CIRCUITS ASSEMBLY

MARCH 2000

www.circuitsassembly.com

Process Control and Optimization

Inspection Strategies

Supply Chain Management

MEMS/PCB Assembly

APEX 2000 SHOW ISSUE

Inspection Strategies for Process Control

Bob Ries

Selecting the right tool for the inspection job at hand can be a daunting task. Understanding the different capabilities of each technology may be helpful.

The competitive drive for high yields on today's surface-mount lines demands some form of in-line, automated inspection method. The question remains: Which one? More than 35 companies worldwide offer automated inspection using laser, vision, x-ray or other technology. Varying specifications make comparisons difficult, and head-to-head evaluations are often long, drawn out and time-consuming, sometimes raising more questions than they answer.

Automated inspection tasks can range from a spot check of critical board areas to a cursory check for component presence and orientation to a comprehensive examination of the whole board. Line lasers measure height in one location, while scanning lasers can profile an entire board. Vision systems look at x and y but do not measure height. X-ray systems can find problems with solder joints and other features that remain hidden with laser or vision systems requiring a direct line-of-site. Determining the best tool for the specific task at hand makes the job easier and faster.

Where in the Process?

Before evaluating individual inspection methods, a manufacturer must assess when to inspect the board. Typically, this assessment is done by collecting data about defects types (Figure 1) over several weeks and determining where the biggest problems are occurring on the production line. Generally speaking, when evaluating this data, remember that, the earlier you find a defect in the process, the less expensive fixing that defect will be. Repair and rework costs vary, but, usually, a five- to tenfold increase in cost occurs after each successive production step.

Inspecting the board immediately after printing checks the paste printing operation itself. Correcting problems at this stage requires merely cleaning and reprinting the board, which is considerably less expensive than repairing the board downstream. Because the majority of surface-mount solder defects result from poor paste printing, inspecting at this stage offers considerable benefits.

Inspecting boards after component placement monitors the operation and accuracy of pick-and-place machines. Even in a controlled process, you may want to inspect every board at this point to locate random errors that are almost impossible to eliminate completely. Post-placement inspection can identify a wrong component caused by an improperly loaded reel or can locate a misplaced part that fell off the vacuum nozzle during placement.

After reflow, boards are ready for in-circuit test (ICT) and functional test before being installed in the final product. Post-reflow inspection yields information on solder joint quality, excess and insufficient solder, and component migration. Note that inspecting for placement accuracy after reflow is ineffective because components sometimes move while in the oven. Post-reflow inspection is gaining in popularity because, as circuit complexity increases, electrical test coverage decreases. Post-reflow inspection is sometimes used to augment ICT.

Today, manufacturers generally do not include inspection equipment at all three stages, both because of the costs involved and the impact on production throughput. Therefore, when evaluating these techniques, each manufacturer must determine where inspection will provide the best results in yield improvement and labor reduction. Figure 2 summarizes the PCB assembly defects that can be detected at various stages during the production cycle.

In-Printer Inspection

Generally, in-printer inspection is the least expensive method of checking solder paste printing quality. Most modern printers include vision capability to locate fiducials on the board. The same camera can look down to examine the solder paste after printing and look up at the board stencil to identify clogged apertures. Cameras provide a two-dimensional (2-D) view of the solder pad coverage and paste registration on the board, which is useful because adjustments and corrections may be made if the printer drifts out of alignment while in use.

The primary drawback of in-printer inspection is slowed speed because the printer cannot

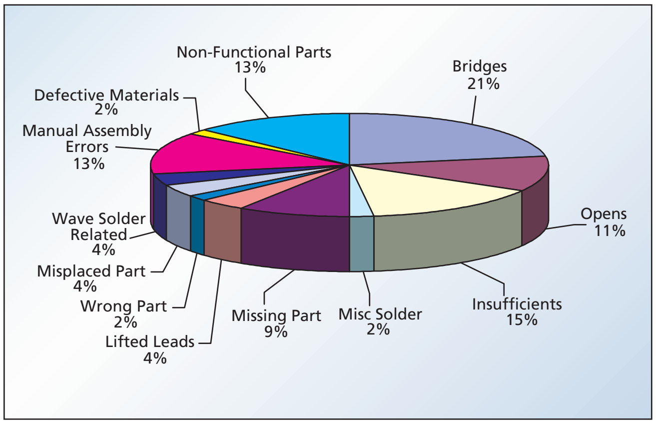

FIGURE 1: A survey of 28 different manufacturers with a total of 285 lines revealed these assembly defects.

inspect one board while printing another. The in-printer method requires printing the board, stopping the line, and inspecting the result, which adds to cycle time and usually makes this method too slow for conventional production.

Additionally, in-printer inspection performs x-y measurements only. Typically, it does not measure volume and, therefore, cannot reliably predict solder-joint quality in the final product. Even cameras that boast some three-dimensional (3-D) capability generally only estimate height based on a single mid-pad measurement. This approach will miss flaws such as dog ears and stencil blockages at the edges of the pad, reducing accuracy and limiting repeatability. For this reason, in-printer inspection is most often used during printer setup. The manufacturer prints the first few boards, then uses the camera to look at the paste and the stencil. Once the process has been verified and calibrated, the manufacturer turns off the camera to avoid slowing the production line.

3-D Solder Paste Inspection

The next step up for post-print inspection is a 3-D in-line system available to perform tasks from sampling to 100-percent inspection. These machines employ various methods of optical triangulation to measure height and use the composite height data to calculate area and volume. Most systems are laser-based, but some entry-level sampling systems use structured white light to take a 3-D snapshot of many solder pads in a single step. The sensor calculates height above a reference plane and creates a 3-D topographical map of the solder deposit, producing a very repeatable calculation of solder volume.

Sampling systems can be programmed on-line relatively quickly. Usually, sampling starts with critical sites for fine-pitch components and ball grid arrays (BGAs). Alternatively, some users inspect the four corners and center of each board to increase the likelihood of identifying problems that appear most often near the edge of the board.

These machines are set up to notify the operator or stop the line if measurements start to exceed the preprogrammed tolerances, or they can be programmed to merely log the data and move on. Volumes higher or lower than the specified limits usually signal problems: A higher volume of paste often causes bridging while a lower volume means insufficient paste for a good connection.

The next level is a high-speed, 100-percent 3-D inspection machine. Traditionally, these systems have employed laser-scanning height sensors, but alternative systems have been developed that use a structured white light sensor. Both types of systems offer the same inspection capabilities as sampling systems, but with 10 to 40 times the speed, depending on what sensing mode and motion speed are selected. These high-speed systems are capable of inspecting most of the board without slowing production and can repeatably measure small chip-scale package (CSP) deposits.

Because these systems inspect thousands of solder paste-covered pads, they must be programmed off-line using computer-aided design (CAD) data. Some software now uses the Gerber data file from the stencil design because the stencil most closely represents the finished board and Gerber data files are available at most production shops.

Finally, a traditional automated optical inspection (AOI) machine with optional 3-D sensing capability theoretically can be used for solder paste, but rarely is, in practice. These systems are capable of very fast, basic 2-D (coverage or area) inspection for 100 percent of a PCB and slow, sampling 3-D inspection based on a moving laser line. Because multiple sensing techniques must be designed into these systems, they usually are the most expensive alternative. The applicability of this approach is limited, however, because 2-D AOI inspection does not capture paste volume and, therefore, is unreliable as a predictor of post-reflow solder joint quality.

Pre-Reflow Inspection

By examining the loaded board after placement and before reflow, a manufacturer can determine component presence or absence, verify identification, measure orientation, and look for any solder problems that were not located during post-paste inspection. While using laser- or white light-based systems is feasible for pre-reflow inspection, this approach is slow and the information produced is limited to height profiles of the components on the board. Most manufacturers prefer to use 2-D optical inspection for pre-reflow, with AOI as the most popular.

Analysis techniques in traditional pre-reflow AOI systems are based on the combination of dozens of image-processing algorithms, or recipes. These decision formulas must be set up for each component type and, many times, require selection and adjustment of 20 or 30 individual parameters. The most common algorithm is template matching, in which the AOI system compares a golden image of a component to the image of the board being inspected. Template matching makes sense in theory but, in reality, may be unforgiving of deviations from perceived good-board appearance and of engineering change orders (ECOs) and other board modifications. Often, acceptable false-call rates are only achieved at or near the end of a production run. When the same assembly is run again days or weeks later, the tuning process usually has to start from the beginning because something—a new type of part or a different material—has changed the board's appearance.

One pre-reflow AOI approach uses statistical appearance models (SAM), which are created from five or 10 examples. After examining the sample components, the system builds its own model, considering normal variations such as color, size and reflectivity. It learns as it goes, expanding the model to accommodate every acceptable variation it sees. Therefore, the more boards it inspects, the better the model becomes. As the model

FIGURE 2: Different defects are associated with each process step. Selecting an inspection technology often hinges on determining what defects are the most problematic.

PCB Assembly Defect Type	Post Print	Post Placement	Post Reflow	X-Ray
Insufficient Solder	X			
Blocked Aperture	X			
Excess Solder	X			
Bridge (Paste)	X			
Misregistration	X			
Missing Part		X	X	
Polarity		X	X	
Misplaced Part		X		
Wrong Part		X	X	
Tombstone/Billboard			X	
Dry Joint			X	X
Lifted Lead			X	X
Bridge (Reflowed Solder)			X	X
Solder Voiding				X
Hidden Lead Solder Defect				X

improves, the false-call rate diminishes, eventually becoming orders of magnitude better than algorithm-based systems.

A final important consideration in evaluating pre-reflow AOI is overall component-placement measurement performance. Because the overall goal of inspection at this point in the line is to provide process control and not just screen out defects, the inspection machine must produce trustworthy, quantifiable results.

Post-Reflow Inspection, X-Ray

After reflow, solder joint inspection is critical for identifying weak or defective connections. ICT usually identifies nonfunctioning boards, but boards with weak solder joints will likely pass ICT only to fail with use. Post-reflow inspection systems, whether AOI, infrared or x-ray, are the most expensive on the line.

AOI enjoys considerable popularity for post-reflow inspection, whether or not the manufacturer has also applied it at the prior step. However, inspecting solder joints is a challenge for any type of automated inspection system because of the wide variations in appearance of acceptable joints. Defining reliable appearance criteria for a good joint is difficult. An ugly-looking connection might be perfectly acceptable, while a smooth, shiny joint may be defective. At this point in the line, AOI can find such failures as solder shorts, component tombstoning and billboarding, lifted leads, off-pad components, and misplaced solder balls. Its primary drawback is that it cannot examine the joints on BGAs, J-lead integrated circuits (ICs), and other surface-mount parts whose connections reside underneath and out of the line of sight. Similarly, heat sinks and electromechanical interference (EMI) shielding can preclude an optical test.

X-ray inspection offers some advantages over line-of-sight techniques such as laser and AOI because it can analyze board features such as solder joints, despite intervening components and other barriers. Although x-ray can look for missing and out-of-place parts, it primarily assures solder-joint integrity by evaluating solder volume and placement, fillet height and other geometric characteristics. It can also detect coplanarity problems between large surface-mount components and board substrates. The primary drawbacks include speed and cost.

Remember that post-reflow inspection is a final screening for defects, not a method of process control. Although defects can and should be identified at this point, the source of the defects cannot be determined and, therefore, corrective action is not possible.

Making a Decision

The surface-mount production engineer considering inspection options has an ever-increasing number of tools from which to choose. Selecting the best tool for the job requires understanding the strengths and weaknesses of each—no all-purpose tool addresses every type of inspection. Before making a choice, the manufacturer must understand the situation on the line: prevalent defects; production speed; operator skill level; budget and payback; and any other factors affected by inspection. The optimum choice results when the inspection system is selected based on knowledge of the technology and a thorough understanding of the particular production environment at hand. ■

Bob Ries *is director of surface-mount systems marketing at CyberOptics Corp, Minneapolis, MN; (800) 746-6315; e-mail: bries@cyberoptics.com.*

CyberOptics Corporation
5900 Golden Hills Drive
Minneapolis, MN 55416
800-746-6315
info@cyberoptics.com
www.cyberoptics.com

8002380 Rev A 3/00