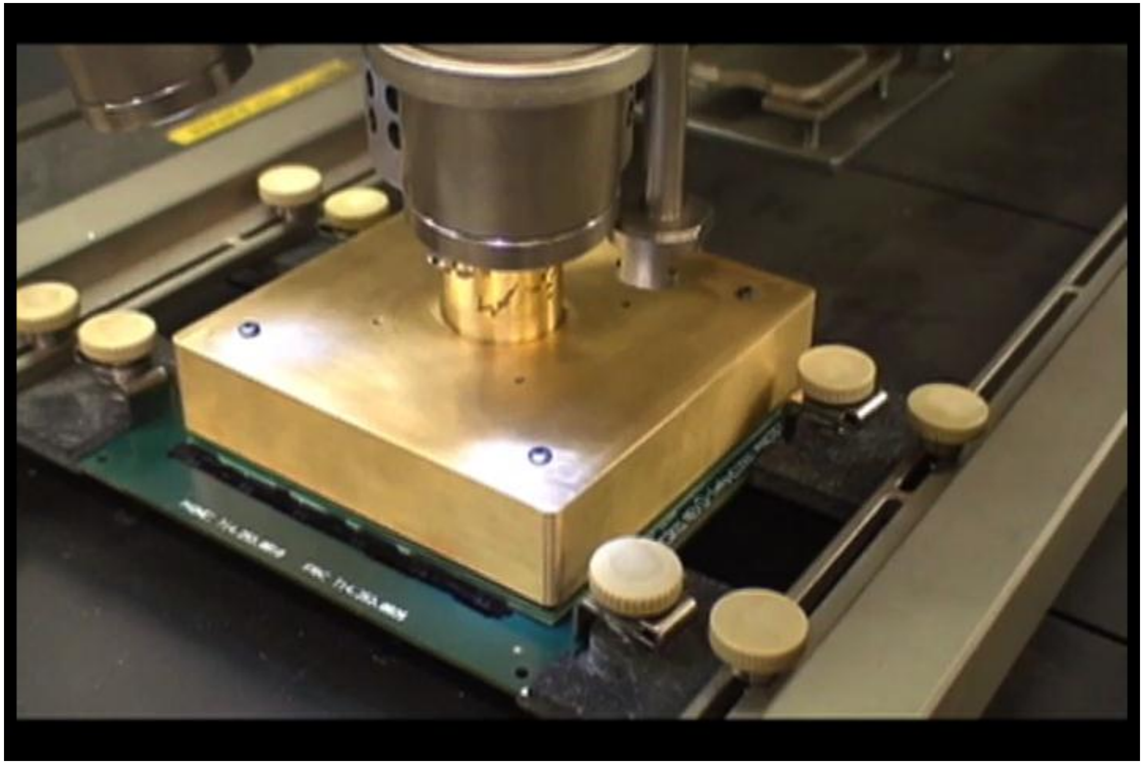
- **IMPORTANT! Rework machines must have significant additional thermal capacity beyond parameters used for large area array test components (actual high thermal mass assemblies)**

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB's

Other Rework Requirements

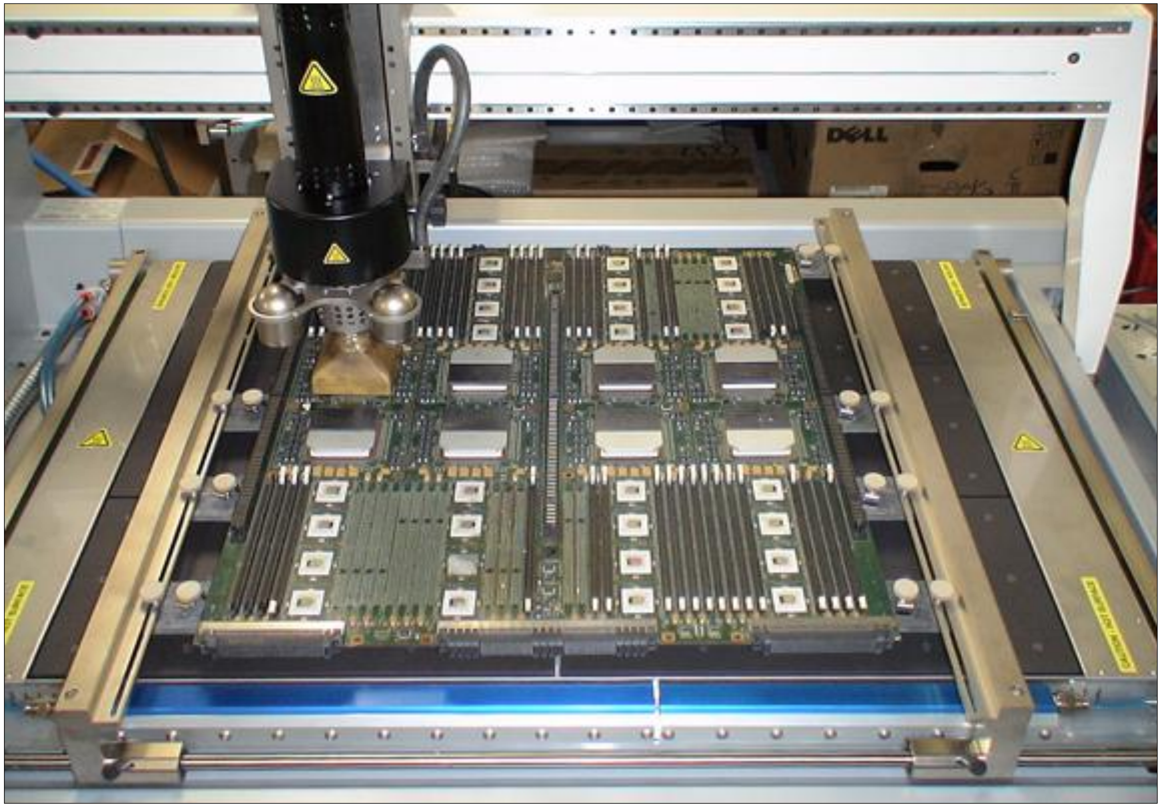
- Large board holding capability
- Large/powerful/uniform board heating capability
- Vision system that can align these very large parts
- Methodology to address coplanarity/warpage
- Capability to remove residual solder from an XXL site safely, quickly and efficiently

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB's



Video Clip #1: 114mm Alignment

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB's



22" x 24" High Thermal Mass PCB on large board rework system
(board courtesy of IBM)

Reworking Large (>50mm) Area Arrays on Large Thermally Challenging PCB's

SUMMARY

	114mm BGA	Other Devices
• Reflow large area arrays within iNEMI specs		
• *10° Delta T*	✓	?
• *Max package temp: 260°C*	✓	?
• *Required joint temp: 235°C*	✓	?
• *TAL: 60-90 seconds*	✓	?
• Alignment	✓	✓
• Coplanarity/warpage	?	?
• Nozzle vacuum capability	?	?
• Safe/fast/effective site solder removal	?	?
• XXL board holding/heating/cooling	✓	✓

Need access to large components/assemblies for process development/testing

2013 iNEMI Roadmap

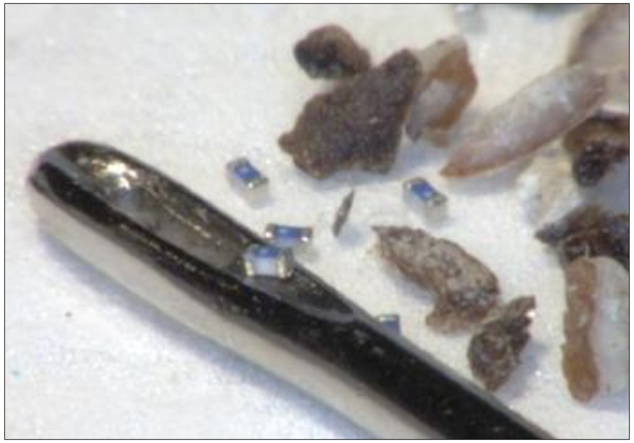
01005 Rework

Establishing a process for reworking large (>50mm) area arrays on thermally challenging PCB's while conforming to the current J-STD-020 spec for maximum package body temperature of 260˚C (BGA connector bodies often blister, melt or discolor at temperature required to reflow the joints).

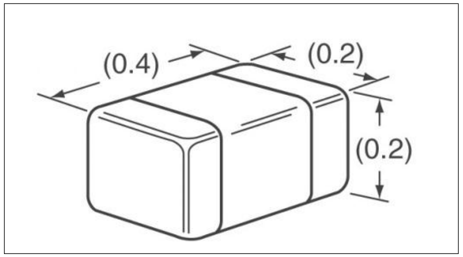
Process for hand soldering 01005 [0402 metric] component.

- Development of fluxes for hand soldering which do not cause a reliability concern if not properly heat activated (dendritic growth > electronic failure).
- Guidelines for external bench top preheaters to aid in hand soldering of Lead-Free assemblies.

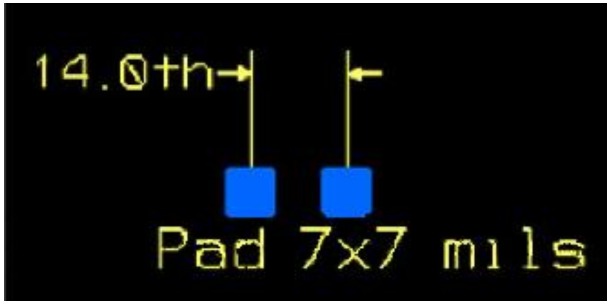
01005 Component/Pad Size



Microscopic size
(01005 in eye of needle and
next to pepper flakes)



Component dimensions (mm)



Pad dimensions (typical)

Microscopic size makes 01005 rework extremely difficult

- Viewing/alignment
- Heating
- Handling

2013 iNEMI Roadmap

01005 Rework

Table 3 ***(2)***

Soldering Process	Parameter	Units	2011	2013	2015	2017	2023
Pb-free	Soldering iron peak temperature used	°C	375	375	375	375	375
	Total contact time	sec	6	6	6	6	6
	Smallest type of discretes being reworked	-	0201	0201	01005	01005	1005
	Type of wire alloy used	-	SAC305/ SnCuNi (low tip dissolution alloys)	SAC305/ SnCuNi (low tip dissolution alloys)	SAC305/ SnCuNi (low tip dissolution alloys)	SAC305/ SnCuNi (low tip dissolution alloys)	SAC305/ SnCuNi (low tip dissolution alloys)

- *Hand solder rework of 01005's is difficult, but possible by a skilled operator*
- *Rapid, conductive heating of ceramic capacitors not viewed as major concern*

01005 Rework: Industry Views

“Post-reflow rework is quite impossible and very expensive to achieve. The soldering tips are too big compared to the size of the package and it is difficult to get a safe mechanical contact to transfer the heat. This manipulation is necessary done under a microscope and becomes risky for the other components and/or the pad.” (3)

“Rework is not recommended when dealing with 01005 components. This is due to the following facts:

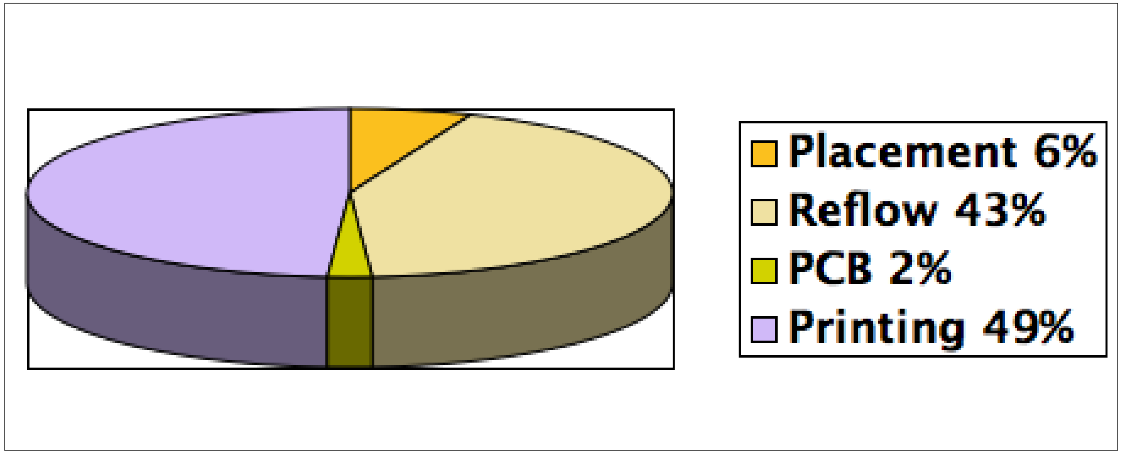
- *Even very fine soldering tips are too big for 01005 components.*
- *A safe mechanical contact to transfer the heat is virtually impossible.*
- *The danger of causing mechanical damage to the component or pad is quite high.*
- *The very small structures and components can not be seen with the naked eye.*
- *Occasionally, one can be done by a highly skilled worker equipped with a microscope, laminar hot air pencil, micro-tweezers and flux.”* (4)

01005 Rework Industry Views

“During the development of the 01005 rework process it was found that heating the circuit board after alignment of the 01005 to the site pads can cause the alignment to change. This was attributed to thermal expansion of the circuit board that caused the position of site pads to shift after the alignment of 01005 to the site. This is the reason no bottom heating is used for rework of these components. 01005 rework is limited to using the residual solder for reattaching replacement 01005’s.” (5)

- *Few (if any) positive comments regarding 01005 rework.*

01005 Rework - Root Cause of Defects (6)



- Printing (insufficients, bridging, excess solder) (49%)
  - *Stencil thickness*
  - *Solder paste*
  - *Printing parameters*

[Existing solder on pads can not be used in half of the cases]

- Reflow (tombstone, floating) (43%)
  - *Heating profile*

- Placement (missing part, face down, bill-boarding) (6%)
  - *Accuracy and repeatability*
  - *Package and tape quality*

01005 Rework Methods

- Hand Soldering
  - *Several hand soldering systems promote 01005 rework capability*
  - *No hand soldering videos demonstrating 01005 rework found*
- BGA Rework Systems
  - *Several BGA rework systems promote 01005 rework capability*
  - *Some 01005 rework videos found*
  - *High equipment cost to rework very low cost components*
  - *Throughput issues (beamsplitter alignment for removal and replacement, fixed process routines)*
  - *Some systems can rework 01005's using residual site solder only (50% of defects)*
  - *Some systems have micro-dispensing capability (repeatability, needle clogging, hot rework environment)*

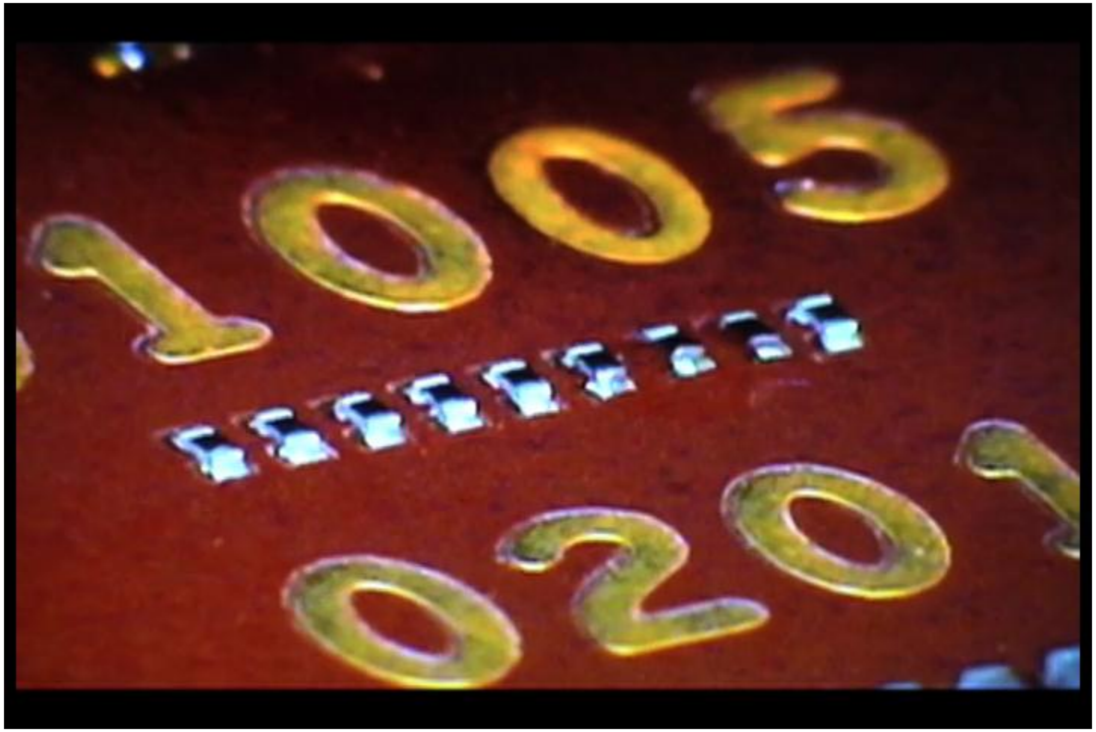
01005 Rework Methods
"Man"/Machine Rework Interface (MMRI)

- Operator has manual control of the rework process but utilizes critical machine features for improved process control
  - *Best of both worlds*
- Advantages vs hand tools
  - *Eliminates the need for manual dexterity*
  - *Improved process control*
  - *Thermal ramping capability (ceramic capacitors)*
  - *High magnification optics*
  - *Methodology for pasting pads*
  - *Convective reflow (self-centering)*
  - *Full board preheat eliminates activation concerns for excess flux*
- Advantages vs BGA rework systems
  - *Lower cost*
  - *Significantly higher throughput*
  - *Eliminates accuracy concerns*
  - *Methodology for pasting pads*
  - *Convective reflow (self-centering)*
  - *Can remove misplaced component and replace in a single step*

01005 Rework Methods
"Man"/Machine Rework Interface (MMRI)

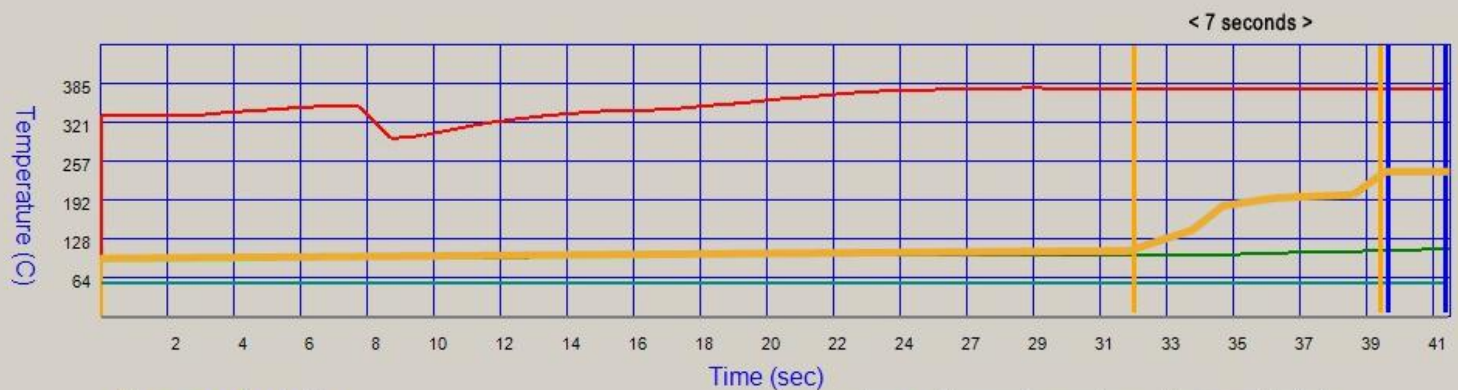
Removal

- Preheat board and nozzle
- Apply halogen free flux with a micro-syringe
- Remove and drop off multiple 01005's (high throughput)



Video Clip #2: Removal of 2 01005's on red board

01005 Removal
"Man"/Machine Rework Interface (MMRI)



Summary Analysis

7 8 1 sec 217

	Minimum (°C)	Maximum (°C)	Slope[+] (°C/sec)	Slope[-] (°C/sec)	Time Above (sec)
Seconds					
Nozzle	297	381	-0.13	0.56	043
Left Preheater	058	058	0.00	0.00	000
Right Preheater	058	058	0.00	0.00	000
Center Preheater	058	058	0.00	0.00	000
TC #1 Board	097	114	0.91	0.56	000
TC #2 01005	098	$\boxed{242}$	$\boxed{12.75}$	0.00	$\boxed{003}$

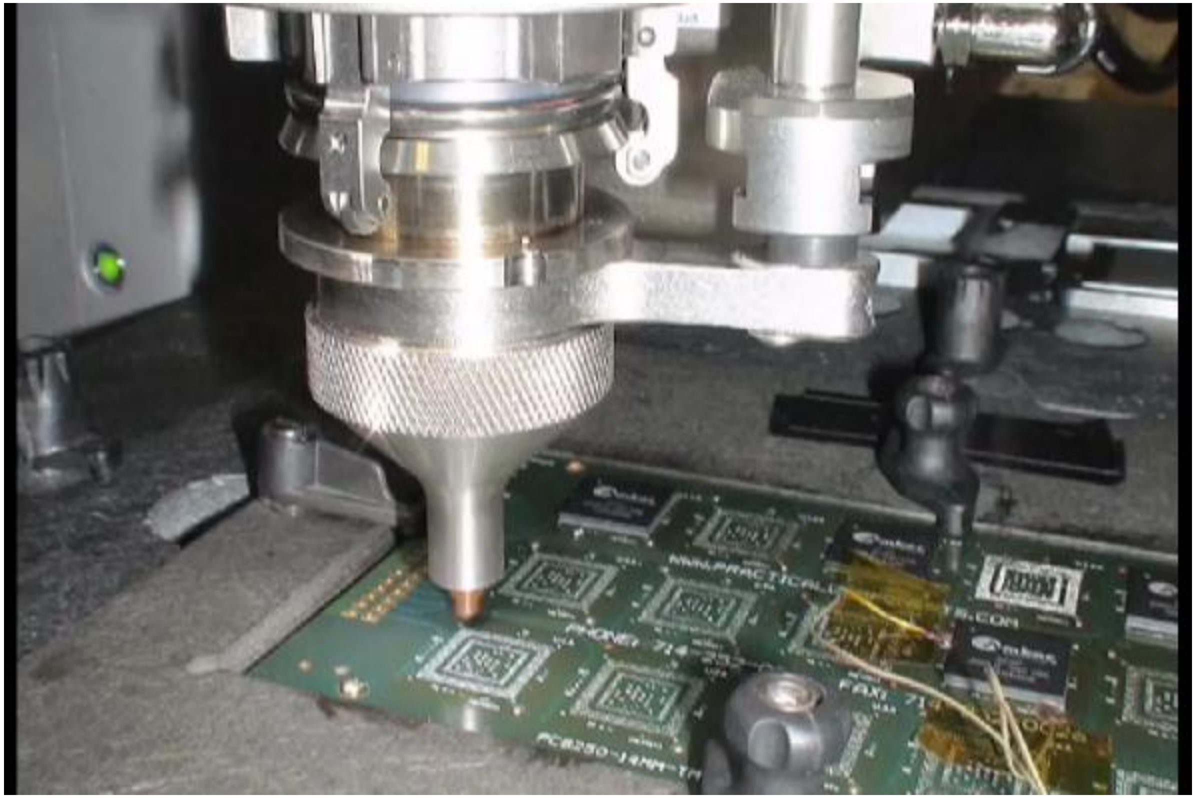
C:\Program Files\Air-Vac-Engineering\DRS25\Export\Run\01005 Removal and SC (SC Tool 380 45 220)-2012-009-028-008-024-020.Csv

01005 Rework Methods
"Man"/Machine Rework Interface (MMRI)

Replace

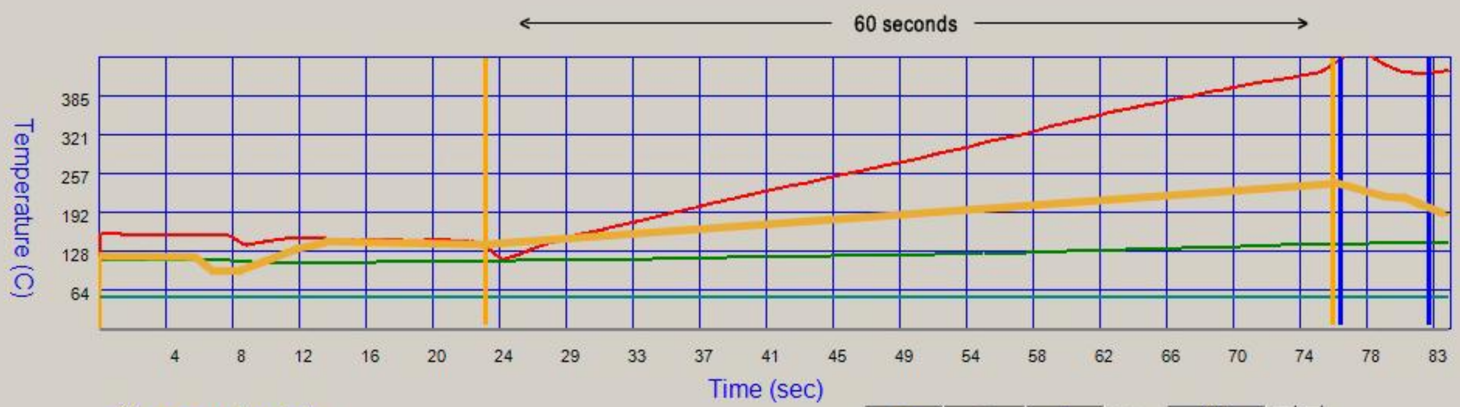
- Apply flux if necessary
- Use micro-site cleaning tool to remove solder from pads
- Use micro-paste transfer tool and precision depth paste application to apply paste to the pads.
- Inspect paste
- Pick, align, place, and reflow
    - *Retract nozzle to allow self-centering*

01005 Rework Methods
"Man"/Machine Rework Interface (MMRI)



Video Clip #3:
• Site clean 2 sites on red board
• Apply paste to 1 site
• Pick/align/reflow

01005 Replacement
"Man"/Machine Rework Interface (MMRI)



Summary Analysis

52 | 57 | 5 sec | 217

	Minimum (°C)	Maximum (°C)	Slope[+] (°C/sec)	Slope[-] (°C/sec)	Time Above (sec)
Seconds					
Nozzle	117	463	5.29	-3.45	045
Left Preheater	054	054	0.00	0.00	000
Right Preheater	054	054	0.00	0.00	000
Center Preheater	054	054	0.00	0.00	000
TC #1 Board	111	146	0.53	0.54	000
TC #2 01005	096	242	1.89	-6.63	017

C:\Program Files\Air-Vac-Engineering\DRS25\Export\Run\01005 Replace (MMRI)-2012-009-028-008-000-015.Csv

01005 Rework - Summary

- Hand soldering rework is difficult but possible for skilled operators

- Alternatives to hand soldering rework

  - *BGA rework systems*
    - Not cost effective
    - Low throughput

  - *MMRI*
    - Manual control with machine-based benefits
    - Higher throughput vs BGA rework systems
    – Convective reflow (controlled ramp for replacing ceramic capacitors)
    - Eliminates manual dexterity required for hand soldering

2013 iNEMI Roadmap

01005 Rework

Establishing a process for reworking large (>50mm) area arrays on thermally challenging PCB's while conforming to the current J-STD-020 spec for maximum package body temperature of 260˚C (BGA connector bodies often blister, melt or discolor at temperature required to reflow the joints).

Process for hand soldering 01005 [0402 metric] component.

- Development of fluxes for hand soldering which do not cause a reliability concern if not properly heat activated (dendritic growth > electronic failure).
- Guidelines for external benchtop preheaters to aid in hand soldering of Lead-Free assemblies.

Hand Soldering Issues

Flux Reliability Issues [7]

- Standard reflow and BGA rework > benign flux residue
- Hand soldering of micro discretes
  - *Uncontrolled flux application > flooding*
  - *Unheated flux*
    - *Residue may create reliability issues (solvent not completely evaporated)*
    - *Lower resistance values*
    - *Dendritic growth > electrical failure*

Solutions

- Use halogen free fluxes
- Incorporate stand-alone preheaters into hand soldering processes

Stand Alone Preheater

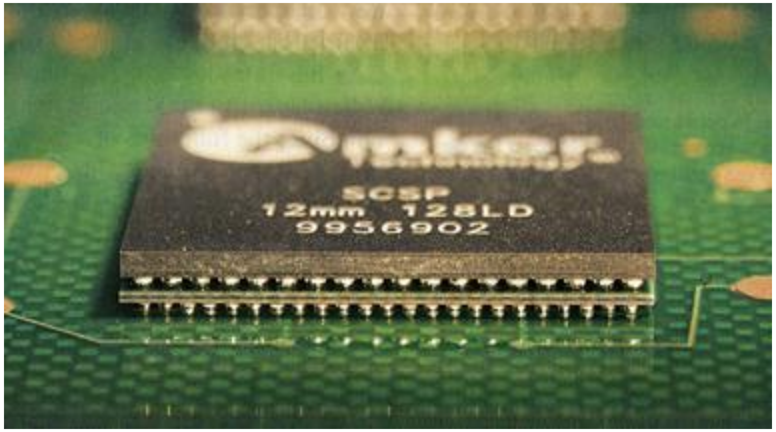
2013 iNEMI Roadmap

Rework & Repair Section: Challenges & Gaps

Industry Standardized Rework Processes for QFN/MLF and PoP Components

- Site redressing processes that prevent lifted pads, solder mask damage and copper pad dissolution.

- Development needed to understand the advantages/disadvantages of:
  - *Paste printing on board*
  - *Paste printing on component*
  - *Component Paste Dipping*
  - *Paste dispensing on board*
  - *Flux application techniques*



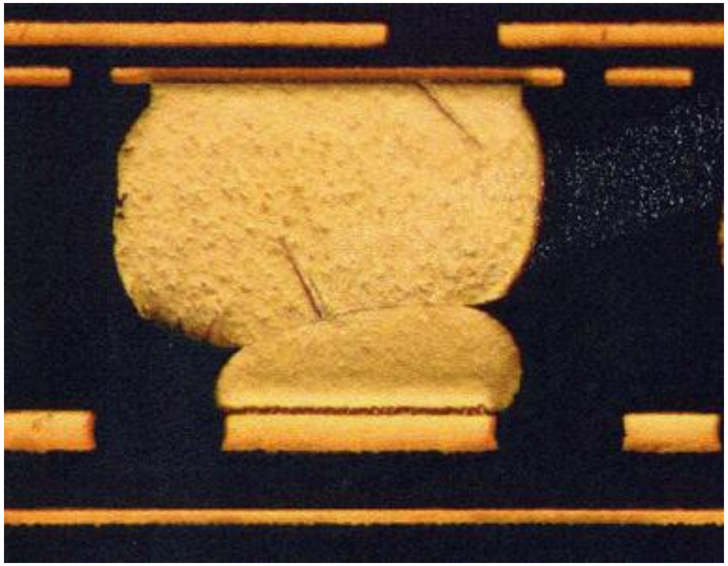
A PoP assembly consists of two fine-pitch BGAs stacked one atop the other. The bottom package is usually a high-density logic device, while the top package is a high-capacity memory device. *Photo courtesy Henkel Corp.*

PoP Rework

Challenges

- Warpage
  - *Thin substrates*
  - *Stacked*
  - *Fine pitch (0.4mm to 0.3mm)*

- Head-in-Pillow: Incomplete coalescence of the solder joint between the PoP sphere and the printed solder paste
  - *Package coplanarity*
  - *Package/PWB warpage (during reflow)*
  - *Poor wetting*
  - *Sphere oxidation*
  - *Silver segregation (cooling)*
  - *Stencil design (insufficient solder)*
  - *Flux exhaustion (extended reflow)*
  - *Solder paste chemistry (higher temperature activation)*

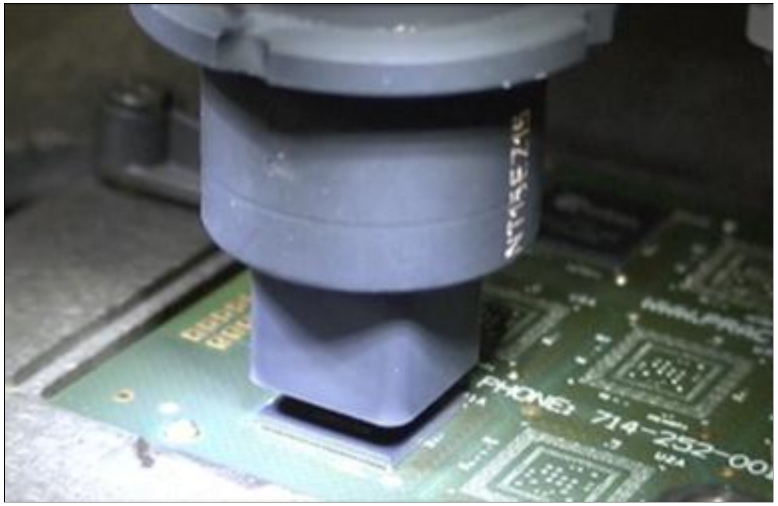


Example of a Head-in-pillow (HIP) defect [(8)].

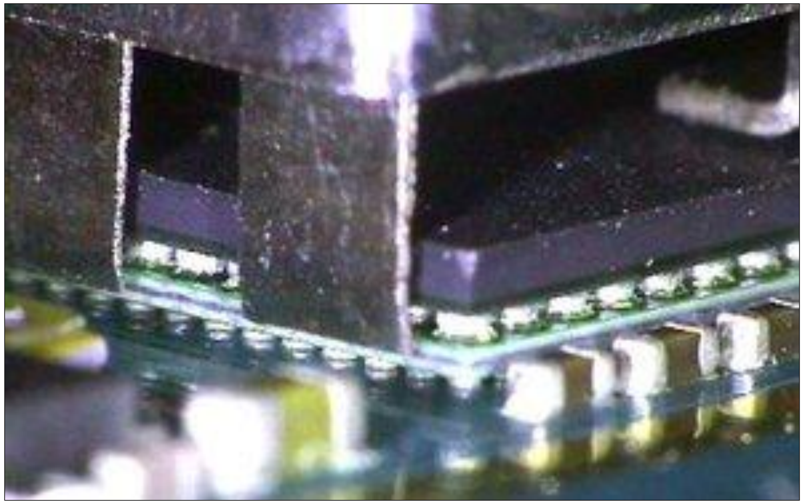
Concave warpage from reflow

PoP Rework

- Remove top and bottom components
  - *Individually (standard nozzle, zero adjacent clearance)*
  - *Together (tweezer nozzle, some clearance required)*
  - *Epoxy corners and remove together (standard nozzle)*

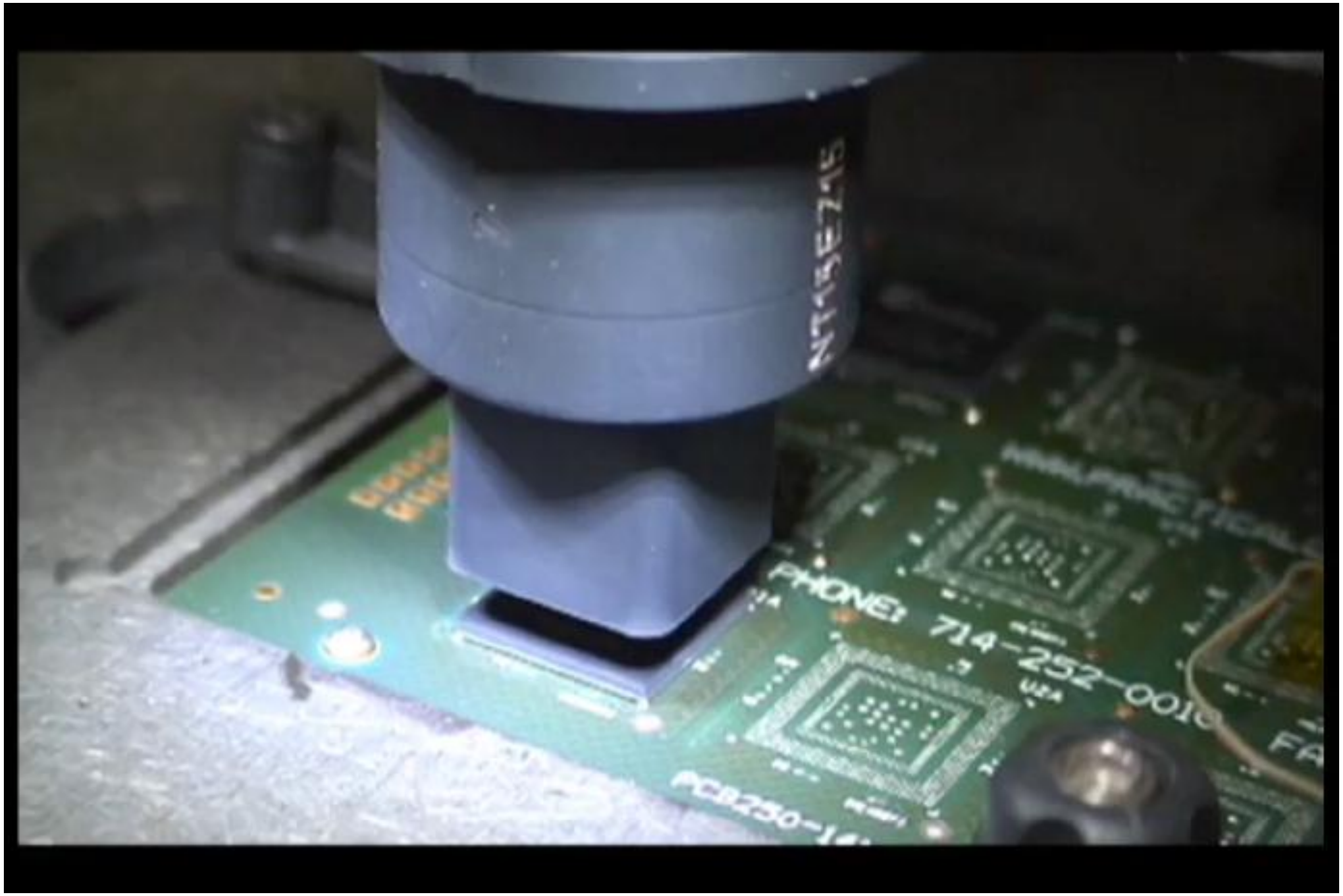


Standard Nozzle



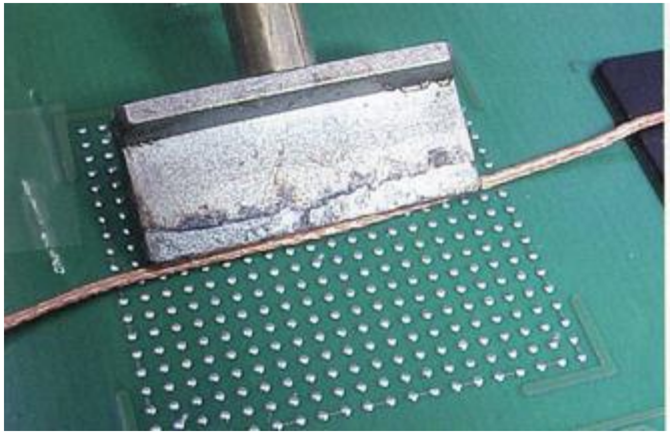
Tweezer Nozzle (Finetech, USA)

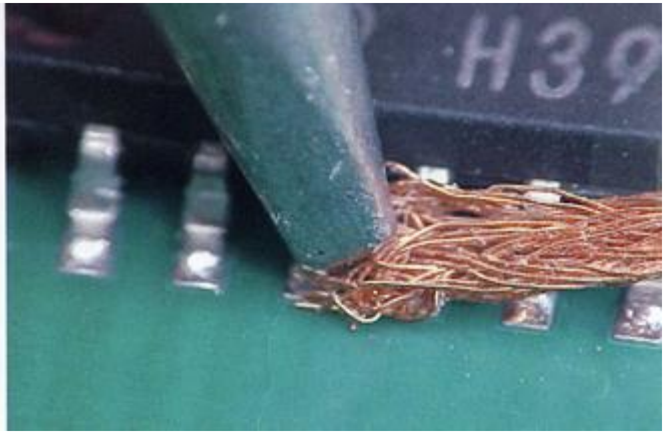
PoP Rework Process - Removal



Video Clip #4: PoP Removal

PoP Rework Process - Site Solder Removal





- Try and avoid having excess braid (left) when desoldering on pads with a wide tip tool. It requires more heat/time. Also avoid braid (right) which has become unwoven.

- By tinning the bit, the solder on the tip will speed up heat transfer to the braid and the whole de-soldering operation. **This is often where damage occurs, when operators do not tin the bit.**

- As the braid and solder are directly in contact with the pad, more care needs to be taken during this operation. It is perfectly feasible to desolder pads in this manner provided care is taken and no pressure is applied to the pads. It is not the heat which lifts or misplaces pads, it is the combination of heat and pressure.

- Manual site dressing is very dependent on operator skill and training.

- Braid still used in 80% of applications! (iNEMI estimate)

PoP Rework Process - Site Cleaning

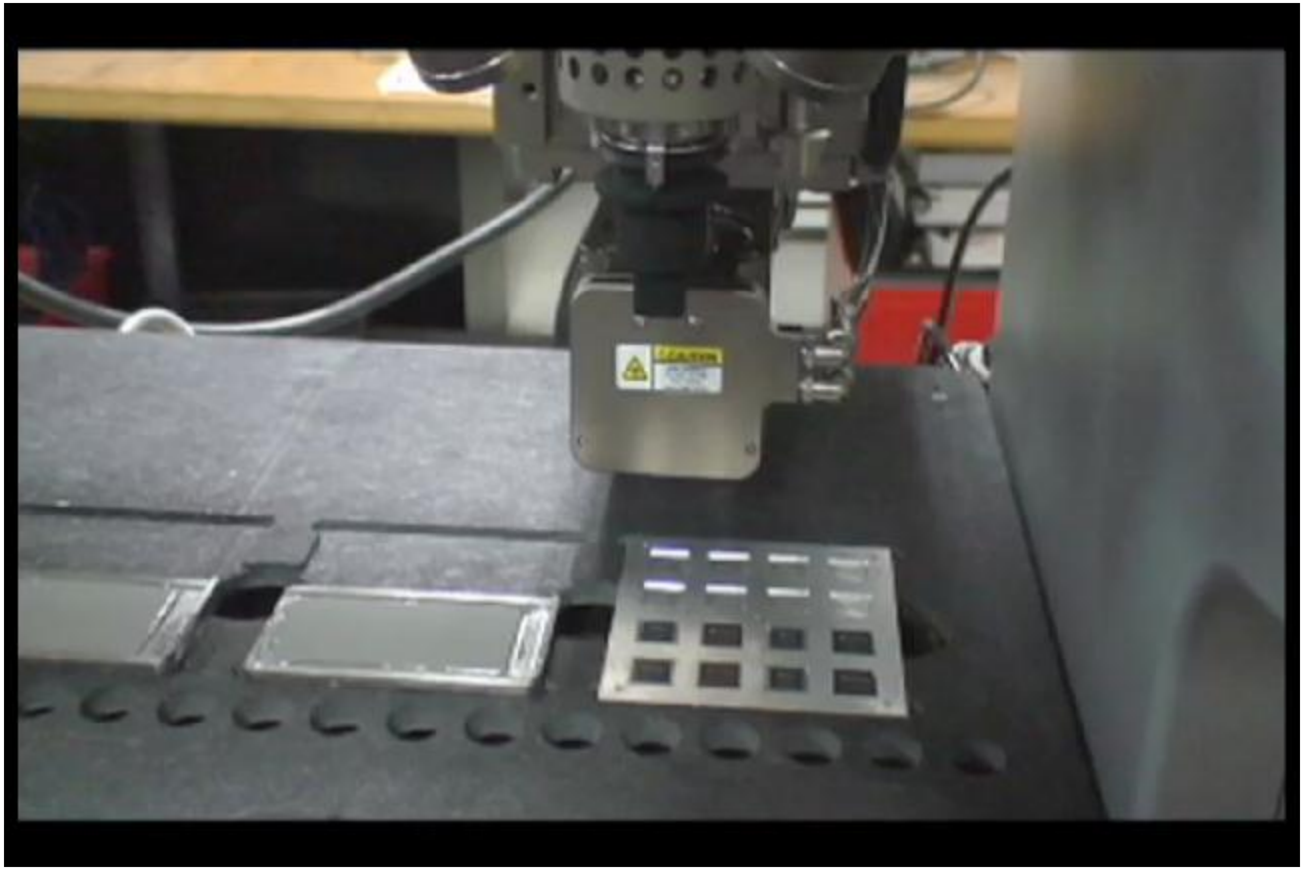
- Non-contact solder removal
  - *Higher lead-free solder reflow temperatures*
  - *Addresses lifted pads and solder mask damage concerns*
  - *Time to redress (versus wick) is an issue*

QuickTime™ and a
H.264 decompressor
are needed to see this picture.

Video Clip #5: Site cleaning

PoP Rework Process - Replace Bottom Package

Pick, Prep, Align, and Place Bottom Package



Video Clip #6: Replace bottom package

PoP Rework Process - Replace Bottom Package

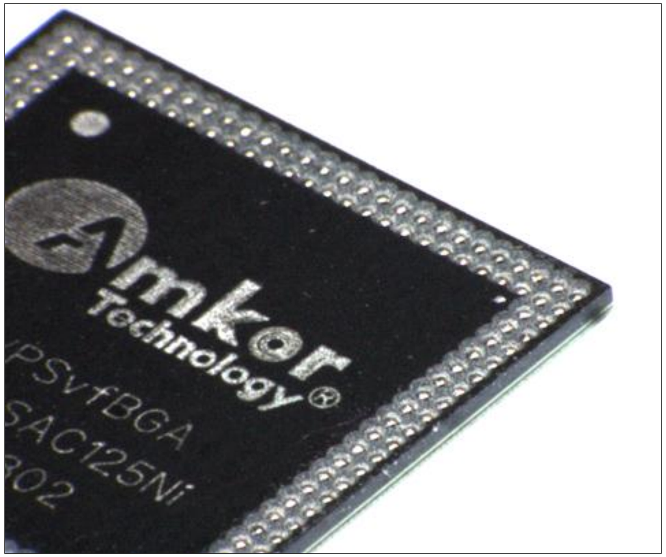
Pick, Prep, Align, and Place Top Package and Reflow both Packages Together

- Some rework manufacturers' recommend separate reflow

Video Clip #7: Replace package and reflow both

PoP Rework

Amkor® Through Molded Via (TMV™) PoP



Thermal Shadow Moire Results [9]

Package/Substrate	TMV/0.30	fcPoP/0.30
Max warpage (um)	-51.8	-136.8

- Laser Ablation Process creates recesses within the dielectric material as opposed to surface formed photolithographic techniques.
- More stable bottom package enables the use of thinner substrates
- Enhanced warpage control

TMV^{TM} PoP Solderability Testing

- **77 x 133mm (3.0 x 5.25”) 1mm thick, 8 layer test board**
- **14mm TMV PoP components**
  - Bottom: 0.65mm (620 I/O)
  - Top: 0.5mm (200 I/O)
- **Fixed: Reflow profile**
- **Variable: Site/component preparation**
- **Dipping paste: Indium 9.88HF - no clean**
- **Flux: Indium PoP flux 8.9HF - LV tacky flux**

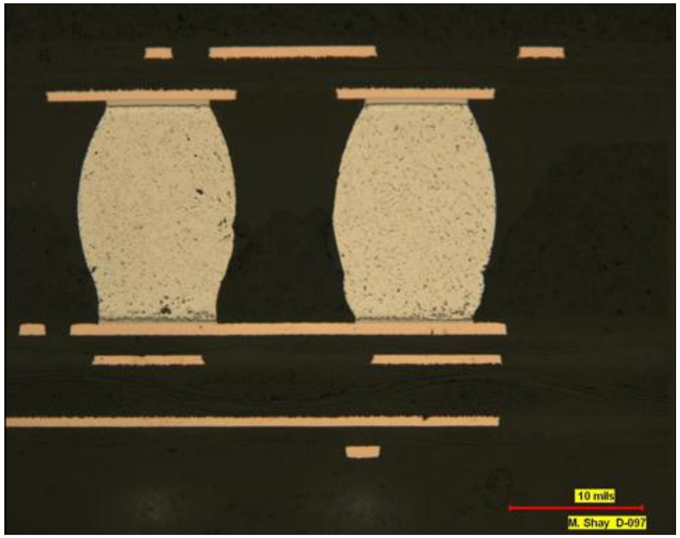
TMV™ PoP Solderability Testing - Component/Site Preparation

	Bottom Package				Top Package		
Placement #	**Flux Site**	**Flux Top**	**Flux Dip**	**Paste Dip**	**Flux Dip**	**Paste Dip**	**Nitrogen**
1-2			✓		✓		
3-4	✓	✓					
5-6	✓				✓		
7-8				✓		✓	
9-10	✓	✓		✓		✓	
11-12	✓			✓		✓	
13-14			✓		✓		✓
15-16				✓		✓	✓
17-18	✓	✓					✓

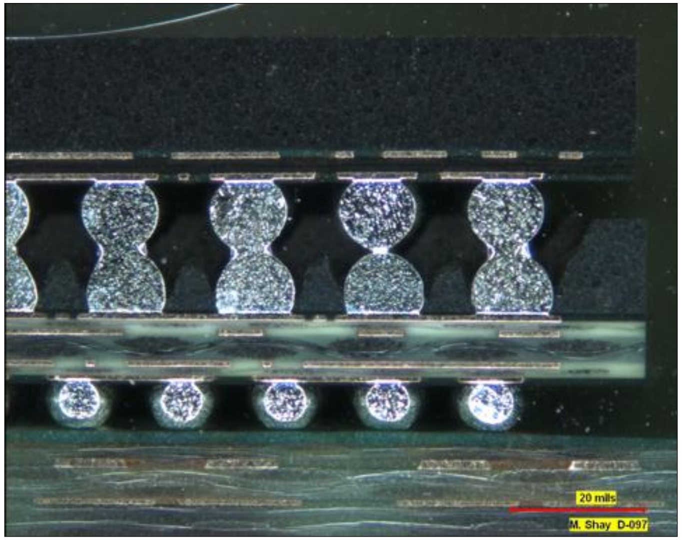
TMV^{TM} PoP Solderability Testing

Cross Sectioning Results

- Manual flux application - was best (higher volume)
- Solder paste dipping only - was worst
- Nitrogen - improved results



Manual flux application



Solder paste dipping

2013 iNEMI Roadmap

Rework & Repair Section: Challenges & Gaps

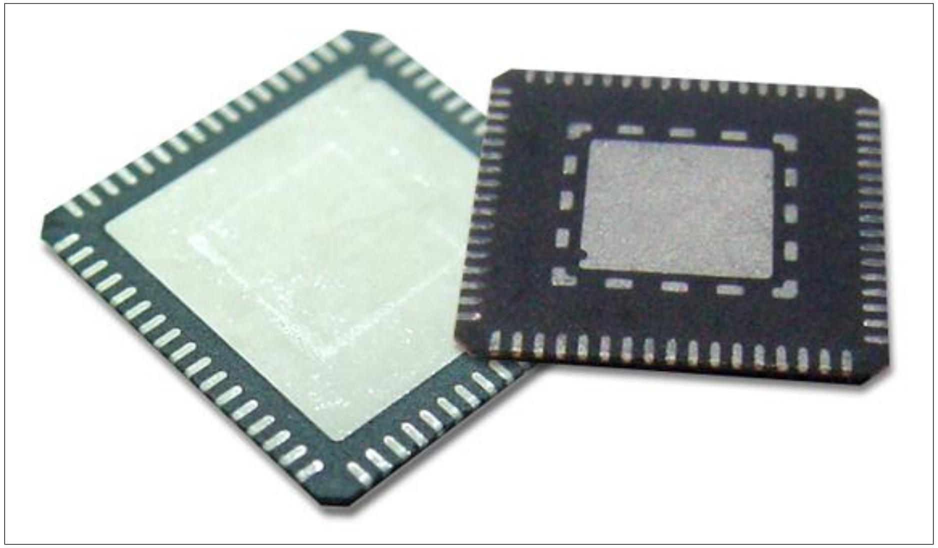
Industry Standardized Rework Processes for QFN/MLF and PoP Components

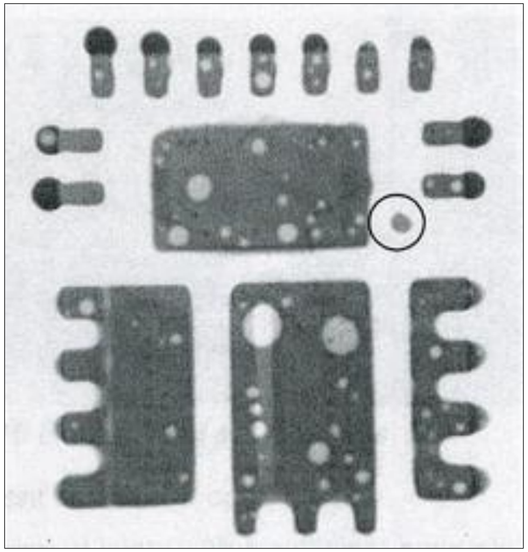
- Site redressing processes that prevent lifted pads, solder mask damage and copper pad dissolution.

- Development needed to understand the advantages/disadvantages of:
  - *Paste printing on board*
  - *Paste printing on component*
  - *Paste dispensing on board*
  - *Component paste dipping*
  - *Flux application techniques*

MLF/QFN Rework

Micro Lead Frame (MLF®)/Quad Flat No Lead (QFN) Rework





Voids and solder ball (10)

- Leadless package
- Exposed die-attach pad on the bottom efficiently conducts heat to the PCB
- Solder paste must be applied either to the site or device
- Key issues:
  - *Large voids (above right) in the thermal pad area from flux entrapment (IPC-A-610: 25% voiding)*
  - *Out gassing > splatter, solder balls from excessive solder coverage of the thermal pad*

MLF/QFN Rework

Typical Rework Process

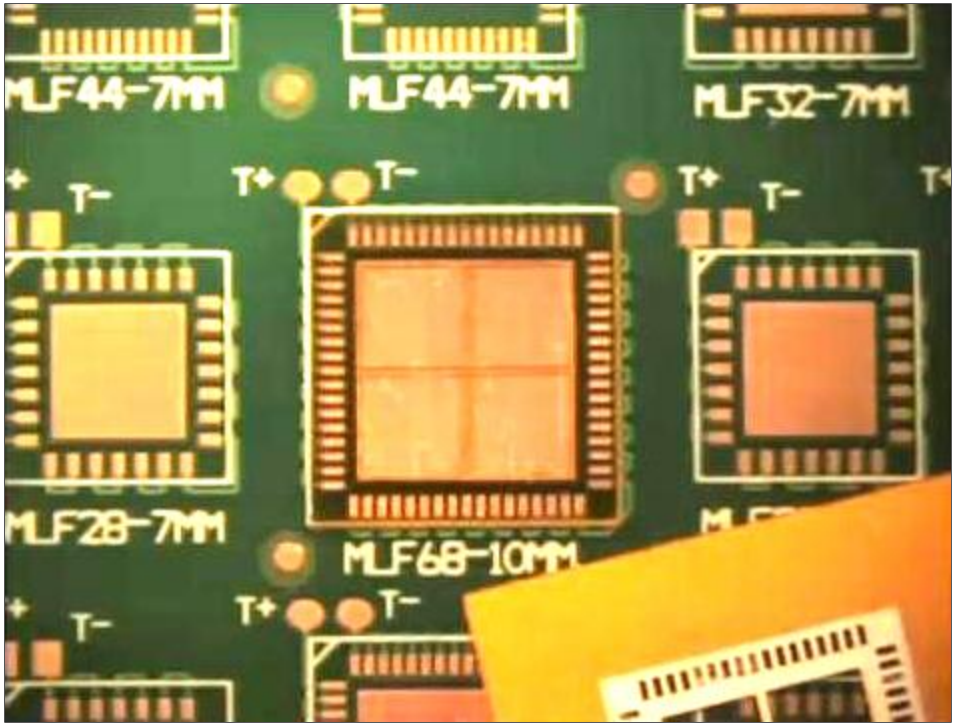
- Remove component (same as a BGA/PoP)
- Remove residual site solder (same as a BGA/PoP)
- Apply solder paste to site or component
- Align/Place/Reflow

- **Solder paste application is the critical differentiator for MLF/QFN Rework**

MLF/QFN Rework

Paste Application Methods

• Paste on site
(metal or tacky polyimide stencil)



• *Board must be cooled down to ambient temperature (reduced throughput)*
• *Adjacent component issues*
• *Errors > potential paste migration*

MLF/QFN Rework

Paste Application Methods

- Solder Preforms [11]

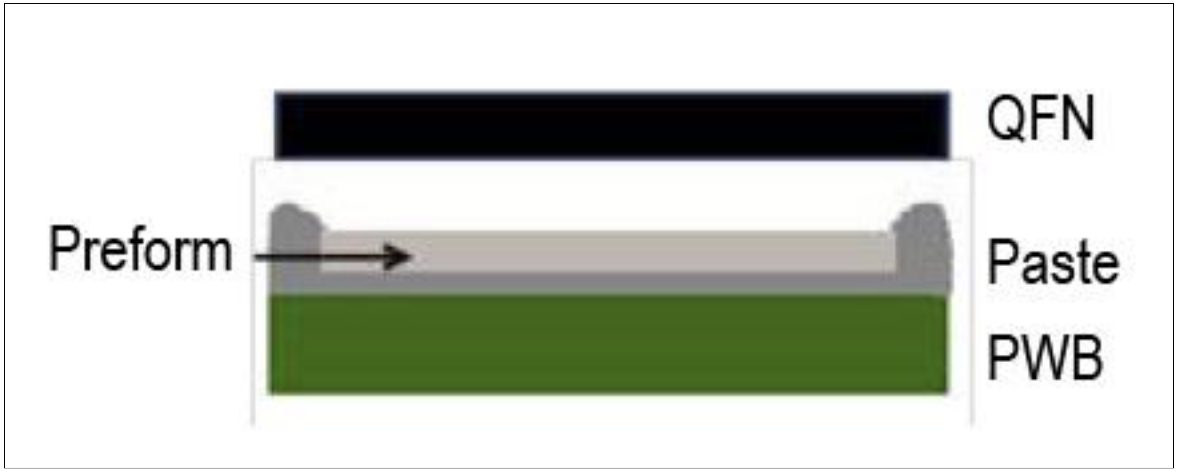
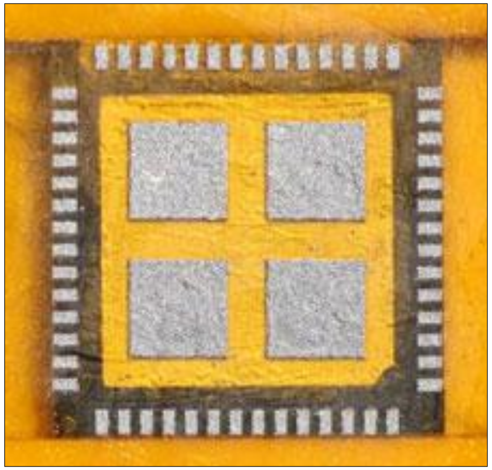


Illustration of a rectangular Solder preform placed on the solder paste.

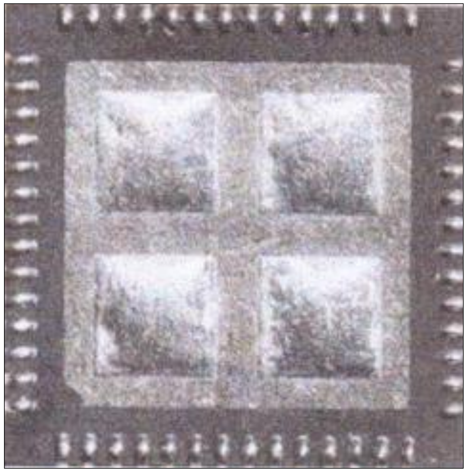
- *Solder paste must be applied to the site (issues)*
- *Preform pushed into the solder paste far enough to allow for the paste to contact the QFN*
- *Reduced flux volume provides reduced voiding*

MLF/QFN Rework

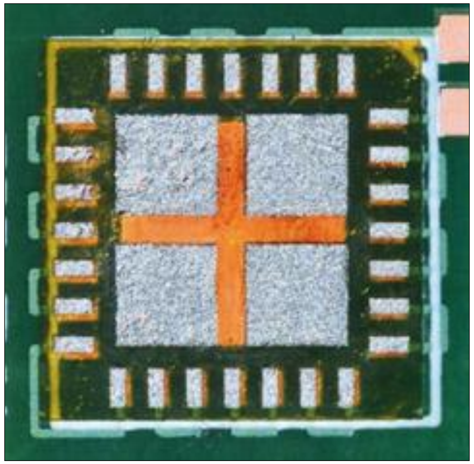
Paste Application Methods - Stay-in-place stencil [12]



Pasted QFN



Reflowed QFN



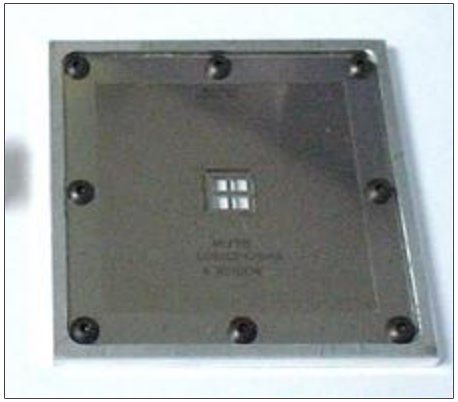
Stay-in-place site stencil

- *Polyimide stencil placed over the land patterns on the bottom of the device*
- *Solder paste applied, reflowed, stencil peeled off leaving a bumped device*
- *Stencil permanently affixed to the PCB filled with paste flux or solder paste*
- *Stencil acts as the receptacle for the bumped part and is then reflowed.*

MLF/QFN Rework

Paste Application Methods

- Paste on device stencil



- *Board can be kept at preheat temperature*

MLF/QFN Rework

Paste Application Methods

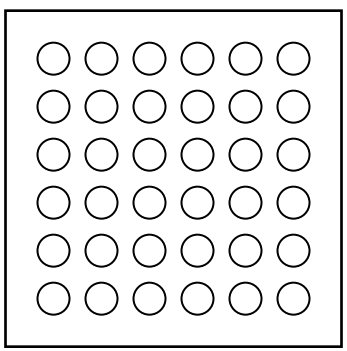
- Multi-Up paste on component stencil (with vacuum hold down)

Video Clip #8

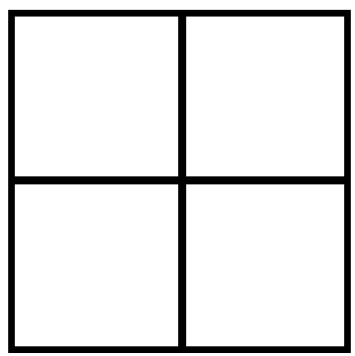
- *Board can be kept at preheat temperature*
- *Maximizes pasting throughput*
- *Pasted components are reflowed separately and stored*
- *Component can then be picked and flux dipped like a BGA*

MLF/QFN Rework: Amkor® Stencil Design Recommendations (13)

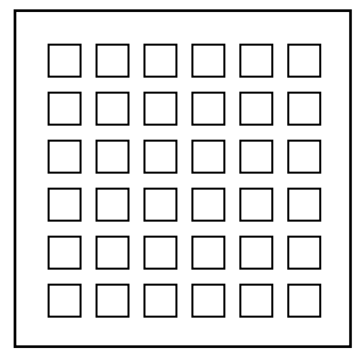
- 0.125mm (.005”) thickness (0.4 and 0.5mm pitch)
- 0.15 - 0.2 (.006 - .008”) thickness for coarser pitch parts
- Smaller, multiple openings on the thermal pad region (50-80% paste coverage)
- Three stencils with different center pad designs



1.00mm diameter circles
@1.2mm pitch - 50% coverage



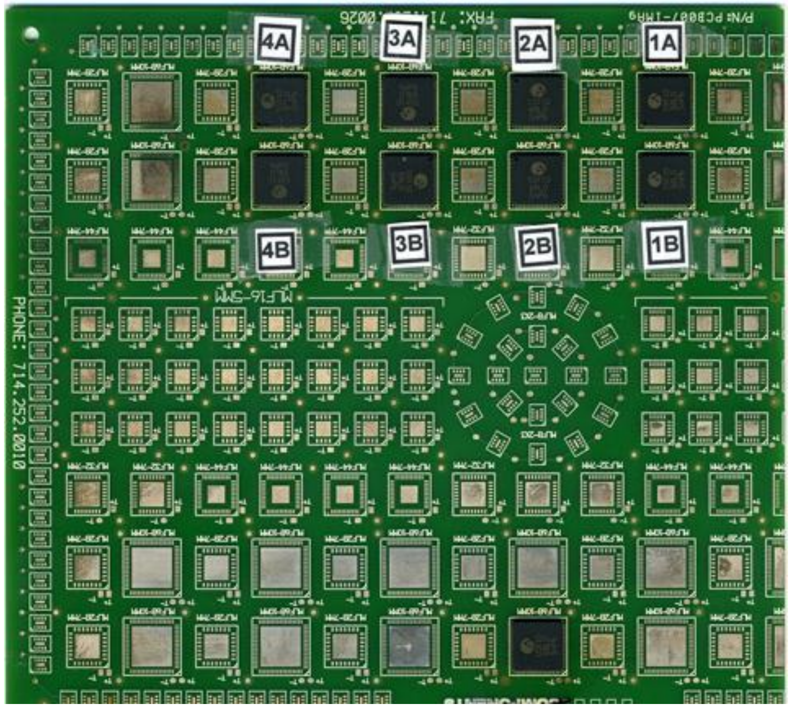
3.10 mm squares @
3.95 mm pitch - 60% coverage



1.35 x 1.35mm squares
@1.5mm pitch - 81% coverage

MLF/QFN Rework: Solderability Testing

- 203 x 140mm (8 x 5.5”) .062” thick test vehicle
- 10mm QFN 68 components (0.5mm pitch)
- Indium 9.0A no clean Pb-free solder paste (type 4 mesh)
- Two thermal profiles
  - *Short Soak*
  - *Long Soak*



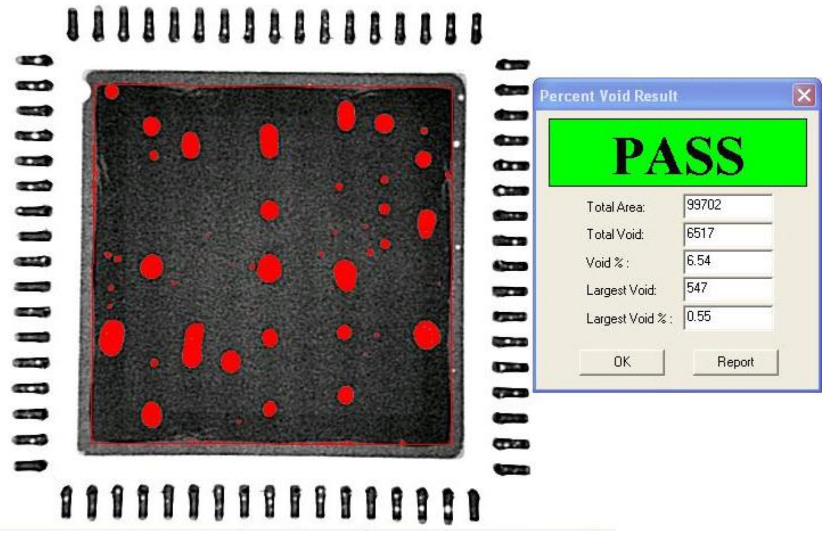
MLF/QFN Test Vehicle

MLF/QFN Rework: Solderability Testing

				Voiding (Center Pad)	
Stencil	**Location**	**% Coverage**	**Profile**	**Total**	**Largest Single**
36 Circles	1A	50%	Short Soak/Ramp	11.5%	1.6%
36 Circles	1B	50%	Long Soak/Ramp	6.5%	0.6%
4 Windows	2A	60%	Short Soak/Ramp	17.9%	2.0%
4 Windows	2B	60%	Long Soak/Ramp	10.5%	1.9%
25 Windows	3A	81%	Short Soak/Ramp	13.4%	2.4%
25 Windows	3B	81%	Long Soak/Ramp	11.3%	1.8%
4 Windows (1)	4A	60%	Short Soak/Ramp	13.3%	1.1%
4 Windows (1)	4B	60%	Long Soak/Ramp	16.1%	1.4%

(1) Using Multi-Up Stencil, Reflowed in Advance of Use, Flux Added to Site

- All eight (8) processes were well below the IPC standard of 25% voiding.
- The long soak stage provided lower voiding with all three stencil designs.
- Lower percent solder paste coverage combined with the long soak stage provided the lowest voiding.
- Pre-bumped QFN's has voiding comparable to pasted parts with the same stencil design.



Component Prep Alternatives Matrix

Approach	Advantages	Disadvantages
Manual Flux Application	• Fast • No custom tooling required	• Lacks volume control and repeatability • Not usable for some applications (QFN/MLF, CCGA’s, SMT connectors)
Flux Dipping	• Volume control and repeatability • Less custom tooling than stenciling	• Slower than manual flux application • Some custom tooling required
Paste on Site	• Maximum paste volume • Volume control and repeatability	• Board must be cooled (throughput) • Adjacent component issues • Difficult to fix errors
Paste on Device (Stencil)	• Fast/easy • Easy to correct error • Machine integrated pick capability	• Custom stencils required for each component • Fine pitch issues (0.4 > 0.3mm pitch) • Less paste volume then paste on site
Paste Dipping (Low viscosity solder pastes)	• Faster than paste on site or paste on device • Less custom tooling than either stencil approach	• Less paste volume than paste on site
Paste Dispensing	• ?	• Slowest process • Board must be cooled • Off machine process (typically) • Tail defects and volume repeatability issues

Rework Conclusions

Large Area Arrays (50mm+) on Large, High Thermal Mass Assemblies

- Expected to meet the same thermal targets as todays BGA’s
- Other challenges include alignment, warpage, nozzle vacuum and board holding capabilities

01005

- Hand soldering is difficult but possible
- BGA rework systems not a cost effective solution. Low throughput.
- “Man”/Machine Rework Interface provides the best of both worlds

PoP

- Significant issues with warpage/HiP
- Amkor’s new TMV™ PoP help to eliminate this issue

MLF/QFN

- Key issues include voiding in thermal pad region and out gassing
- Stencil design critical

Acknowledgements

Mario Scalzo (Indium Corporation)

- *Providing all solder pastes and fluxes used*
- *Technical insight*

Chuck Richardson (iNEMI)

- *Use of the 2013 iNEMI Roadmap; Rework*

Micro-Board Processing Inc. (MPI)

- *X-ray analysis for the PoP and MLF/QFN solderability tests*
- *01005 assembly*

Endicott Interconnect Technologies

- *PoP cross sectioning*

Don Morgenstern and Ron Wachter (Air-Vac Engineering)

- *PoP and MLF/QFN solderability tests*

Co-workers at Air-Vac Engineering who helped me with this paper