Improving Yield in SMT Production Environment

Asian electronics manufacturers face a number of challenges today. They are under increasing pressure to improve quality, increase yields, increase profitability and reduce wastage. An operational area which can help manufacturers achieve all these objectives is contamination elimination from the SMT production process. This article looks at the impact of contamination, its key causes and effective solutions which can be put in place.

Contamination within surface mount assembly causes three main faults: process, solder and reliability. Particulates can enter the manufacturing process at any stage and directly affect the yield. Eliminating contamination improves the solder and reliability of the assembled board, and provides efficient production processes, improves yield rates and directly impacts on profitability.

Eliminating process problems means efficient and effective production, providing faster turnaround and higher profit margins. In PCB manufacture where margins are being squeezed, optimising manufacturing process can mean the difference between growth or decline. PCB assemblers face the many problems of escalating costs, tighter tolerances, finer pitch, increased customer demands, which are forcing assemblers to re-assess their processes without raising overall manufacturing costs. Contamination on the bare PCB is directly responsible for lower production yields.

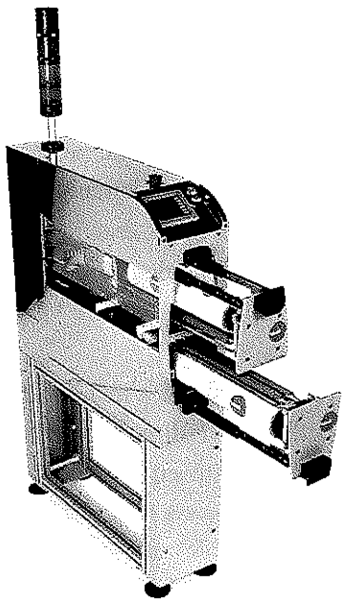

Fig 1 Teknek's SMT Contact Cleaning Machine

Surface mount manufacturing problems caused by board contamination fall into three broad categories:

- Process faults which cause lower board production rates;
- Solder faults which lead to high reject rates;
- Reliability faults which experience of high failure rates at the test stage or in the field.

Sources of Contamination

Particulates can enter the production process from a number of sources. With the possible exception of fineline circuitry, PCBs are rarely assembled in a sterile clean room environment. It is estimated that 80% of these contaminants are introduced into a clean area by people and products, 15% is generated by the products themselves and 5% is created by the room and filtration system.

Key sources of contamination include:

- Human hair. Each person loses, on average 50 hairs a day.
- Lint. Although the specialist cleaning cloths used in the production area are supposingly lint free they can still snag when used for wiping pad areas and stencils, producing loose fibres which can enter the production process.
- Clothing fibers. Fibers from an operator's clothing are a frequent source of contamination.
- Dust. The majority of dust particles are created by shed human skin. On average a person breathes 700,000 skin flakes per day and discards one layer of skin every day.
- Production plant. The plant itself is a prime source of contamination with particles from ceilings, floors, shelving and packaging materials entering the production line.
- Epoxy dust. The routed edges of boards can cause dust.
- Solder paste. Misprinted boards are often wiped "clean" and sent down the line again.
- Solder resist. This can flake away causing surface contamination.

Hair, fibers, dust and plant sources can be reduced

Problems Caused by Contamination

Solder Problem	Cause
Mis-shaped pad, causing poor joint integrity	Paste printed on to contamination
Open circuit	Contamination in solder paste joint
Tombstone due to mismatched forces at each end of the components as the two pads reflow	Paste printed on to contamination
Blow-out (solder balling) during reflow	Paste printed on to contamination
Short circuit	Contamination joins two pads
Mis-shaped pads, tombstoning and poor joints	Contamination in stencil apertures
Solder balling, leading to possible short circuits	Paste remnants from a previous print
Process Problem	**Result**
Stencil blockage	Line is stopped while stencil is cleaned
Adhesive nozzle blockage	Line is halted while nozzle is cleaned or replaced

through well implemented and practical clean room environments. They cannot, however, provide a 100% guarantee of elimination and are expensive to implement and run.

Many assemblers place blame on the bare board supplier for epoxy dust and loose solder resist. However, though the board may leave the supplier in the best possible condition, it must still travel by land, air, and sea to arrive at the customer. The boards then often have to be put into storage before use. Added to this is the fact that the board packaging can cause static attracting particles and one can see that the potential for contamination is substantial.

Every time a PCB requires rework the reliability of the board is compromised. Misprinted boards can be wiped clean but frequently traces of solder paste may remain on the board.

Modern packaging techniques, multilayer boards and tighter gaps between pads all combine to make contamination elimination more vital than ever. Particles as small as 1μm can lead to poor solder joints, open circuits, short circuits or poor joint integrity leading to a field failure. Multi-chip module (MCM) technology requires a smaller pitch between the center of the pads, giving extremely small tolerances between each pad for solder paste replacement. Demands on fine pitch technology mean that even small particles of dust now appear bigger relative to the pad size. The finer the pitch the more problems contamination can cause. Reworking of BGAs can be a difficult and time consuming task. Moreover, solder joints are not easily accessible for rework, therefore these PCBs demand a very high first time pass.

A typical surface mount PCB may be populated with a high density of components with a few hundred connections (solder pads). Therefore, the failure rate of these solder joints must be kept to a minimum. On average up to 60% of boards fail at the test stage due to solder faults.

Starting off with a clean substrate helps increase production yields as residues only become more difficult to remove with thermal and chemical processing. The Table shows the most common problems caused by contamination.

Solder Faults

The application of adhesive on to the board for correct component placement is an important factor, the presence of particles on the board hinders this process in a number of ways. On applying the solder paste it can be placed on top of a micron of contamination; this reduces the joint integrity and can cause tombstoning due to the mismatch of the pads heat coefficients during reflow. Contamination can reduce the adhesion of the solder paste, creating open circuit or bridge joints creating short circuits.

Process Faults

Stencil printing of solder paste is a vital step in the surface mount process. The quality of the print is directly correlated to the process yields. Regardless of stencil type the ideal apertures width over the stencil thickness is 1.5 and it is very difficult for good paste release to be achieved if less than the 1.5 ratio. Any contamination or dust that is drawn into the stencil aperture reduces this important ratio and therefore directly effects the quality of printing. The number of times

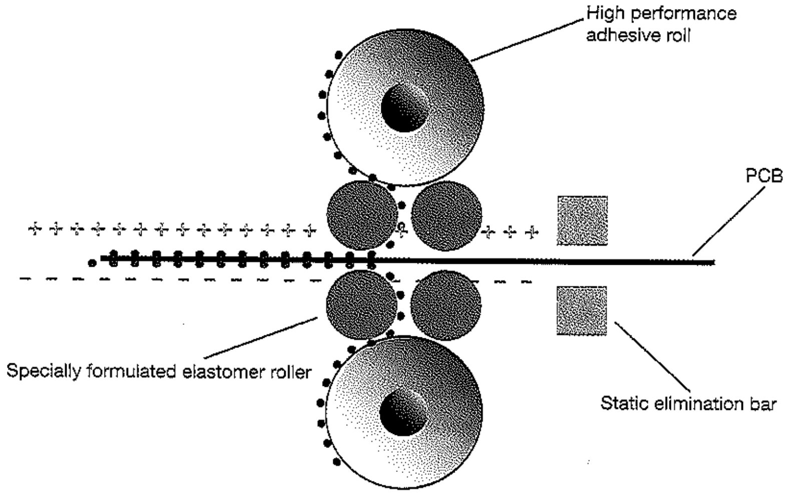

Fig 2 Contact Cleaning Process

the production line must be stopped to clean and wash the stencil is radically decreased by removing particulates from the board before printing. As the pitch and aperture gets smaller for fine pitch technology the maximum stencil thickness becomes thinner to maintain this ratio and the size of the particulate contamination becomes increasingly detrimental to the manufacturing process.

Application of adhesive paste by the needle must be precise to ensure accurate positioning and that the appropriate quantity is applied. To apply the paste pressure is passed down the nozzle. Once the paste is released it causes a slight backward suction which can draw up any particulate on the board or stencil causing a blockage. As a result production lines must be halted while the nozzle is changed. If the particulate is removed from the board prior to printing this downtime can be avoided and the costly replacement of nozzles can be eliminated.

Implementing the appropriate solutions can help reduce the incidence of contamination in the process without increasing manufacturing costs.

Clean rooms are often considered to be the solution to all dust and contamination problems. They have filters to reduce the number and size of airborne particles and have airflow systems designed to reduce the risk of particles falling below critical velocity and being deposited on work surfaces and substrates. However, the ability to configure clean rooms for the random positioning of operators and mobile equipment can adversely affect the system. Moreover, raw materials with dust contamination will still be introduced to the clean room environment.

Contact Cleaning

Manufacturers have found that one of the most effective means of removing particulates from PCBs is just-in-time contact cleaning.

Contact cleaning is a three-stage process (Fig 2). Firstly a special elastomer roller gently cleans the surface of the board removing loose particulates down to 1μm in size. Secondly, these particles are transferred to a reserve-wound roll of pre-sheeted adhesive film where the particles are captured for inspection and disposal. The last stage is a power anti-static system which prevents re-contamination by airborne particles.

Removing dry unattached particles from a bare board immediately before the application of solder paste or any adhesive application reduces solder defects, and increases production yield and product reliability.

Contact cleaning systems have an advantage over other systems using air blowers or vacuums to clean boards. Such systems cannot break through the boundary layer or static layer of air on the surface of the substrate so are restricted in the size of particle which can be removed. As contact cleaning systems actually touch the surface of the substrate this method is much more effective.

The application of effective contact cleaning technology has been proven to improve yields by over 90% and reduce reject rates by in excess of 50%. Contact cleaning equipment varies enormously in terms of quality so should be chosen with great care. For example, the chemical composition of the elastomer roller is very important. Purchase an inferior roller and it will leave behind contamination leading to more rejected boards, decreased yields and increased downtime.

In summary the incidence of contamination can have a detrimental effect on SMT production in terms of lower yields, more rework and greater wastage. Employing effective solutions such as contact cleaning technologies can help manufacturers and assemblers eliminate contamination from the production environment.

by Sheila Hamilton,
Technical Director,
Teknek Ltd

Power Cycling Induced Failure Mechanisms

The specific cooling conditions of hybrid electric cars and the trend to higher current densities in power electronic applications demand the operation of semiconductors at junction temperatures above the common level of 125°C. The maximum operation temperature is mainly limited by the reliability of the assembly and interconnection technology to reach the number of required temperature cycles during lifetime. Known weak points are bond wires and solder joints.

The Power Cycling Test

Fig 1 shows the run of dissipated power as well as upper and lower level T_{high} and T_{low} of the die temperature T_j by which a power cycling test can be described.

IGBT forward voltage drop V_{CE}, forward current I_C, virtual junction temperature T_j and heat sink temperature T_h are continuously monitored during the test. The thermal resistance R_{thjh} is calculated from these values. Since both V_{CE} as well as R_{thjh} may increase during the test, one must distinguish both factors. Therefore either an increase of V_{CE} by 5% or an increase of R_{thjh} by 20% are defined as failure limits.

The V_{CE} failure limit is reached after approximately 10,000 cycles. The sudden leaps in the V_{CE} curve indicate lift-off of single bond wires. Already after 6,000 cycles a slow increase of R_{thjh} can be observed, a sign of solder fatigue. The failure limit of R_{thjh} is reached after about 11,000 cycles. The progressive increase of R_{thjh} raises the temperature T_{high} and thus enlarges the thermal stress for the bond wires at the same time. Also solder fatigue is a significant failure mechanism in this test; it could even be the main failure mechanism.

Bond Wires

For a long time bond wires were thought to be the main weak point for power cycling. There is no adherence in the center of the bond area. The bond wire process could be improved significantly. In the viewpoint of high temperature applications, bond wires seem to be no longer the main weak point, if a proper technology is used. It was shown that by the use of a low temperature joining technique (LTJT) the power cycling capability could be raised even for cycles up to ΔT of 160K.

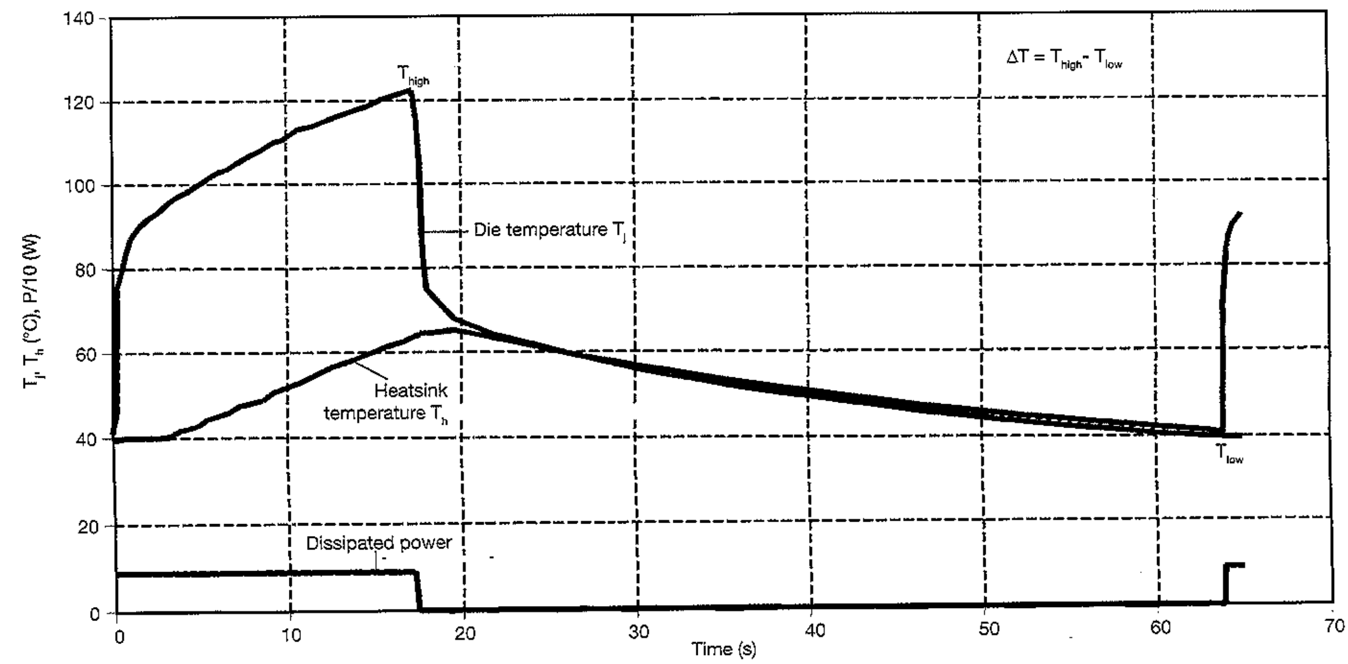

Fig 1 Characteristic Variables of Power Cycling Test Dissipated power and resulting temperatures of junction and heat sink