www.globalsmt.net

GLOBAL SMT & PACKAGING

The Global Assembly Journal for SMT and Advanced Packaging Professionals

Volume 11 Number 5 May 2011

ISSN 1474 - 0893

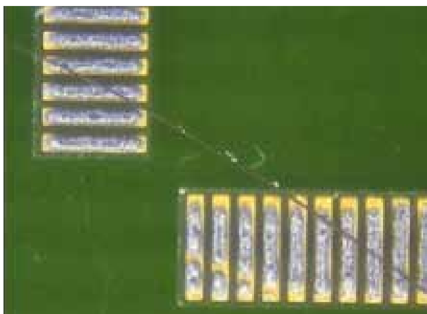
SOLDER PASTE STENCIL MANUFACTURING—IMPACT ON PRECISION & ACCURACY

X-RAY INSPECTION OF SEMICONDUCTOR DEVICES THAT USE COPPER WIRE INTERCONNECTIONS • BARE BOARD CONTAMINATION • ENHANCING SOLDER JOINT RELIABILITY OF CERAMIC COMPONENTS • BLIND VIA HOLE FAILURES • ARE FACTORIES EFFICIENT?

Gary Stoffer Interview Inside

Bare board contamination

Sheila Hamilton, Teknek, Renfrewshire, Scotland, UK



Technology and processing developments within SMT are moving bare board cleaning from a nice-to-have feature to an absolute necessity if yields are to be protected and/or improved.

This article examines the causes and impact of contamination and what solutions are available to tackle the problem.

Market trends

There are a number of factors that mean bare board cleaning is becoming a necessity including:

- Sourcing from low cost economies where cleanliness standards are less stringent
- The adoption of laser marking, which is a considerable source of contamination
- On-going miniaturisation
- Rising material, labor and fuel costs squeezing margins, leading to demand for higher yield
- Adoption of 3D solder-paste inspection

Board contamination

In general the contamination found on boards can be categorized as follows:

Environmental

Dust, clothing and hair fiber

People

People are a considerable source of contamination including skin, hair and clothing fiber

Packaging

Boards are typically packaged in shrink-wrap and often with paper separating sheets, both of which can contribute loose contamination to the board surface.

Manufacturing process

The PCB manufacturing process typically does not conclude in a clean room and will involve much handling during AOI, repair, punching, routing and packaging. These steps will leave residual contamination.

Laser marking

This increasingly used process generates significant amounts of carbonated debris that constitutes a risk to your print process and end of line yields.

Finally, when packaged boards are separated, they generate considerable amount of electro-static charge that will attract local contamination, be that from the environment or people.

Solder paste printing

It is generally held that 75% of end of line faults are related to the solder-paste print process. It therefore stands to reason that a key component in solder-paste print be properly prepared for this crucial process step.

3D solder paste inspection (SPI)

In an effort to improve quality and meet customer quality expectations there has been widespread adoption of 3D solder-paste inspection technology. 3D SPI can identify 100% of solder defects and will helpfully classify them as excessive, insufficient, offset, misshaped or bridging. There are many factors within the printing process that can influence these fault codes, including stencil design, printer set-up and paste. What about contamination?

What influence does contamination have on yields at SPI?

Established and sophisticated electronics assembly businesses manage the print process very closely and will typically have yields in the 91-94% range. The need to improve these yields is relentless as margins are undermined by competition and increasing input costs. How then do we drive yields further in a well-run, controlled assembly environment?

A significant part of the answer in this scenario is to understand the influence of contamination and seek to address it through cleaning of the bare board.

Symptoms of contamination include:

- Stencil apertures becoming blocked and causing misprints
- Craters after reflow caused by contamination having volatized
- Tomb-stoning (Manhattan Effect)
- Poor solder wetting
- Poor joint integrity
- Dry solder joints
- Excess or insufficient solder

Process Improvement

With over 10 years of experience in this application, Teknek can say with absolute confidence that removing contamination as a variable will add 4-6% to yield at SPI. Time and time, again this assertion is proven in the field with leading automotive, OEMs and EMS companies around the world.

The effect of cleaning the boards is usually measured using SPI equipment after the boards have been printed. Trials have shown a dramatic reduction in board failures with contact cleaning before the solder paste print process. In one study using a Koh Young S.P.I., Teknek found that:

- Overall failure rates fell from an average of 9.5% to 5%—an improvement of 47%.
- Excessive solder dropped from an average of 3.4 to 1.3%—a 62% improvement.
- Bridging falls from an average of 1% to 0.7%—an improvement of 30%.
- Insufficient solder falls from an average of 2.1% to 0.8%—an improvement of 61%

Boards that fail SPI

When a board fails SPI there are two possible outcomes.

Scrap

Some assemblers are required to scrap boards that fail SPI. This may be because of the safety critical nature of the end product or simply that they understand the full cost of rework. If the failure occurs during the first pass then the costs are easily assessed e.g. the board, paste, time, labor and disposal. On the second pass the same equation applies but will include components and more line time so the numbers are considerably higher.

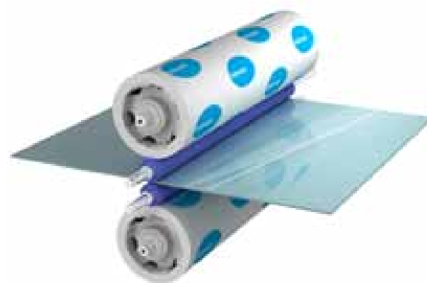
Rework

Each time a board is reworked, the quality and reliability is compromised. Misprinted boards can be washed down and wiped clean, but this will leave residual traces of solder paste, which are in turn a threat to inspection and end of line yields. (These will not necessarily be detected at the final electrical inspection at the end of the line.)

Contact cleaning technology

Now that we have established that PCBs arriving from the manufacturer are not pristine, what strategies can be put in place to reduce wastage, improve quality and increase yields?

A bare board is fed into a contact cleaning system at the start of an automated production line to remove surface contamination prior to the stencil printing process.



With contact cleaning, two tacky rubber rollers lift contaminants off both sides of the PCB at once. The rubber rolls then pass off the contaminants to the adhesive rolls.

Many manufacturers have tried brush, vacuum and blower systems to remove debris from the boards, but the most effective way to achieve clean PCBs is to use specialist contact cleaning equipment to clean the boards. This equipment uses a series of specially formulated elastomer rollers that make intimate contact with the bare board, gently removing dry unattached particles from the surface. (These rollers will remove contaminant particles down to one micron in size.) A reverse wound, pre-sheeted, adhesive roll then runs in contact with the elastomer rolls and transfers the particles to the adhesive. Once saturated a layer of the adhesive is removed, exposing fresh adhesive and the cycle starts again. Adhesive life is related to the contamination levels but our experience indicates a typical user will get between 400 and 600 panels for each sheet of adhesive. The adhesive sheets can also be used to investigate the root cause of the contamination—a useful diagnostic tool in its own right!

The cleaning device should also incorporate anti-static systems to eliminate any charge as the boards move through the production line. Static monitoring sensors are available to measure static levels within your pre-determined criteria.

Contact cleaning equipment will provide the highest levels of board cleanliness through the physical contact with the board. This contact overcomes the challenge presented to non-contact system by the "boundary layer."

Return on investment

Our experience indicates that in most cases the return of investment can be measured in months rather than years. This will be a function of the preceding yields and those after installation of a bare board cleaner. Also included in the equation should be the cost of rework or scrap for SPI failure and the associate material, labour and overhead losses. This of course can be broadly calculated during an initial consultation. The very best way to get certainty is to arrange a trial installation that will enable you to directly compare yields at SPI and end of line with the cleaner against the norm without.

Adoption

The benefits of bare board cleaning are accepted by most automotive electronics businesses, yet the broader EMS market has been slower to adopt. Often they will cite a lack of customer demand for this feature. Bare board cleaning is a significant opportunity for the assembly businesses in respect to margins and is a clear indication to prospective customers that you are in control of your process and capable of meeting their high quality requirements.

As the surface mount industry faces greater miniaturisation and demand for increased yield, contact cleaning will become increasingly accepted as a standard process in a line. It is important that the concept is fully understood and used as a tool in the quest for continuing yield control of a modern SMT line.

Sheila Hamilton is Technical Director of Teknek and is responsible for keeping the company at the forefront through product performance, capability and applications. Sheila joined Teknek in 1987 as technical director after working as a product designer (yachting equipment) and power station engineer. She has also run her own consultancy in the field of electronics component packaging. Sheila has a BSc in Mechanical Engineering from Glasgow University and a MBA from Strathclyde University. In addition, she is a recipient of two Smart Awards in the field of Electromagnetic Interference.