

Through-Hole Assembly Options for Mixed Technology Boards

Ross B. Berntson, Ronald Lasky, Ph.D., PE, and Karl P. Fluke
Indium Corporation of America
Utica, NY

Keywords

Solder Preforms, Pin-In-Paste, Intrusive Reflow, Mixed Technology, Selective Wave Soldering, Through-Hole Connectors

Abstract

Surface mount assembly has dominated its through-hole predecessor since the early 1990s. The higher density and lower ultimate cost of SMT makes it a preferred assembly technology. However, the mechanical strength of through-hole connections continues to make through-hole the technology of choice in assembling connectors. This presentation will describe the primary methods currently used for through-hole connector assembly: 1) selective wave solder, 2) pin-in-paste (PIP),¹ reflow, 3) hand soldering and 4) solder preforms. We will show how solder preforms are an excellent alternative when PIP provides insufficient solder.

The wave solder method requires specialized equipment and processes to solder connectors. Pin-in-paste reflow evolved as a way to accomplish through-hole assembly without additional equipment or process steps. In the PIP method, the additional solder required to fill the through-hole barrel is deposited by overprinting the pad in the area of each connector pin, using standard SMT equipment. During reflow, the solder wicks to each pin forming the solder fillet.

This paper explains why pin-through-paste reflow methods based on overprinting solder paste have become more challenging due to an increasing use of Organic Solderability Preservative (OSP), fine-feature devices (e.g. fine pitch connectors) and densely populated PCB layout designs that conflict with requirements for successful use of step-stencils. This paper also shows an example where solder preforms were used to provide extra solder volume for each pin. This work demonstrates how solder preforms provide a viable manufacturing solution to ensure complete through-hole solder joints.

Introduction

The design engineer's task to determine the best board configuration for assembly is made increasingly difficult by tighter component densities, changing board thicknesses, more fine-pitch devices, changing surface finishes, greater power demands, increasing reliability concerns, Pb-Free, and much more. All of these challenges are aggravated by ever increasing demands for *lower costs*, which can be translated into greater throughput, fewer processes, less equipment and higher yields. A key obstacle to successful design and manufacture of mixed technology boards (surface mount technology, SMT and pin through hole, PTH components) is achieving an acceptable through-hole component solder fillet with reasonable yields, costs, and process development. The many tools and tricks available to the design and process engineer to overcome this obstacle will be outlined in this paper with particular attention paid to novel solutions (Table 1).

Table 1 - Options for Assembly of Through-hole Components on Mixed Technology Boards

Technique for Mixed Technology		Pro	Con	When Applicable	Other Comments
WAVE PROCESSES	Wave Solder (may include glue & wave)	<ul style="list-style-type: none"> • Low material costs • Established process in many plants 	<ul style="list-style-type: none"> • Consistent yields are difficult to achieve in wave solder processes • Additional process & equipment to maintain 	Large, single sided SMT with numerous connectors	
	Selective Wave – pallets	<ul style="list-style-type: none"> • Low material costs • Utilize existing equipment • High volume of solder for thick boards & mechanical connections 	<ul style="list-style-type: none"> • Difficult to achieve high yields • Not flexible – board specific pallet • Cost and maintenance of pallets (cleaning) • Additional process & equipment to maintain 	Two sided SMT boards with edge through-hole devices	Common on computer backplanes
	Selective Wave – solder fountain with XY table	<ul style="list-style-type: none"> • Low material costs 	<ul style="list-style-type: none"> • Unproven • High start-up costs • Start-up time - programming • Additional process & equipment to maintain 	Uncertain	Many facilities experimenting with systems
PASTE PROCESSES	Pin-In-Paste (PIP) – overprint only (may be combined with a step-stencil)	<ul style="list-style-type: none"> • Proven process • High yield • Utilize existing equipment • No additional processes or equipment to maintain 	<ul style="list-style-type: none"> • Large quantity of flux residue – may impact ICT • Difficult to design with increasing component density • Transition from HASL to OSP may diminish solder volume • Increase maintenance of placement equipment (solder paste tails) 	Thin PCBs that have minimum barrel fill requirements and low-density boards with sufficient overprint space.	Common for mobile products and computer peripherals. When combined with a step stencil, this process has been observed at high reliability manufacturers such as automotive systems and high-end computers.
	Pin-In-Paste (PIP) – overprint + dispense	<ul style="list-style-type: none"> • Additions of solder available to solder joints after component placement and on 3D board surfaces 	<ul style="list-style-type: none"> • An even larger quantity of flux residue expected • Additional process & equipment to maintain 	Niche applications where board space is not available to permit sufficient overprint or a hold-out area for a step stencil.	Not common – observed at some power converter and computer manufacturers
PREFORM PROCESSES	Solder Preforms (may be combined with a flux coating)	<ul style="list-style-type: none"> • Exact amount of solder required for an optimum fillet can be applied to each connector pin • Minimum flux residue 	<ul style="list-style-type: none"> • Adds cost to the BOM • May require a fixture to retain perform in correct location 	May be necessary if an exact amount of solder with minimum flux residue is required.	Common in military, automotive and power device applications. Connectors are available with preforms already attached.
	Solder Preforms + Solder Paste	<ul style="list-style-type: none"> • Exact amount of solder required for an optimum fillet can be applied to each connector pin • No additional flux required • Utilize existing equipment • Minimizes the extent of overprint • Eliminates the need for Step-Stencils 	<ul style="list-style-type: none"> • Adds cost to the BOM • May add time to placement operation 	Excellent solution for high-reliability mixed technology boards with many fine-pitch devices.	Common in numerous markets where connector strength is critical and overprinting or step-stencils are not possible.
Mechanical	Solder Paste + Glue/Clip/Screw	<ul style="list-style-type: none"> • A mechanically strong joint can be achieved with an incomplete barrel fill. Solder only provides the electrical connection 	<ul style="list-style-type: none"> • Adds cost to the BOM • Additional process & equipment to maintain • Additional material to manage 		Not common but observed.
Single Point Processes	Robotic Soldering with Solder Wire	<ul style="list-style-type: none"> • Flexible solution • Low material costs 	<ul style="list-style-type: none"> • Throughput is diminished because only one pin can be soldered per operation • Start-up time - programming • Additional process & equipment to maintain 	High mix, low volume assemblies with all pins accessible to robot's heat source and wire.	High-mix, low volume solutions or where only limited points need additional solder.
	Hand Soldering with Solder Wire	<ul style="list-style-type: none"> • Flexible solution • Low material costs 	<ul style="list-style-type: none"> • Labor cost • Through-put cost • Difficult quality control 	High mix, low volume	Most engineers avoid this option.

Objective of Through-hole Soldering

An acceptable through-hole solder technique must produce a quality solder joint as defined by *ANSI/J-STD-001C* and pictured in *IPC-B-610*. Most flux systems whether in solder paste or wave systems are capable of delivering acceptable wetting as defined by wetting angles and pull tests. Consequently, for this discussion we will focus on the minimum acceptable criteria for fillet formation for Class 3 assemblies. The criteria for barrel fill and fillet size is surprisingly lenient.

ANSI/J-STD-001C specifies 270° circumferential wetting on solder destination side (for PIP, bottom side) and 330° circumferential wetting on the solder source side (for PIP, top side). In addition, only 75% of the barrel vertical dimension needs to be filled. These standards permit a solder joint that is not dissimilar to the joints depicted in Figure 1.

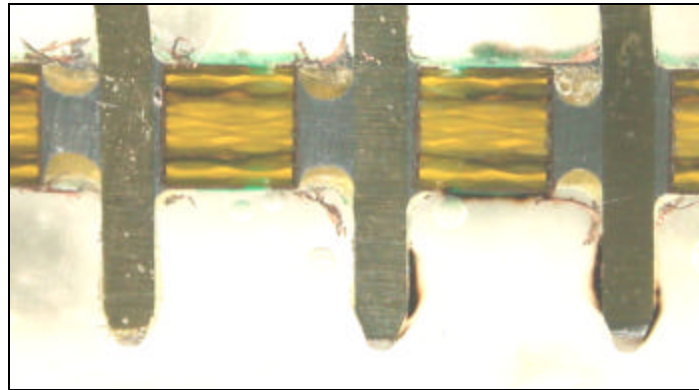


Figure 1 - Worst Case Solder Fillets from a PIP Process

However, these standards do not account for the important criteria of inspection. Although the above solder joints may be reliable, without cross-sectional analysis no joint is visible due to the pooled flux residues. Because no-clean processing is still the dominant technology, this inspection difficulty is not easily overcome. In addition, connectors often have unique mechanical stress requirements not foreseen during the development of the JStandards. These mechanical stresses can exacerbate fatigue failures on thin boards and connectors with few PTH pins. Consequently, an operational standard for most manufacturers is to meet the *ANSI/J-STD-001C* specifications while also producing a solder joint that is readily inspected by line operators. An example of these ideal joints is depicted in Figure 2.

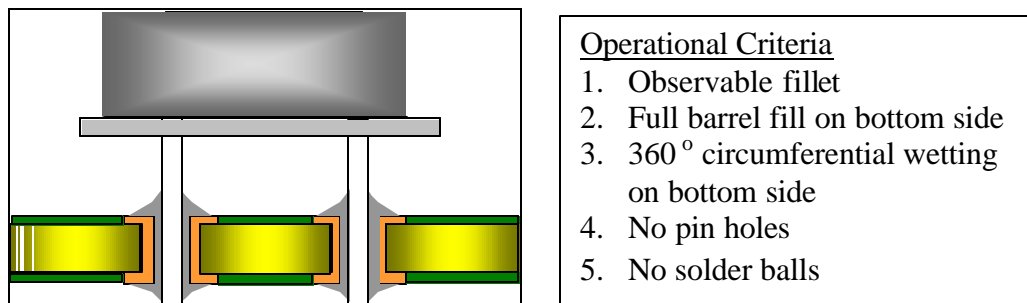


Figure 2 - The Objective is to Deliver a High First Pass Yield with IPC Conforming Through-hole Solder Joints and Fillets that Can be Inspected

Wave Solder Processes

Wave soldering remains an important solder technology despite its perceived and real challenges. Even wave soldering's detractors admit that it is still an excellent low-cost solution for boards with all or mostly PTH components. In addition, many devices we use every day have wave soldered PCBs including our recently purchased toasters, baby monitors and children's toys. This low-cost technology's advantages diminish as the ratio of PTH components to SMT components decreases. The exact crossover point is much debated and is very situational. The debate is more complex when evaluating available processing equipment versus purchasing and installing new equipment. A decision tree used by a current PCB assembler included the following variables:

1. PTH component count versus SMT component count
2. Single-sided or double-sided SMT components
3. Adequacy of existing wave equipment infrastructure
4. Space availability
5. Operator expertise
6. Program yield and through-put requirements

Three primary solutions have been developed to extend the wave solution to more complex assemblies. (1) For assemblies with only a few SMT components on the bottom-side of the board, glue can be used to hold the components in place prior to wave soldering. To avoid insufficient solder (shadowing) on these components, a glue and solder paste process may be required prior to the wave solder step. (2) The use of pallets that shield bottom-side SMT components is very common

especially when only the edge connectors require PTH soldering. This solution is increasingly difficult if the PTH components are dispersed throughout the PCB. An example of a typical pallet with only edge connector location exposed is depicted in Figure 3. (3) Selective wave solder machines are becoming more common as assemblers tackle the challenges of more complex, mixed technology PCBs. These systems incorporate an XY(Z sometimes) table that manipulates the PCB over a focused flux spray and a pin-point solder fountain. These systems offer the advantage of flexibility and the ability to expose only desired regions of the board to flux, solder and heat. However, the authors' experiences with these systems at PCB manufacturers are mixed. Equipment costs, start-up programming, space and process variability are mitigating some of the benefits. As this technology is proven to be robust, it will become an increasingly attractive option.

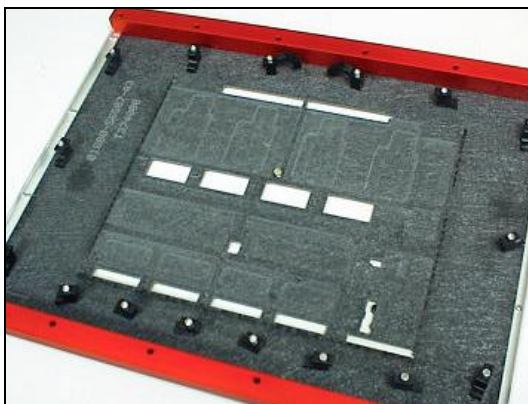


Figure 3 - Only Edge Connectors are Exposed to the Wave Flux and Wave Solder
Photo courtesy of QMIN
<http://www.qminx.com/Wave.htm>

Solder Paste Processes

The typical PIP process as developed by McLenaghan and others² is incorporated directly into the SMT process. PIP offers the important advantages for mixed technology boards with double sided SMT and/or limited PTH requirements. These advantages include high throughput, current process & equipment utilization and good yields. This PIP process requires the following:

1. The stencil will be designed such that the correct amount of solder paste can be printed onto the through-holes to form an acceptable joint after reflow.
2. The solder paste will be amenable to the PIP process (usually Type 3 powder).
3. The stencil printing process should be optimized.
4. The through-hole components will be selected such that they can withstand reflow soldering temperatures.
5. The through-hole component leads should be rounded at the end.
6. The through-hole component leads should be of the correct length to form a good solder joint.
7. The through-hole components will be mounted correctly.
8. The reflow profile will be acceptable.

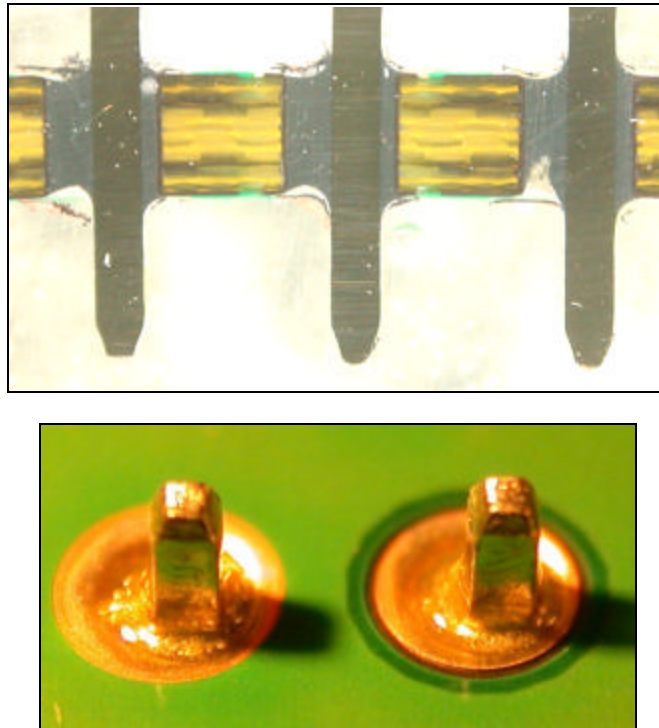
The greatest challenge in the PIP process is to design a stencil that delivers an adequate amount of solder paste to the through-hole component. The geometry of an acceptable solder joint as depicted in Figure 2 can be used to estimate the required solder paste volume. Solder paste is 50% by volume solder and 50% by volume flux vehicle. Consequently, the solder volume required for an acceptable fillet is only 50% the volume of solder paste required from the printing process. Solder pastes rheology considerations, such as transfer efficiency, barrel fill and tackiness, are critical to success. Pressurized print systems, such as ProFlowTM (DEK) and the RheopumpTM (MPM), can improve barrel fill.

The PIP process has been utilized by numerous manufacturers and many of the shortcomings and tricks to success have been noted. Some of these include:

1. High metal load solder paste (>95%) – Although this solder paste is exceedingly thick, it can achieve adequate transfer efficiency with a pressurized print system or multiple blade strokes.
2. Step-stencils – When overprinting does not deliver adequate volume, a step stencil from >10mil thick to <6mil thick can be achieved. Solder paste transfer efficiency is a concern in these situations. Also, sufficient hold-out space to accommodate the stencil non-planarity is required to avoid bridging or shorts.
3. Dispensing solder paste – When residue levels are not a concern, dispensing solder paste can overcome printing limitations. As stated in Table 1, process and equipment support is a consideration.

4. HASL vs. OSP – HASL may deliver 200 – 500 micro-inches of solder to the joint, which translates into 1mil of solder paste deposited. Although the HASL coat is thinnest at through-hole locations where blow-through is possible, an untested switch to OSP is not recommended. The wetting characteristics of OSP may also diminish wicking into the barrel. Note that many assemblers are using OSP with PIP processes.
5. ICT Test Point – The large quantity of residue may limit the effectiveness of ICT and may require test points or the use of a probe-testable solder paste. The introduction of test points may further constrain the available overprint area.

A typical solder joint from an overprint with a 4mil step is depicted in Figure 4. Note that the solder joints still have significant cavities, and flux residue limits inspection. In addition, the manufacturer of this assembly increased the overprint to such an extent that large solder beads can be found at via locations when the solder coalescence is not adequate to pull the solder into the joint (Figure 5). We considered increasing the solder paste barrel fill during printing, but our experience indicated that most of this solder paste is pushed out of the hole and the pin design could not hold this paste through the rest of the process. A redesign of the connector was not an option.



**Figure 4 - Solder Joints Formed from PIP Process with an Overprint and Step-stencil
Note the Existence of Cavities and the Extensive Flux Reside**

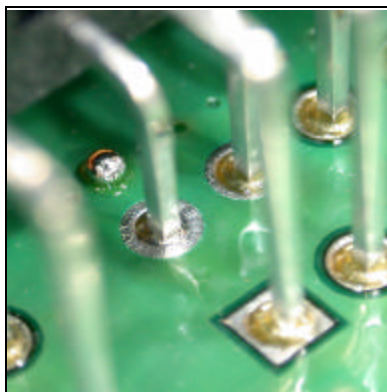


Figure 5 - Solder Bead Caught at Via Location Due to Excessive Solder Overprint

New process development was required to eliminate defects associated with the current process, increase yields, ease inspection and improve quality while controlling costs. The solution was a combination of *solder preforms* + *solder paste*.

Solder Preform Processes

Solder preforms are stamped from rolled solder ribbon to the desired XY dimensions (Figure 6). Preforms are available in numerous shapes, sizes and alloys and are specifically designed to deliver an exact quantity of solder to a desired location. Examples of preform shapes can be found in Figure 7

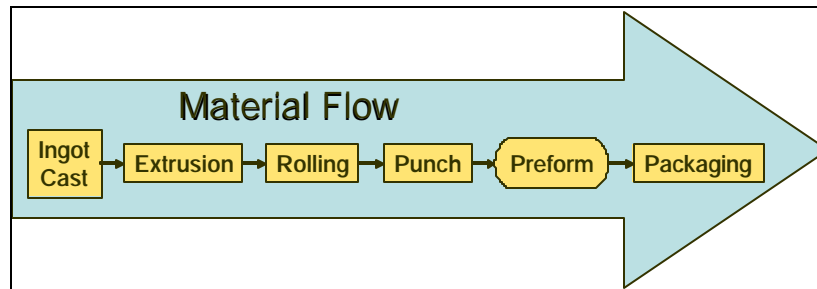


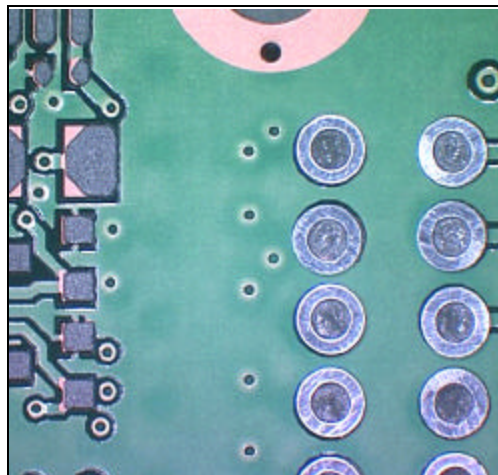
Figure 6 - Solder Preform Manufacturing Process



Figure 7 - Solder Preforms are Available in Many Shapes and Sizes

The solder preform process for through-hole connectors on mixed technology boards utilizes current equipment and process expertise.

- Step 1: Solder paste is printed at the site of the connector pin. The solder paste deposit may be an overprint to provide a wicking path for the preform. For example, a preform can be placed in the overprint portion and during reflow the preform solder will wick to the pin. Additional flux is not required because sufficient fluxing activity is available in the solder paste.
- Step 2: Preforms are placed in the solder paste using automated placement equipment. The preform shape may be a washer or a 0603 segment as depicted in Figure 8. The 0603 shapes are adaptable to high speed placement from either tape-and-reel packaging or bulk feeding systems (Figure 9).



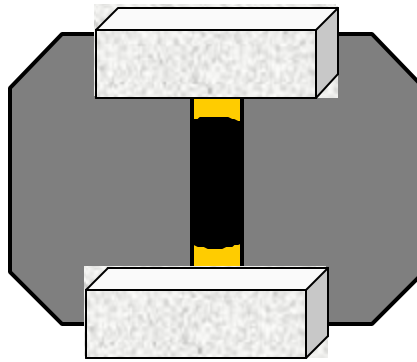


Figure 8 - Solder Preforms Placed in Solder Paste. The Washers are Placed Directly Over the PTH Site and the 0603 Segments (blocks) are Placed on both Sides of the Pin



Figure 9 - Solder Preforms are Available in Tape-and-reel and Bulk Packaging that Integrates with High Speed Assembly

The manufacturer that supported this work was able to achieve excellent results with the *solder preform + solder paste* solution. The benefits were:

- Full barrel fill that is easily inspected (Figure 10)
- No loss of through-put
- Elimination of step stencil with reduced defects at fine-pitch devices
- Reduction of overprint which eliminated solder beads and balls
- Increased flexibility for future designs

Variations of this solder preform process may be appropriate for some assemblies. Three important variations are:

1. Linked array preforms – To minimize placement steps, especially in semi-automated scenarios, a linked array can be used. The webbing that holds the solder deposits in the correct configuration melts during reflow and wicks to the pins. Examples of linked array preforms are depicted in Figure 11.
2. Solder preforms with no solder paste – The solder paste is necessary if tackiness is required to hold the preform in the correct orientation. Where fixturing is possible, flux coated preforms can be directly applied to the solder location.
3. Connectors preloaded with preforms – Several connector manufacturers offer preforms preloaded onto connectors. The preforms can be crimped to each individual pin.

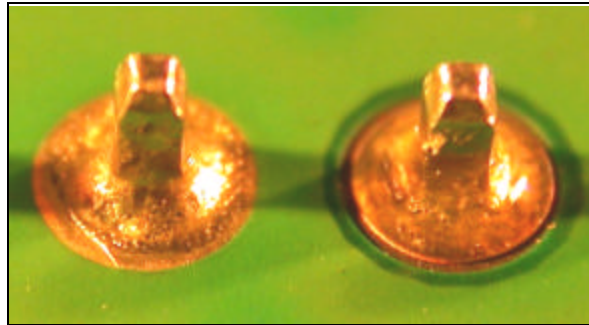
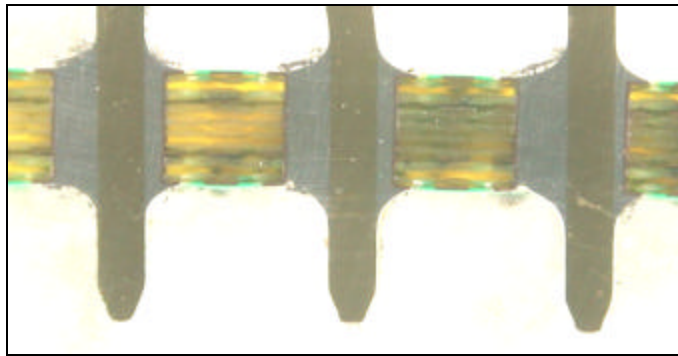


Figure 10 - The Solder Preform + Solder Paste Solution Delivered Excellent Through-hole Solder Joints

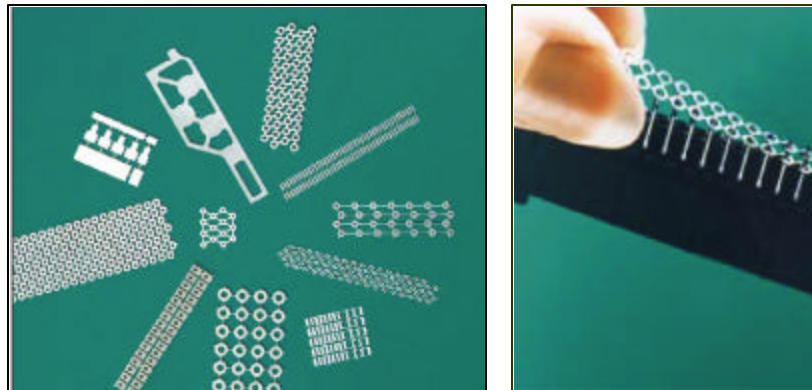


Figure 11 - Linked Array Preform Examples

Mechanical Processes & Single Point Processes

Solutions to many mixed technology challenges can be relatively simple improvements to standard techniques. For example, many manufacturers have some touch-up locations for hand soldering. This process can be automated with advanced robotic solutions. When the connector requires more mechanical strength, glue, clips or screws might be a better solution than increasing solder volume. However, with all these options, additional materials and processes may be required.

Conclusion

Mixed technology circuit boards will continue to challenge design and process engineers. Fortunately, the toolbox is full, and by working creatively with suppliers, an optimum solution can be found. Novel solutions such as *solder preforms + solder paste*, *linked array preforms* and *selective solder machines* should be considered for new designs and troubling current assemblies.

Acknowledgements

We thank our customers who supported this project with expertise, time and materials.

Authors

The authors welcome your questions and comments. We can be reached at Indium Corporation of America: 1-800-4-INDIUM or our email addresses RBerntson@indium.com, RLasky@indium.com, KPfluke@indium.com.

References

1. Lasky, R. C., Jensen, T. J., *Practical Tips in Implementing the “Pin in Paste” Process*, SMTAI, September 2002, Chicago, IL.
2. McLenaghan, A. J., Creyr Innovations, ajm@creyr.net.