ABSTRACT
0.3mm pitch is the next level of miniaturized area array packages for mobile and wireless devices. The components may be attached to the PCB by a variety of methods that include stencil printing solder paste or dipping the balls in either solder paste or tacky flux, and then reflowing. Following PCB attachment, they must be inspected, and depending on the application underfilled, tested and possibly reworked as part of the PCB assembly process.

This paper reviews the assembly process development project for 0.3mm devices. It addresses the test vehicle, process options, material selection, inspection and analysis methods, and discusses considerations of the individual aspects of the assembly process.

Key words: 0.3mm, CSP, microCSP, WLP, microBGA

INTRODUCTION
As part of the continuing trend of miniaturization, 0.3mm pitch is currently the next level of component size and spacing reduction. As with all other “new” components in the SMT miniaturization trend, designers are ready to enjoy the benefits of faster, cleaner signals and sleeker, thinner devices, but before the new packages can be designed into the next generation of portable electronics, their circuit assembly processes must be developed and proven robust.

TEST VEHICLES
The series of studies used several different 0.3mm CSP packages, with I/O counts of 196, 400 and 676 in various stages of the process characterization and development process. The final confirmation builds were all performed with 400 I/O packages as seen in Figure 1 and are the primary focus of this report. All package sizes shared similar ball array characteristics as described in Table 1:

<table>
<thead>
<tr>
<th>Item</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pitch</td>
<td>0.3mm</td>
</tr>
<tr>
<td>Solder ball diameter</td>
<td>0.2 ± 0.03mm</td>
</tr>
<tr>
<td>Solder ball height</td>
<td>0.17 ± 0.03mm</td>
</tr>
<tr>
<td>Total height of the component</td>
<td>0.57 ± 0.03mm</td>
</tr>
<tr>
<td>Solder ball alloy</td>
<td>SAC305</td>
</tr>
<tr>
<td>Packaging</td>
<td>Tape &amp; Reel</td>
</tr>
</tbody>
</table>

The test vehicle shown in Figure 2 was designed to accommodate the three different I/O devices. The 4-layer PCB was 1.0mm (40mil) thick, constructed of high-temperature FR-4 and used a high-temperature organic solderability preservative (OSP) pad finish. Non-solder mask defined (NSMD) and solder mask defined (SMD) pads of various sizes were designed into the test vehicle for
experimentation on optimizing pad geometry and configuration.

![Image](image_url)

**Figure 2.** 0.3mm CSP Test Vehicle

Prior to assembly, all test components and materials were thoroughly inspected, including:

- PCB pad size and definition solder mask thickness and alignment, cleanliness.
- Component solder ball diameter, height and pitch
- Stencil aperture size, shape and wall geometry.

**ASSEMBLY AND TEST PROCESSES**

Typical SMT assembly processes follow the sequence stencil print-place-reflow, with optional automatic inspection steps after each major process. The miniature interconnect feature size of CSP packages in high density PCBs, when combined on PCBs with large feature components like QFN thermal pads or RF shields, often challenge the capabilities of broad band stencil printing. As an alternative to printing, they are sometimes dipped in flux or solder paste prior to placement. In the assembly process flow chart shown in Figure 3, the purple blocks illustrate the assembly sequence using a dip process, the green blocks illustrate the print-only process, and the blue blocks represent the remainder of the assembly process common to both methods.

![Assembly and Test Flowchart](image_url)

**Figure 3.** Assembly and Test Flowchart
Dipping vs Printing
The method by which soldering materials are applied depends on a number of factors, and each process has distinct advantages and disadvantages.

Dipping is an established process that is used extensively in flip chip assembly and is now commonly deployed for package on package (POP) processes. All major pick and place manufacturers offer dip unit options on their fine pitch equipment. Using a dip process requires dedicated hardware, specially developed materials, and the generation of additional process control and equipment maintenance activities. It also adds cycle time to the placement process, which may impact line balancing and production throughput. However, dipping enables rapid deployment of ultra-fine pitch technology on PCB assemblies that would otherwise require the delicate balance of broadband stencil printing, and is often the more conservative choice.

Printing does not require any additional hardware to purchase or maintain, it uses less expensive materials, and it does not impact cycle times or production rates. It does, however, require Type 4 solder paste, which is slightly more expensive than Type 3. The area ratios of stencil apertures for 0.5mm devices is in the 0.50 range, requiring newer, more expensive stencil technologies and higher levels of process monitoring and control.

Dip Materials
In earlier stages of process development testing, multiple dip options were investigated, and materials included tacky flux, epoxy flux, and types 5 and 6 solder pastes. Material tackiness is a considerable factor in the selection of the dipping medium. High tack forces can overcome the vacuum of the nozzle and retain the CSP device in the dip tray. Compatibility with print solder paste reflow profiles should be considered when selecting any flux or paste dip material. Additionally, proper handling practices should be employed with dip materials: they should be stored at their specified temperatures, syringe tip down, allowed to thaw at room temperature for at least four hours, and remain unopened until ready for use.

PROCESSING CONSIDERATIONS
Stencil Printing
To produce consistent, acceptable solder volumes, the paste printing process requires diligent control. A stencil printer with a minimum alignment repeatability of 0.025mm (1mil) @1.67 Cpk should be used. Stencil under wipes will likely be more frequent than SMT processes with coarser pitches, and the use of compatible chemical solvents may improve process performance.

A 100µm (4mil) laser cut foil with release-enhancing coating inside the apertures is suggested in conjunction with Type 4 solder paste. Square apertures are recommended. If the PCB pads are 0.200mm (8mil), apertures should also be 0.200mm (8mil). If the PCB pads are 0.175mm (7mil), apertures should be enlarged to 110%. These aperture sizes represent a 0.50 area ratio, which is very near the edge of the process window, so any reduction of aperture size is strongly discouraged.

Print parameters will vary with solder paste formulation; print optimization DOEs are suggested. As with any SMT printing process, support tooling is a critical element; therefore, a dedicated vacuum tooling plate is preferred.

Dip and Placement
The dip and place process approved for production was developed on a standard placement platform. The machine was equipped with a fine pitch placement head, 7.7um/pixel high resolution vision system, and a rotary dip flux unit. The process sequence was pick-dip-vision-place. Alternatively, adding another vision system analysis prior to dipping to check for ball presence is possible, but the extra vision review further extends the cycle time.

Dip depth is a critical factor in the flux/paste dip process. A wet film gage with 10um or better resolution should be used to verify the material depth in the dip tray, as shown in Figure 4. For the 0.17mm ball height in this test, the dip depth was controlled at 0.9 -1.0mm. For different ball heights, the general guideline is 40-60% of actual ball height. Flux/paste depth should be checked with the wet film gage periodically. Frequency of the checks will depend on production rates.

![Figure 4. Flux dip depth verification](image)

Reflow
The combination of highly miniaturized and larger components on a thin PCB may require a 10-zone oven to manage thermal differentials across the board. Thermal profiling of the exact assembly configuration is required for this new technology; a 0.4mm drill bit will be required to successfully embed thermocouples in the balls of the CSP package on a profile board. Thermocouples should be located in a center ball, on a corner ball, and on the body of the package. Individual solder pastes dictate unique profiles for optimal soldering performance; generally speaking, the profile parameters should include a 60-90 second soak, a
TAL of ~60sec, a peak temp of at least 235°C, and a nitrogen-inerted reflow atmosphere.

Rework
Rework considerations are similar to those for any small area array package: a nitrogen-equipped, hot gas rework machine is required. A standard or custom nozzle configured for the specific package dimensions should be used. Custom profiles should be developed for removal and replacement of the device. In profile development, thermocouple attachment methods and locations should be the same as those used for the mass reflow profile development: one on a center ball, one on a corner ball, and one on the package body. Additionally, one more thermocouple should be mounted on a component adjacent to the reworked package to monitor the thermal effects of the rework process on nearby devices.

Standard removal practices should be employed, and site redressing should only be performed with semi-automated, non-contact scavenging units; manual site redressing will likely damage pads and/or solder mask.

Reprinting paste on the pads is not a viable practice because the feature sizes are too small for manual printing to produce repeatable paste deposits. Devices must be dipped in solder paste or flux material. Using a standard pickup tube and nozzle, a dip plate block should be used to control the dip depth to 40-60% of the ball height prior to placement and reflow.

All desoldering, redressing and resoldering operations should be performed under nitrogen.

Underfill
The underfill process was developed on an in-line, high accuracy jetting dispenser. Use of an active feedback control system to maintain consistent dispense volumes is recommended. Best results were achieved with an L-pass pattern with 2 passes and at least 5 seconds between passes, as shown in Figure 5. Flow times were 30-35 seconds.

A substrate heater is required to assist with capillary flow of the underfill; a nozzle heater is recommended. Heater settings depend on the specific underfill material; multiple underfill materials were approved for use as a result of the DOE. Each materials' technical data sheet should be referenced for the optimum temperatures.

PROCESS CONTROL AND VERIFICATION
Stencil Printing
Print deposit volume and positional offsets data should be collected for every deposit on every PCB, as seen in figures 6 and 7. The accuracy of the solder paste inspection (SPI) machines should be no less than 10% on the smaller 8mil deposits.

Figure 6. Paste deposit volume boxplot.

Figure 7. Paste deposit positional offset chart

As with any fine feature printing process, best practices should be employed, particularly with respect to the mechanical setup of the stencil printer: a dedicated vacuum
board support block should be used, squeegees should be leveled and their pressure calibrated, squeegee pressure should be sufficient to clear all the paste from their path across the stencil, the stencil should make direct contact with the PCB, and solder paste material should be at proper working temperature and viscosity.

**Dip and Placement**
Dip depth should be monitored periodically using a wet film gauge (Figure 4). The dip reservoir should be cleaned once per shift using only lint-free wipes, as even small fibers can ruin the assembly process. Dip depth should be verified on a device using an endoscope with measurement capability and by visually checking the footprint, as shown in Figures 8 through 10.

![Figure 8. Visual paste dip verification with endoscope](image)

**Figure 8. Visual paste dip verification with endoscope**

![Figure 9. Unacceptable footprint post-dip](image) ![Figure 10. Acceptable footprint post-dip](image)

**Figure 9. Unacceptable footprint post-dip**

**Figure 10. Acceptable footprint post-dip**

100% ball inspection in the placer’s vision system is recommended. The typical production process performs vision inspection after ball dip. Additional inspection prior to ball dip is optional, and suggested for new package types or suppliers.

![Figure 11. Vision system 100% ball presence verification](image)

**Figure 11. Vision system 100% ball presence verification**

During process development DOEs, placement accuracy was verified via endoscope and X-ray before and after reflow. Pre-reflow inspection is not required in production processes, but post-reflow inspection is.

**Reflow**
Reflow ovens are usually self-monitoring with respect to belt speed, zone temperature and nitrogen environment. Regular calibration and periodic checks should ensure in-control performance. Proper thermal profiling will ensure the PCB experiences the appropriate thermal cycle.

Soldering results of the reflow process should be checked by Automatic X-ray Inspection (AXI) as shown in Figure 12. 100% of the devices should be inspected. If AXI flags a defect, the defect should be verified with oblique angle X-ray (Figure 13) prior to performing rework. Visual inspection using endoscopes can also be performed to verify defects if they are on the outer edge of the array (Figure 14).

![Figure 12. AXI X-ray image (no defects visible)](image)
Rework
Do not attempt to rework 0.3mm CSPs on a generic library profile. Individual, assembly-specific profiles should be developed for each device location on each assembly number.

Boards should be baked to remove absorbed moisture prior to rework. Excessive topside force should be avoided during component removal to minimize squeeze-out solder balling.

A semi-automated, non-contact pad redressing unit is recommended. Flux residue must be fully cleaned from the pads after redressing. Visual inspection is also required to verify the success of redress process. The average thickness of the solder residue on the redressed pads shown in Figure 15 should be less than 20µm.

Underfill
For a robust underfill process, an active feedback control system that automatically adjusts dispense volumes is recommended. A non-contact laser height sensor and package corner alignment algorithms provide the most precise dispensing locations. Additionally, built-in nozzle cleaning is recommended to be performed at regular intervals based on production rates.

During process development, the underfill process was verified by flat-section analysis and C-mode Scanning Acoustic Microscopy (CSAM). A flat section image is shown in Figure 18.
Solder Joint Verification

During the process development, the solder joint quality was verified by pry test and cross section. A typical pry test result is shown in Figure 20 and cross section images are shown in Figure 21.

Production verification of underfill processes should be non-destructive in nature, i.e. CSAM. A typical CSAM underfill image is shown in Figure 19.

**Figure 18.** Flat section of underfilled area

**Figure 19.** CSAM image of component underfill

**Figure 20.** Solder joint pry test results (PCB side)

**Figure 21.** Cross sectional images of 0.3mm CSP solder joints

**ADDITIONAL CONSIDERATIONS**

**Keepout zones** are preferred to be a minimum of 4mm around the perimeter of the component to accommodate the pad redressing nozzle during rework operations.

**Device handling** should be performed with plastic tip tweezers or vacuum pens to avoid damage to fragile package substrate or balls.

**Lint-free wipes** should be used for all solder paste printing and dip fluxer maintenance to avoid fiber contamination.

**Ball height** varies with different package types. **Dip depth** should be determined as a percentage of actual solder ball height. Endoscope inspection may not be possible for very low ball heights post reflow.

**Flux residue** may affect underfill flow rates. Residue should factor into soldering material selection.
Requalification is required if any changes are made to the approved materials, tooling, or other elements of the assembly and inspection process.

CONCLUSIONS

0.3mm CSPs, the next generation of miniaturized, high density electronic packages can be assembled using modern SMT equipment.

Through multiple DOEs, two viable process flows have been developed and proven. Both assembly processes are very similar; the only differences are the soldering material for the devices. One process stencil prints solder paste over the entire PCB, including the 0.3mm pads; the other prints paste for every device except the 0.3mm CSP. In the case of the second process, the CSP’s balls are dipped in solder paste or tacky flux just prior to placement. Both process flows have advantages and drawbacks that must be considered when determining the best process for the product.

Because the 0.3mm CSP is the most miniaturized SMT component to date, best practices should be employed throughout the entire assembly process. These include proper setup and tooling in stencil printing, proper setup and monitoring of flux dip equipment, high-resolution vision alignment and placement capabilities, dedicated reflow profiles, best-in-class rework practices, tightly controlled underfill dispensing, and extra attention to material handling.

This paper presents summarized findings of long-term, multi-site component qualification and production process development activities. For more detailed information on any part of the DOEs or their findings, please contact jonathan_wu@jabil.com.

ACKNOWLEDGEMENT

The authors would like to thank Jabil managers Zambri Samsudin and Paul Neathway; contributors Hien Ly, Nabel Ghalib, and Arnold Reta; all participating supplier-partners, and former team members Howard Chen, Idris Mansor and David Sbiroli for their support on this multi-site component qualification and process development project.