s0201 Process and Yield Improvement During Launch to Production

Jason Fullerton
Metrologic Instruments
Blackwood, NJ

Introduction

Part miniaturization is inevitable in surface mount assembly. With each shift to smaller component types, new critical process parameters emerge and old process parameters become significantly more important. New failure modes emerge and challenge the manufacturing engineer to adapt existing equipment and design parameters to maximize yield as assemblies incorporating these miniature components are introduced to current manufacturing processes.

The use of 0201 passive components has significant advantages for PCB designers; part density can be increased and PCB size can be decreased. In conjunction with incorporating 0201 passive components, microBGA components also are attractive complements in achieving these goals. However, careful consideration of land layout, surface finish, and solder stencil apertures must be undertaken to ensure high yield assembly. The manufacturing engineer must ensure that proper stencil materials and fabrication processes are specified. Existing board support methods must be examined and strategies modified or improved. Equipment must be examined for the capability to process miniature components. Additional tooling may be required for both handling and placement of miniature components. Non-standard programming techniques may be required to increase component handling rates and placement yield.

PCB Assemblies Utilizing Miniature Components

This paper focuses on the optimizations performed on one particular assembly that is installed in a small form factor barcode scan engine. The PCB as processed during SMT assembly is a 96-up panel with overall dimensions of 279.4mm x 279.4mm x 0.79mm on standard FR-4 laminate. Each individual image is 24.50mm x 14.25mm and incorporates double-sided assembly on a 4 layer circuit design.

In addition, a test PCB assembly exists that has contains a 10 x 10 array of 0201 components on both sides, and microBGA components on the second pass assembly side. This PCB is a single image design, with overall dimensions of 279.4mm x 279.4mm x 1.58mm, on standard FR-4 material incorporating 4 internal copper layers.

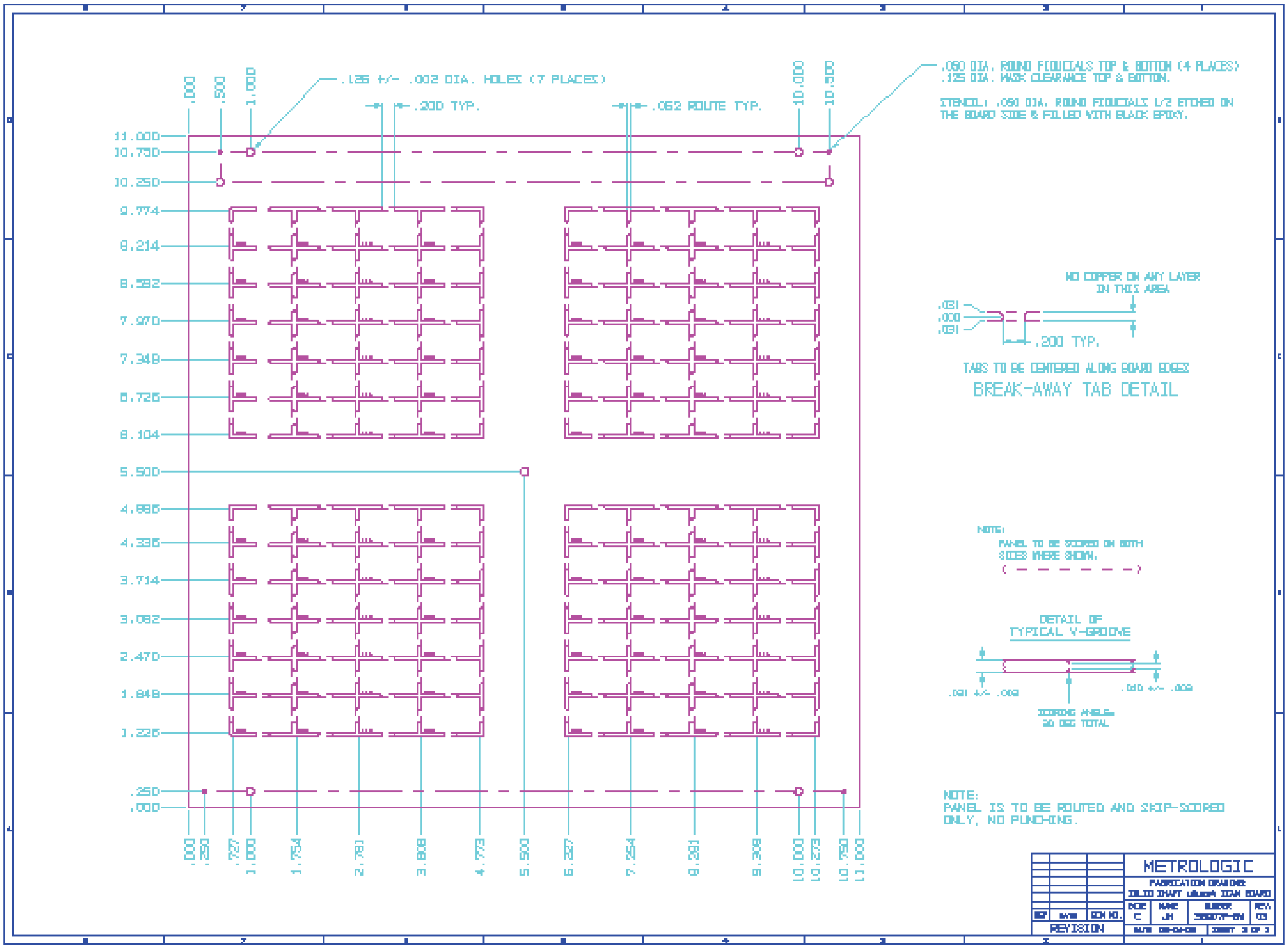

Figure 1 – Panel Drawing (dimensions in inches)

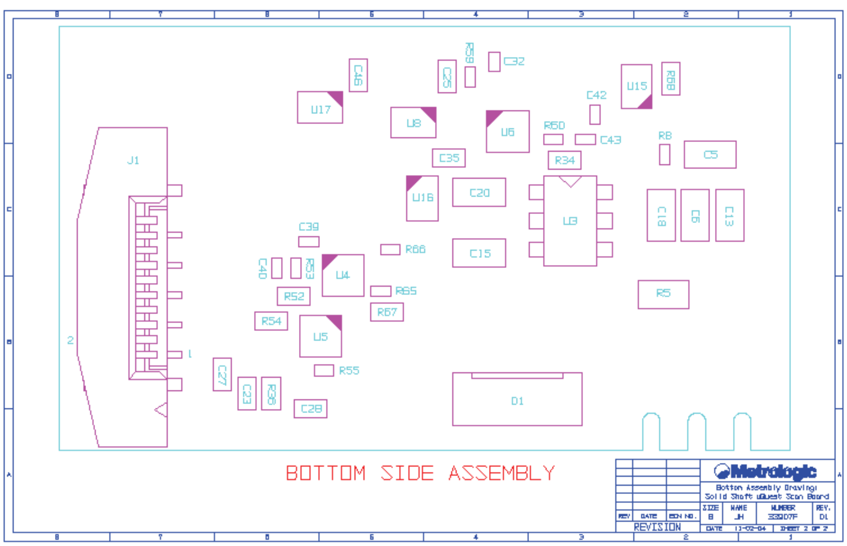

Figure 2 – PCB Bottom Side Assembly Drawing

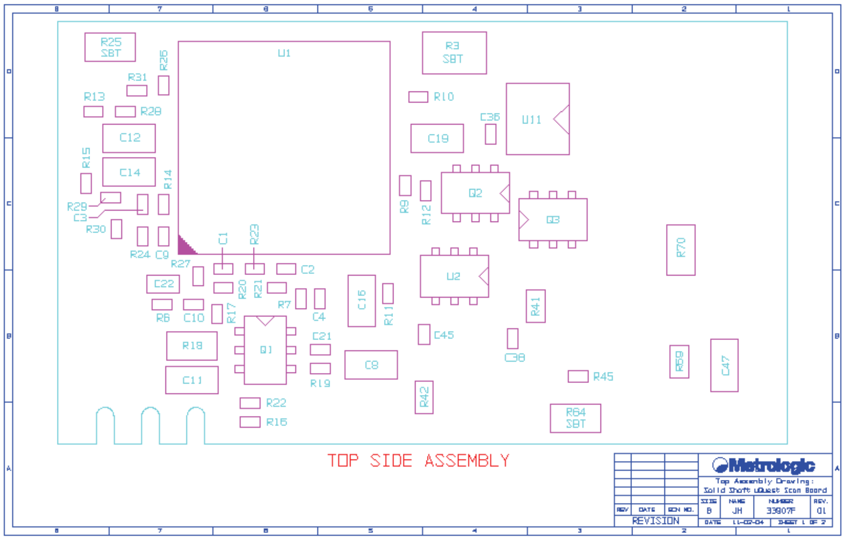

Figure 3 – PCB Top Side Assembly Drawing

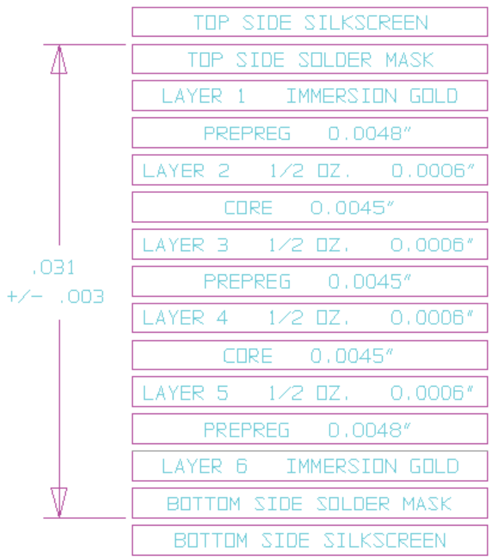

Figure 4 – Layer Stack Specification (dimensions in inches)

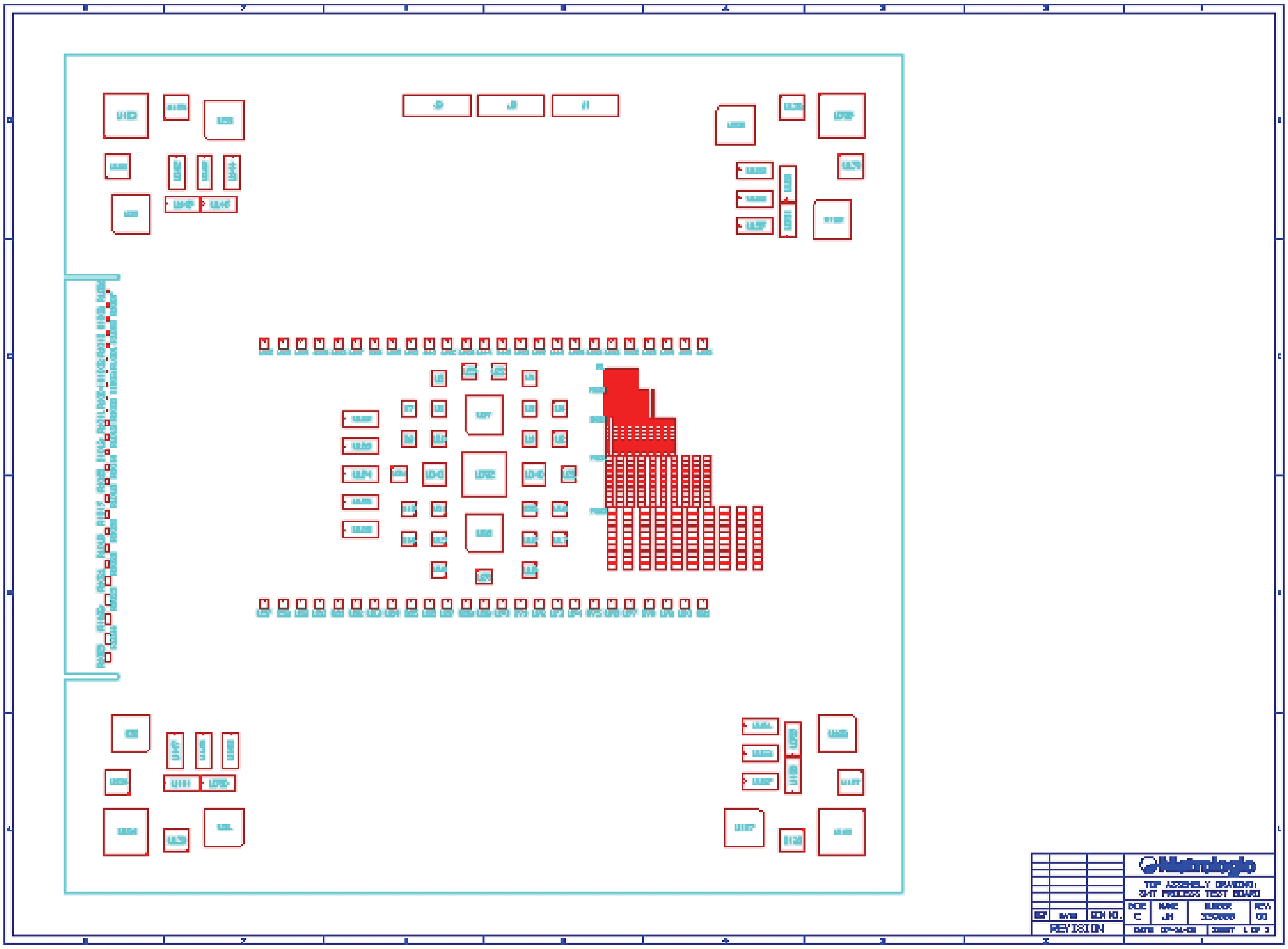

Figure 5 – Process Test Board

PCB Design Considerations

It almost goes without saying that the first area that impacts the yield of PCBs is design. This is especially important when incorporating miniature components. There are several areas where careful attention to design criteria can increase the likelihood of successful high yield manufacture.

The layout of PCB lands for 0201 components can have significant impact on yield rates through solder reflow. The land pattern originally incorporated on this product was generated by an industry standard land pattern generating program. During prototype and process development builds, a high rate of opens caused by components standing on one end, commonly referred to as tombstone defects, was incurred. During development builds tombstone defects were the top defect mode, representing over 50% of all defects. The land pads were spaced at 0.254mm (0.010”) and sized 0.305mm (0.012”) wide x 0.229mm (0.009”) long. An alternative land pattern[1] was incorporated for the production release revision of the PCB. The new land layout is spaced at 0.229mm (0.009”) and the lands are 0.381mm (0.015”) x 0.305mm (0.012”). Tombstone defects experienced since implementation of the new land pattern are rare and generally attributable to other special causes.

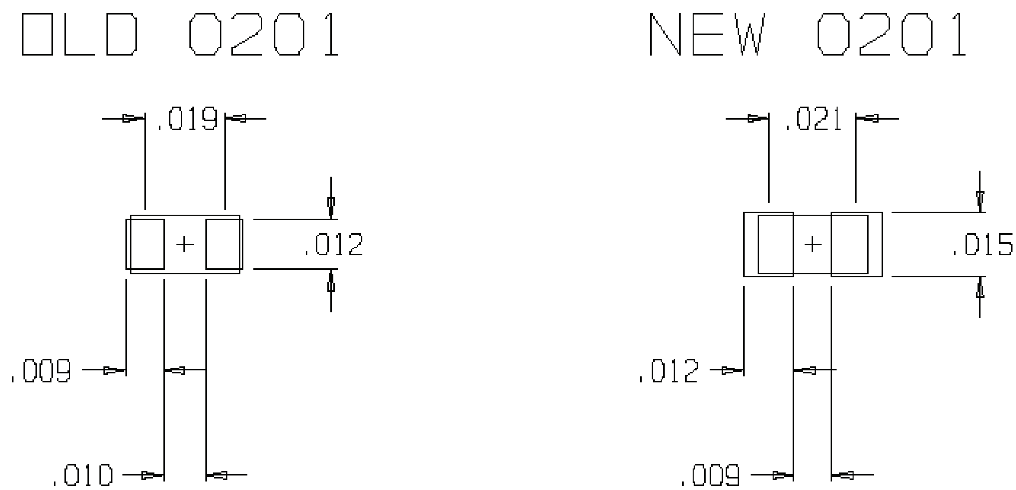

Figure 6 – 0201 Layout Comparison (dimensions in inches)

MicroBGA components require even more attention to surface mount pads because, more importantly than assembly defect rates, component reliability is significantly affected. Solder joints are not directly inspectable for workmanship. The component package type implemented on the scan engine circuit board is used in three locations. It attaches with eight interconnects in a 3x3 grid with the center location missing. The terminations are spaced at 0.5mm and are constructed with 0.16mm – 0.18mm diameter solder bumps.

The manufacturer of this component has published a detailed application note[2] regarding the various important criteria for a reliable 2nd level interconnect between the PCB and the component. Among the recommendations is the use of non-surface mask defined (NSMD) lands by designing a solder mask relief around the land. Compared to surface mask defined (SMD) pads, NSMD pads take advantage of the tighter tolerances for the etch process used by PCB manufacturers to form the lands and traces. This also facilitates the transfer of supply to low cost, long lead suppliers when regular production launches.

Stencil Design Considerations

The standard rules apply to stencil apertures at the sizes required for 0201 components and micro BGAs. Area ratio (the ratio of the surface area of the aperture walls to the area on the PCB exposed through the stencil) is directly related to the transfer efficiency of the solder paste through the stencil. Recommended minimum acceptable area ratio is 0.6 for laser cut stencils[3]. Chemically etched stencils require a significantly higher area ratio and are not suitable for use with miniature components.

The original apertures used with the early pad design were 1:1 with the lands and resulted in an area ratio of 0.64 on a 0.102mm (0.004") thick laser cut stencil. The result was significant failures at solder paste inspection as well as non-solder defects after reflow during development builds. Another result of these small apertures was increased clogging which would not clean with understencil wiping. This required stopping the printer to remove the stencil and clean, followed by setting the equipment up again. The clogging occurred as early as the second print. This scenario is certainly not suitable for regular production build volumes.

Also resulting from the same source as the new land design was an improved aperture design: 0.280mm (0.011") x 0.356mm (0.014") with an area ratio of 0.77. Accompanying the new aperture was an increase is solder paste inspection yields and reduced non-solders. In addition, clogging was reduced to a manageable level and second wipes of clogged apertures regularly resulted in clearing the aperture to an acceptable level. The microBGA manufacturer also recommends an optimized stencil aperture. The aperture is 0.300mm square which results in a 0.74 area ratio on the same size laser cut stencil.

An electroform stencil was tested using the 0201 patterns on the Process Test Board. Electroform stencils are manufactured using an additive process and are comprised of nickel, as opposed to laser cutting's removal process from a preformed sheet of stainless steel. The resulting aperture is smoother than a cut opening, allowing much smaller area ratios to print properly. The electroform stencil used the same 0201 aperture design described above but was manufactured with a 0.127mm (0.005") thickness. The resulting area ratios of 0.62 (for 0201) and 0.59 (for microBGA) are well within the process window for electroformed stencils, but at the lower end if not outside the window for laser cut stencils. The resulting paste transfer efficiency performance was comparable to the laser cut stencils with the added advantage of 25% more paste volume due to the extra thickness.

Solder Paste Printing Process Considerations

Process parameters for solder paste printing are dependant on the equipment being utilized as well as the brand of paste incorporated. Most paste manufacturers offer a variety of formulations and miniature components require the formulation that is targeted at "fine pitch printing". Generally this is accomplished through flux formulation and viscosity of the overall mixture. Speeds and pressures should be set to conservative settings out of respect of the challenging apertures present on the stencil. Metal squeegee blades are recommended over polymer blades due to the increases sharpness of the print that results over the stencil.

Specifically, this product is manufactured with a Type III paste and the printer uses metal squeegee blades. The pressure setting is at the low end of the recommended range for the paste and the speed is also at the lower end of the recommended window. These conservative settings have proven to be reliable for both first print performance as well as over the duration of a production sized lot.

Paste inspection is also important to detect unprinted pads before they are passed to subsequent processes. As the microBGA apertures are the most challenging for the printing process as well as most difficult to inspect post-reflow, they should be the focus of inspection to the extent that is possible. In addition, inspection of as many 0201 pads as possible should be incorporated. The amount of inspections possible is limited by the time required on the equipment being utilized. Inspecting every micro BGA or 0201 pad is not feasible if it increases the cycle time of the printing step to the point that it constrains the entire process. In this case, the process engineer needs to determine an inspection strategy that maximizes the number of pads inspected as well as provides good coverage of the entire PCB (both across its length as well as its width).

Component Placement Process Considerations

A Sanyo TCM-3500Z surface mount assembly machine is utilized in the assembly of the printed circuit board subassemblies. A variety of equipment is available on the market that are capable of placing miniature components, but the users must contact the original manufacturer of the equipment to determine if any modifications or upgraded hardware are recommended. This particular chipshooter involves a stationary rotating turret transferring components from standard reels of pocketed paper tape to the printed circuit board on a moving table. Both the nozzles that transport the components and the feeders that hold the tape are specially designed for the purpose of assembling 0201 components on this equipment.

Feeders for these components are required to present the parts in a predictable and repeatable location, and must do so many times per second. Good maintenance and handling of the feeders is required to ensure their long term reliability. Dedicated feeders for each individual part and one set for each machine reduce the handling the feeders undergo, especially if the feeders are stored loaded with their parts when not in use. Spare feeders provide the opportunity to eliminate feeder problems with a known good feeder when troubleshooting process problems.

Feeder adjustment for the particular feeders utilized is performed by comparison to a reference feeder fixture at an offline repair station. A microscope camera is used and its image is displayed and centered on a PC monitor. The feeder under adjustment is then placed under the microscope and adjusted until aligned to the reference fixture.

Nozzles, by the nature of the parts they transport, are precision items. Proper maintenance is key to ensure proper functioning. Regular cleaning is recommended to prevent buildup inside the nozzle that could interfere with pick and cause the nozzle to drop the component before it reaches the placement location. Wear is also a significant consideration as uneven pick surfaces can cause problems with picks and placements.

A standard testing method employed for surface mount placement machines is performing placements on a PCB that has double-sided tape placed on it. It has been demonstrated in practice on this assembly that the tape can leave residue on the nozzle surface or inside the vacuum port that causes the nozzles to malfunction until removed and cleaned. If a test run is required, limiting the active nozzles (and preferably replacing those nozzles with spares or deactivating them before commencing assembly) is recommended to avoid this problem and the difficult troubleshooting that accompanies it.

Improper placement settings can be a cause of nozzle damage. Component thickness, PCB thickness, and placement level settings can cause the nozzle to be driven into the board if improperly set. This can cause the face of the nozzle to be chipped or cause parts to damage the inside diameter of the vacuum port, rendering the nozzle useless. Improper settings can also prevent the nozzles from properly seating to the surface of the PCB during placement and cause the parts to be dropped on the board. These components are very unlikely to remain in the location where they are dropped and may interfere with further assembly.

The microBGA incorporated on the scan engine assembly is large enough to be handled by the same nozzles as 0603 components and therefore doesn't require special nozzles. The parts are supplied in 8mm x 4mm pitch embossed tape and are used with standard feeders.

Proper vision recognition of 0201 components is essential to correct placement. The chipshooter manufacturer has published a manual[4] specifically addressing machine settings for these components. It is important for the process engineer to obtain recommendations from the equipment's manufacturer before attempting to handle and place miniature components.

The vision manual also includes information regarding the storage and maintenance of the placement nozzles in the form of drawings of suggested fixturing. Although the details will vary based on the specific form of nozzle employed, the basic concept is a block with drilled holes where the nozzles can be stored tip up. Regular cleaning in isopropyl alcohol is suggested using a plate drilled similar to the storage block except long screws act as legs to support the plate and nozzles while also allowing cleaning fluid to flow throughout the cleaning chamber.

Placement of miniature components requires the use of local fiducials when building a multi-circuit panel to correct for the tolerances present in the bare PCB. Original designs of the scan engine circuit assembly provided a round local fiducial with a diameter of 0.635mm (0.025") but this proved too small for the recognition system to accurately identify and resulted in a significant number of circuits placed with a uniform offset from optimal. The production release design of the circuit enlarged the fiducial to 1.016mm (0.040"). Although this is smaller than the standard size, proper and precise adjustment of the recognition system provides sufficient accuracy to eliminate the shifted placement defects. It is important to regularly adjust and optimize the recognition lighting and settings in order to maintain a repeatable recognition of the fiducials.

Placement accuracy of the equipment expected to assemble 0201 components is an important consideration. The determination of whether the equipment is capable of accurately placing 0201s must be investigated in relation to the PCB pad layout and the standard workmanship requirements. For Classes 1 and 2 the requirement is no greater than 50% side overhang; Class 3 assemblies are required to maintain no greater than 25% side overhang. End overhang is not permitted in any class.[5] Using the maximum length and width of 0201s (0.063mm and 0.033mm respectively) and the rated placement accuracy of the equipment employed (+/-0.102mm in this case), it can be determined what the maximum distance from true center a component edge can be expected to be placed by an in control process:

$$MaxOffset_x = \frac{MaxLength_{0201}}{2} + |PlacementAccuracy|$$
$$MaxOffset_y = \frac{MaxWidth_{0201}}{2} + |PlacementAccuracy|$$

In this case, $MaxOffset_x$ = (0.063/2 + 0.102) = 0.134mm and $MaxOffset_y$ = (0.033/2 + 0.102) = 0.119mm.

With this information, the minimum pad size can be determined for a particular part/machine combination:

<u>Class 1 & 2 Side Overhang/Pad Width</u>

$$PadWidth_{\min} = 2 \times MaxOffset_y$$

<u>Class 3 Side Overhang/Pad Width</u>

$$PadWidth_{\min} = \frac{MaxWidth_{0201}}{2} + 2 \times MaxOffset_y$$

<u>All Classes End Overhang/Pad End to End Distance</u>

$$PadEndDist_{\min} = MaxLength_{0201} + 2 \times MaxOffset_x$$

In the subject case (a Class 2 Assembly), the minimum pad width is 0.238mm (0.00937") and the minimum pad end to end distance is 0.331mm (0.0130"). This initial analysis of pad design and placement capability is important to perform before attempting to assemble with 0201s to determine if the equipment is in fact capable of producing quality assemblies at the manufacturer's rated capability.

Placement Defect Modes

The defect modes for 0201 component placement fall into two main categories in the experience of the author: missing or scattered. Missing components can be caused by deficient solder paste, where insufficient adhesive forces are present to hold the part in place. A missing part can also be caused by inaccurate placement. A component dropping from a nozzle prior to ultimate placement can also cause missing parts. This can lead to nozzle contamination with solder paste causing more dropped components or poor pick rates. Missing part defects are characterized by the inability to locate the missing component anywhere on the assembly.

Contrasted with missing components are defects caused by scattered components. These defects all imply a missing component at some location, and that component is located at some unintended position. The components can end up soldered to an unused pad or to other components. They can also interfere with further assembly by being located in a position that is intended for another component.

One characteristic of assembly with miniature components is the tendency for defects to "cluster". Very often, one defect can be causative of another. This is the case when a part breaks free from the solder paste holding it in place, comes to rest on another pad set, causing another part to scatter to an unintended location. A single improper placement can cause a chain reaction that significantly decreases yields for that PCB.

The most difficult aspect of miniature component assembly defect mode analysis is that all causes present nearly the same symptoms, especially in cases of scattered components. Only through systematic troubleshooting and effect measurements can root causes be determined, corrected, and verified when presented with a case of low yields. The ability to perform this analysis is only possible with careful and deliberate implementation of a miniature component placement process as experience is the best method of building a troubleshooting base knowledge.

PCB Support During Processing

Another extremely important factor in the printing process is PCB flatness. Single-sided assemblies lend themselves to easy support using some variation of a flat table below the PCB. However, double-sided assemblies require a different solution usually in the form of pins that are either manually placed or a system that locks a series of pins in place to conform to the assembly being processed. A major drawback to pin systems is that they cannot be placed with the required density to support an assembly such as the scan engine PCB with a uniform flatness.

In order to counter this problem, a carrier was designed to transport and support the assembly during processing. The carrier is manufactured out of a static dissipative material that is tolerant to and dimensionally stable at temperatures up to 350°C. Pockets are incorporated as reliefs against the components present during second pass assembly and to allow the carrier to be used for both sides during Automated Optical Inspection. The PCB is located using pins that mate to holes in the framing of the PCB pallet. These pins are installed to leaf springs that allow the pins to drop away during stencil printing and allow the stencil to sit flush to the PCB.

The advantages of this carrier are significant. No longer are set fixture pins expected to ensure complete flatness of PCBs under assembly, both during paste printing and placement. The PCBs don't bow when traveling on conveyors between machines. In process inspection, which by necessity must be performed under a microscope, can be performed with a much lower risk of disturbing the components held in place solely with the tacky properties of very small deposits of solder paste.

Solder Reflow Considerations

Solder reflow of miniature components is the process most similar in behavior to standard SMT assembly. Assuming sufficient solder paste is present and the component was placed accurately, a miniature component is treated just like any other low mass part for profiling purposes. A Ramp-to-Spike, or Tent, Profile should provide defect free reflow and has been the only profile utilized on the assemblies discussed. The solder paste manufacturer should be consulted to determine if there are recommended reflow parameters for the particular paste utilized. Assemblies should be outfitted with thermocouples and measured as it passes through the reflow process to verify proper temperature exposure, especially where microBGAs are present. It is vitally important to process microBGAs at the proper temperatures as their solder joints are not visually inspectable. Lead free assembly of miniature components can be treated just like any other lead free assembly with regard to temperature parameters. Again, it is important to measure the parts during reflow and ensure that solder paste manufacturer recommendations are followed.

Conclusion

Assembly using miniature components is a process that requires attention to every detail. The process itself is very unforgiving and a small number of original placement defects can cause a large number of defects through a clustering effect. Design of the PCB and the stencil requires detailed attention. Placement equipment must be analyzed for capability to assemble with miniature components, have all proper optimized settings for miniature component assembly, and be properly maintained at all times. Root cause analysis of defects during placement is difficult as most defect modes present the same symptoms; the ability to troubleshoot process issues is gained through experience developing the process. PCB support using a dedicated carrier provides significant advantages, especially with double sided assemblies.

References

1. Universal Instruments Corporation. 2000. Process Parameters Optimization For Mass Reflow Of 0201 Components.
2. National Semiconductors Corporation. 2004. Application Note 1112: Micro SMD Wafer Level Chip Scale Package.
3. Shea, Chris, Bruce Moloznik, David Hemmelgarn. 2003. Printing and Profiling Fine Feature Devices. *Proceedings of APEX 2003*.
4. Universal Instruments Corporation. 2001. UIC Manual ID No. 47890601: Small Component Front Lighting Recognition.
5. IPC. 2005. IPC-A-610D: Acceptability of Electronic Assemblies. Bannockburn, Illinois: IPC.