The role fine pitch surface mount (SM) devices play in subassemblies is considerably more critical now than ever before. At 20 mils (0.5 mm) or less, fine pitch demands a well thought-out and orchestrated balance involving PCB design, fabrication, assembly, placement, and careful rework on SMD pads.

Otherwise, disjointed operations quickly usher in major challenges and issues. At the start, for instance, oversights at design layout can create a domino effect. That’s especially damaging in terms of excessive time and increased cost to OEMs who opt to rely on different design, fabrication, and assembly vendors with little to no coordination for this important aspect of fine pitch design.

At Design
There are a number of considerations for creating a pad stack for a fine pitch design. As shown in Figure 1, a pad stack is a combination of pads associated with the different layers of a printed circuit board, and these pads are used for routing the signals on the PCB. Design considerations are many, but some of the most prevalent include if the pads are defined by solder mask or non-solder mask, the silk screen, and fiducial marks. When implemented properly, these characteristics significantly contribute to the effective use of PCB real estate and subsequent design efficiency.

The question comes up as to why fine pitch assembly demands closer attention with PCB design layout taking on greater importance. The answer is PCB real estate is dramatically shrinking in today’s design, especially for small handheld devices. Fine pitch devices are used on boards that are smaller, more compact. A great number of components are placed on the top and bottom side of the board, hence a very crowded PCB. At the same time, these compact board designs call for considerably more power and, subsequently, greater amounts of heat are generated. Also, electromagnetic compatibility considerations must be taken into account, and PCB designers must be watchful for signal line losses.

Rework is another major consideration during the design phase. Pad stack mistakes must be avoided; otherwise, rework becomes highly challenging simply because fine pitch pads are extremely small. Applying heat to depopulate and install fine pitch devices can have a devastating effect on the small pads. Because they are so small, there’s the possibility of the extreme heat causing a pad to peel off from the surface of the PCB. Also, if heat is applied multiple times, there’s a possibility of solder mask peeling off from the surface of the PCB itself.

It’s also important to clearly define fiducial marks that pick and place machines rely on for precise component placement. Again,
the reason is fine pitch devices must be placed extremely precisely because the area of a pad is extremely small. If those markings aren’t properly defined, component placement is offset, requiring solder paste to be reapplied and placement to be performed again.

Therefore, it is crucial that the silkscreen and fiducials marks precisely comply with pad stack requirements. Plus, maintaining sufficient distance from pad-to-pad must be assured because if a pad size is too large, it increases the possibility of shorting other SMD pads of the device, especially if the board is under-etched.

Tolerances must also be tightly controlled when using a pad wizard or any other kind of software used to create the pad stacks within the CAD tools. Tolerances should be closely and frequently inspected when using a pad wizard so that everything is precisely defined. By doing so, pads are not only properly created but routing can also be correctly performed.

When it comes to fine pitch micro-BGAs, it becomes even more crucial to properly define those pads. The reason is to have a proper fan-out for a dog bone structure or any one of a myriad of BGA fan-out strategies. This and other considerations targeted at the ultimate in precision are based on experienced design layout. That precision, or lack of it, ultimately has a direct bearing on fine pitch device performance at fabrication and assembly.

At Fabrication

Fine pitch devices require extremely tight tolerance control for the correct level of etching. Hence, selecting the right and experienced fab shop is highly important. If a board is over-etched and the SMD pad size is at 20 to 30 percent less than the required pad specification, then sufficient mass is not left on the board for a stable solder joint to be performed at the assembly stage. Otherwise, defects such as BGA voids are created (Figure 2).

Consequently, proper placement of a fine pitch SMD is not possible because the mass of the pad is out of proportion with the component lead size. This creates a mismatch between the pad on the PCB’s surface with the leads of an SMD component. A weak solder joint is thus created, and, over time, that solder joint can be damaged and cause a failure, especially if the application under-}

Figure 3 Bridging Results From Too Much Solder Applied During the Printing Process
The Big Deal Over Fine Pitch Assembly

Figure 4 Stencil Printing Performed Using a Printer Designed for Fine Pitch Devices

populated with fine pitch devices. But hot air solder level (HASL) should be avoided because it is not as precisely controlled a process on the pad compared to gold or silver. Because it is not, the uniformity of deposition is not present, thereby making fine pitch assembly less than optimal.

While immersion gold or silver is ideally suited for fine pitch, OSP should not be overlooked because it also produces an extremely flat surface. However, regardless of the finish—gold, silver, or OSP—there are various tradeoffs associated with each. As shown in Table 1, those are shelf life, cost, reflow cycles, and solder joint flatness. Immersion silver and gold are expensive metal alloys, thereby increasing the cost of producing the bare boards.

At production levels, these finishes could cost 5 to 10 percent extra, depending on the amount of exposed surfaces. Therefore, it’s best from a cost/performance point of view to select a finish that can be cost justified in a particular end product.

As for shelf life, HASL has about 18 months, but OSP has only 6 months. Immersion silver has a shelf life ranging from 12 to 16 months, while immersion gold is the most durable at 24 months.

OSP cannot undergo more than two to three reflow cycles. If the need for rework continues, then the SMT pads on the OSP finish begins peeling off. Thus, OSP is not the best finish for rework. On the other hand, immersion silver or gold can undergo six to eight reflow cycles.

Using immersion gold or immersion silver results in a flatter PCB surface finish. Consequently, the flatter PCB surface finish is considerably more conducive to a perfect fine pitch PCB assembly compared to an HASL finish. Also, these finishes withstand higher temperatures and there is a much less likelihood of the pads being peeled off from the board surface when it is exposed multiple times to higher temperature cycles.

Assembly

Stencil design plays a vital role in fine pitch assembly, which means that its thickness and proper definition aperture rank high in importance for perfect assembly. It’s also important to define the correlation between the board itself and the aperture sizes for the SMD pads. Should the aperture be opened at a 1:1 ratio in respect to the size of SMD pad? Or should it be greater or less than a 1:1 ratio? This depends on the amount of solder to be deposited on every pad of the fine pitch SMD. It is not a good idea to apply too much solder paste because it will create the possibility of shorts or bridging (Figure 3) between different SMD pads. At the same time, too little solder paste should not be applied in the printing process because it will create the possibility of opens within solder joints or perhaps cold solders.

Properly defining the aperture requires several key considerations and carefully applied steps. The printing mechanism should also be carefully planned out, keeping in mind the squeegee pressure, paste type, as well as cycle time for dispensing. Stencil printers need to be calibrated to every beginning of the assembly shift unless it is autocalibrated. Residues of previous solder paste stuck to a stencil’s top and bottom sides must be cleaned as often as every two hours when printing is being performed. Squeegee blades must be cleaned every two to three cycles in the case of regular assembly, but for fine pitch devices and BGAs, it should be cleaned after every cycle.

Also, the amount of stencil usage is often overlooked. The same stencil, when used a few hundred times on fine pitch SM devices, should be a candidate for replacement. Also, stencil printing must be performed by a printer designed for fine pitch devices (Figure 4). With fine pitch devices being so close to each other and squeegee pressure being applied so often, the distance between pads may begin to widen. As a result, more solder paste than necessary is applied where it’s not required. Sometimes there are cracks that start appearing between fine pitch SMD pads, thereby needing replacement.

Also, the solder mask must be precisely applied on the board populated with fine pitch devices. The entire pad of a fine pitch SMD must be exposed so that a complete pad surface is available for assembly. The pad should not be covered by any part of the solder mask or by silk screen.

If the silkscreen is not properly placed and an SMD pad is covered, even partially, then that portion of the pad will not properly solder to the fine pitch device. Nonstandard or weak solder joints are the result. To avoid this problematic area, it’s important to clip protruding portions as part of the silkscreen printing process.

Laser solder jet printing, a brand new technology, is providing a more effective method of dispensing the solder paste directly on the SMD pads compared to traditionally using the stencil. It not only improves the process control, which is quite crucial for fine pitch SMD assembly, but also can be used very effectively when a higher density of components is involved. Laser printing deposits solder paste using laserjet technology, which can fine tune the paste deposition in volume, area coverage, or pad shape on the fly practically without touching the PCB.

If one side of the board is mostly popu-
lated with fine pitch devices, it dispenses small amounts of solder paste. The other side of the same board may contain an analog section, for example, with lightly populated components and the laser printer can be changed and controlled to handle this contrast in population density. In this instance, with the analog portion and its larger devices, considerably higher paste dispensing is required, and the laser printer can be changed to operate accordingly.

Solder paste and its chemistry play a large role in fine pitch assembly. If solder paste is more granular, meaning it has a more thick substance within the granules of the paste mixture, it tends to overshoot when printing is performed. This means nongranular or slightly less granular paste should be used for fine pitch SMD pads. This way, pads are properly covered to produce a perfect or near perfect solder joint.

An effectively developed thermal profile is another factor in the proper mix for accurate fine pitch assembly. The correct profile coupled with placing the thermal profiles at the right sections of the board is essential. There is no room for error here. If fine pitch devices are not properly placed due to a poorly created thermal profile, or they are skewed for whatever reason, then costly rework will ultimately result.

**Conclusion**
Correctly and accurately performing fine pitch device placement requires leading-edge technology. EMS providers using older placement technologies confront an array of issues. Therefore, it is critical to have the capability to effectively deal with extremely small devices; for example, highly accurate 16 mils (0.4 mm) placement for micro-BGAs.

When certain fine pitch SM devices, like micro-BGAs, are used, chances are other fine pitch components like 402s and 201s are used on a board as well. Hence, an EMS provider is technologically stretched to the limit as well as challenged by this shrinking PCB real estate size. Consequently, to maintain the highest caliber of accuracy for fine pitch assembly, EMS providers must continue to be ahead of the technology curve.

Zulki Khan is founder and president, NexLogic Technologies, Inc., San Jose, Calif.
Email: zk@nexlogic.com

---

One important aspect of successful fine pitch assembly is for the finish to provide an extremely flat surface.