# Controlling Contamination And Improving PCB Yields

*by Sheila Hamilton, Teknek*

Today's electronics sector is under severe pressure to improve margins, increase efficiency, eliminate waste, and ramp up production volume levels. However, once the latest lean manufacturing techniques have been employed is there still room for further improvement? In the opinion of the authors, there is. One of the key areas for yield improvement is to remove contamination and static from the PCB environment. This article looks at how contamination can be measured and controlled and perhaps most importantly, what solutions can be put in place to negate the effects of contamination on PCB production.

Surface contamination is an issue that electronics manufacturers are taking very seriously. Wastage is a major problem, especially with higher value multilayer boards and as track and gap widths become smaller. Less wastage leads to greater throughput, lower costs and ultimately, increased profitability.

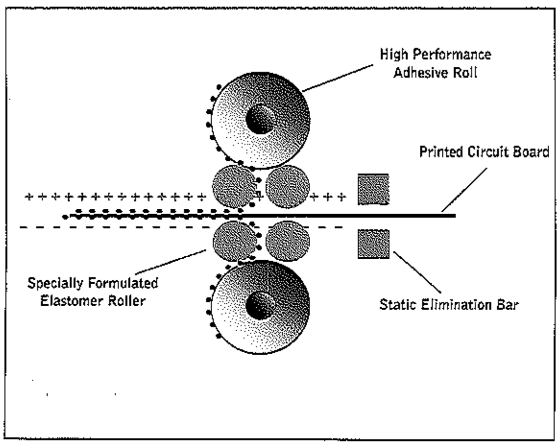
Contamination is inevitable because of a number of factors. On the whole, lines are not located in cleanrooms. When the PCBs are separated, a static charge is generated which acts like a magnet for surface and airborne particles. In addition, the PCBs can release epoxy dust from routed edges and loose glass fibres can contaminate the board.

Other key sources of contamination include human hair, dust, lint from cleaning cloths, fibres from clothing, dust, skin flakes, epoxy dust, solder paste, solder, conveyor debris and packaging materials.

A number of problems in the manufacturing process can be caused by contamination including: blocked stencils, blocked adhesive nozzles, solder balling, tomb stoning, poor shape of pad, reliability of joints and long-term joint integrity.

The first step in eliminating contamination is to carry out a contamination audit to identify the sources of contamination in the PCB environment. Audits using air sampling alone can quantify the number of particles in the air, however this neither identifies the type of particle nor does it take account of any particles which are too large to be airborne. Therefore a special contamination audit technique has been developed which uses a hand-held elastomer roller to pick up contaminants from any surface within the production environment. These particles are then transferred to a special adhesive pad for easy identification of the main contaminants in an area of the production facility to enable the most appropriate, cost effective measures for reducing contamination and increasing yield to be taken. The results of these tests are transposed on to a Contamination Matrix. There are two types of Contamination Matrix.

*Figure 1 – The contact cleaning process*




The first type is a Contamination Matrix for each department on which all the samples taken in that department are logged, together with their analysis results. This analysis makes it immediately obvious where housekeeping procedures or manufacturing protocols need to be changed or improved within the department. It also highlights what the major types of contamination are. The second type is a summary Contamination Matrix which covers the facility as a whole. This shows each department with their total contamination levels logged and is helpful in illustrating the flow of contamination throughout the plant. Careful analysis of these matrices can reveal the major areas where contamination poses a high risk to process yields and appropriate action can be taken to reduce the contamination levels.

Once remedial action has been taken, the contamination audit should be repeated in that area to measure the change in contamination levels. Close monitoring of improvement in the process yield should also be carried out. The revised matrix can then be overlaid on the original one to show where improvements have been made.

### Contamination Audit methodology

The Contamination Audit measures both the air quality and the type and concentration of surface contamination within the audit areas. The objective of the audit is to identify the sources of contamination through careful analysis of the samples and observation of the production processes.

ding in Textiles) with funding by the Belgian Science Policy.

TFCG Microsystems believe that real-life stretchable appliances will be hybrid. They will contain rigid or flexible components connected with stretchable circuitry. The circuitry will stretch and bend like rubber or skin while preserving its conductivity. At the lab, the aim is to combine stretchable interconnection technology with flexible circuit technologies, as developed in the EC-SHIFT project. An example is the Ultra-Thin Chip Package, or UTCP, which is only 100 micron thick.

## Golden undulating horseshoe patterns are more flexible

Recently, researchers at the lab reported on a new method to design and make such elastic interconnections for stretchable electronic circuits. They embedded interconnection wires with a 2D undulating pattern in an elastic silicone film. The 2D springs were designed and optimised in co-operation with the IMEC IPSI/REMO group, which is specialised in mechanical modelling and reliability prediction. They differ from other comparable designs in the following optimisations. First, the researchers inferred, based on finite element analysis, that an undulating horseshoe shape is the ideal form for the connection wires. It dissipates the stretching and flexing stresses better than comparable elliptical patterns. Second, they further improved the stress resistance of the interconnections by splitting each interconnection wire into four parallel wires with a smaller width. Lastly, in the initial technology development phase, they chose gold as a material for the wires, because of its high ductility, which again allows for greater stress resistance.

The resulting interconnection wires consist of four parallel tracks, each 15µm wide, and are made of a 4µm-thick gold layer. The tracks are coated with a 2µm-thick nickel layer for soldering to components. At regular intervals, in positions where the deformation stress is calculated to be minimal, neighbouring tracks are cross-connected. This allows for fail-safe operation in case of fabrication errors or mechanical failure.

## Assembly practices

The interconnection wires are embedded in a silicone polymer substrate: polydimethylsiloxane (PDMS). In itself, PDMS is an electrical insulator, but it can be made conductive by adding silver particles. While not as good a conductor as copper or gold, this modified polymer can carry a signal over very short distances. Should a wire be overstretched, resulting in a micro-crack, the surrounding polymer will still conduct the signal, bridging the gap created by the crack.

The team at TFCG made interconnections with different angles and radii for the horseshoe shape. They tested the circuits by stretching them in the longitudinal direction to the point of electrical failure, which would be caused by a rupture in the gold tracks. The best interconnection stretched from 3 to 6 centimetres without failure. Moreover, all interconnections recovered their conductivity when returned to their normal length.

To assemble the stretchable interconnections and the more rigid electronic components, joints were soldered using normal electronics assembly methods. Next, the silicone polymer was molded around the assembly, taking care not to cause any bubbles in the silicone. For each design, a dedicated mould was made. This mould takes into account the locations on the assembly where the rigid parts are located. At these locations, the silicone wrapping should be thicker so that the circuitry is locally less stretchable.

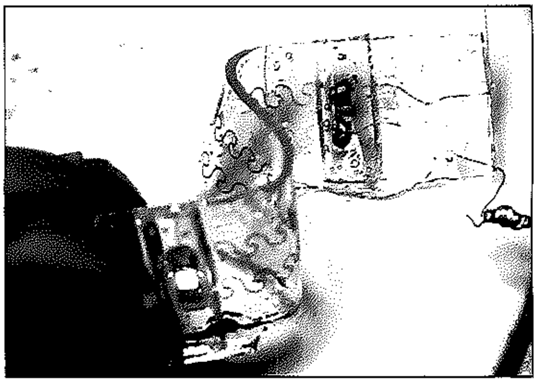
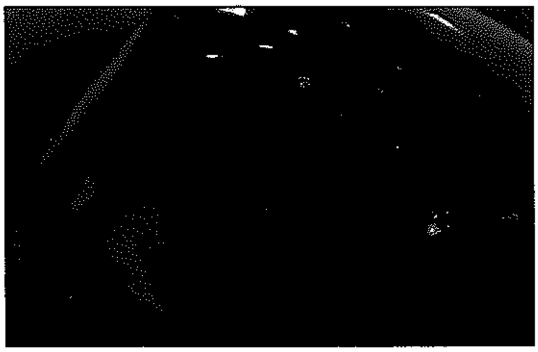


*Figure 2 – Blue LED embedded in 500µm thick silicone substrate*

*Figure 3 – Led powered by inductive coil embedded in PDMS*



## What the future holds

Researchers are now studying the use of conducting materials other than gold. Copper, the standard material in PCB manufacturing, is the obvious candidate. Wiring with copper is of course much more cost-effective than with gold, but the main reason for choosing it is the drive to develop a technology that is compatible with existing industrial PCB and assembly practices. This will greatly facilitate the transfer of the technology to a production environment.

Within 3 years, and based on its current research results, the TFCG Microsystems lab expects to have technology and a demonstrator that can be commercialised. They believe that the first flexible and stretchable appliances will most probably be used in intelligent clothing, with medical applications following later. The circuits that are being worked on will not only be flexible and stretchable, but also washable – which is a big step forward for intelligent clothing. For first commercial products, expect to see clothing with signalisation, using LEDs and sensors, for example to track movements.

The first step in producing the Contamination Audit is to develop a process flow map. There are two basic types of process from a contamination point of view. There are those which generate contamination as an inherent part of the process such as drilling or wet processing, and those which are sensitive to contamination like solder paste printing, resist application and exposure.

Next, plans of every area to be audited should be prepared. These plans should show the positioning of all the equipment in the room and the layout of the doors, as well as any other points where materials or products enter the production area. One corner of the plan should be reserved for notes detailing the type of clothing operators are wearing plus any other relevant details. The plans will be used to identify exactly where the samples have been taken so that each Contamination Audit can be repeated in precisely the same manner.

An instrument called a tyndleometer is used to measure the air quality. This provides a qualitative measure of the number of particles in the air and is ideal for environments where the volume of particles would quickly block up a standard air monitor. The sampling process is very quick which allows real-time analysis of air quality at various locations within a room. Sampling positions are marked on the department plan. Surface contamination samples are collected using a special hand-roller (such as the Teknek DCR roller) which can pick up loose particles down to one micron in size. The roller is rolled over the surface to be examined and the particles are then transferred onto a special adhesive sheet with a grid pattern on it. This concentrates and permanently holds the contamination for later analysis. The sample is then covered with a release sheet containing details of the precise location of the sample. Any relevant notes about the sample area should be noted on this sheet, peeling paint on equipment, for example.

Key areas to be sampled include floors, walls, ceilings, inside surfaces of PCB equipment, the PCBs, transport systems and operators.

Once the samples have been collected, the primary method of analysis is to use a microscope (minimum 60x magnification) to inspect each square in the sample grid. Using reference photographs each type of contamination can be identified and the numbers of each type of particle logged on the Department Contamination Matrix.

Any unusual or unidentified contaminants in the sample should be photographed for further investigation. These results should also be entered into the Contamination Matrix.

Left unchecked, contamination can lead to serious problems which will impact on PCB production levels. Potential problems include:

- dust and other forms of particulate contamination are major causes of rework and scrap in the production of high density circuitry;
- problems associated with dust are long standing in the industry but the situation has become more acute with the advent of higher packing densities in circuits. As line widths reduce, the amount of tracking per unit area grows, with the result that there is a much greater risk of a particle of contamination, of sufficient size to cause a fault, falling on the PCB. The risk also rises in proportion to the number of layers in the circuit board.

Cleanrooms are often considered to be the answer to all dust and contamination problems as they incorporate air filters and ionisers to remove airborne particles and static charges from the air. There are three main classifications of cleanroom: Class 100, Class 10,000 and Class 100,000. These classes refer to the number of particles, 0.5 microns or larger, which are permitted in each cubic foot of air. It is often assumed that because of this limit it is only necessary to clean the parts on entry to the cleanroom and they can wait to be processed without any danger of recontamination. However, Class 10,000 and Class 100,000 can still have a substantial number of particles above 5 microns and when these land on the substrate it can lead to serious problems. Another drawback of cleanrooms is that they are expensive to install and the procedures to maintain the stringent levels of cleanliness are very time consuming.

Another method which is commonly used in the electronics sector is to wipe the underside of the stencil with a dry or wet cloth. Dry cloths can be used immediately prior to processing and so reduce the time in which airborne contamination can affect the part. If solvent is used with the cloth, residue from the chemicals can cause problems such

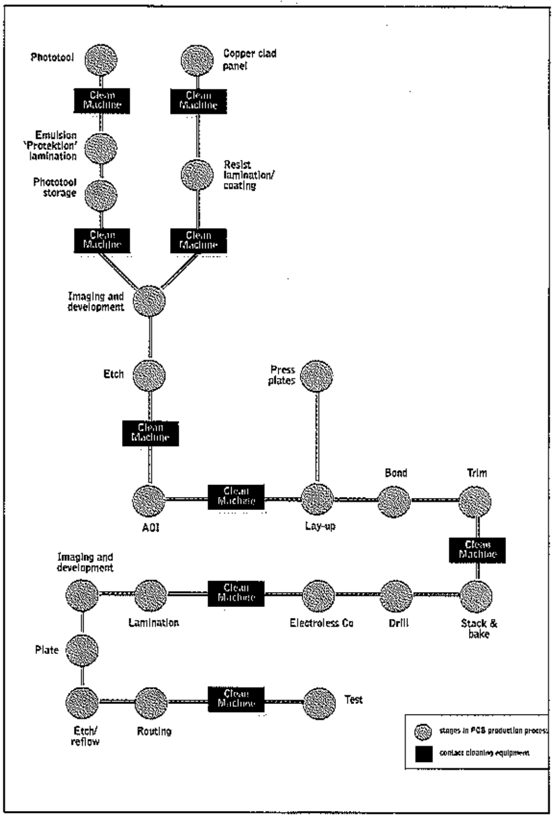



*Figure 2 – The areas which can benefit from just-in-time contact cleaning*

as resist delamination when the board is processed. The cloths themselves can cause contamination with loose fibres attaching themselves to the boards leading to considerable downtime and production yield reduction. Another source of contamination in a PCB production line is airborne particles caused by solder fumes. Airbourne particles range in size from 0.01 to 1000 microns so if the pitch of the component leads is 20 microns these particles can cause problems. In fact, the finer the component lead pitch, the greater the risk of particles causing errors, wastage and rework.

The most effective solution is considered to be pre-cleaning the boards using contact cleaning equipment to provide 'Just in Time' cleaning of the board immediately prior to it entering the production process.

Such equipment uses a highly-engineered elastomer roller to remove particles (down to one micron in size) from the surface of the substrate. The particles are then transferred to a roll of specially formulated adhesive film which stores the particles for later examination and disposal. Once the board has been cleaned in this manner it is then put through a static neutralization unit to remove any static charge which could attract particles and recontaminate the boards (Figure 1). The key benefits of using contact cleaning to pre-clean boards are:

- improved yields;
- faster production;
- higher quality;
- higher reliability of joints;
- less rework and wastage.

Other actions that can help minimise the incidence of contamination include:

- establish housekeeping protocols to document and monitor the residual level of contamination in the room;
- the air supply to the machines should be filtered;
- routine cleaning of inside the production line equipment should be part of the preventative maintenance procedure.

**Key areas for contact cleaning**

Lamination is not the only circuit manufacturing process which is dust sensitive. The requirement for 'Just in Time' contact cleaning technology is evident in many other departments of circuit manufacture (Figure 2) including:

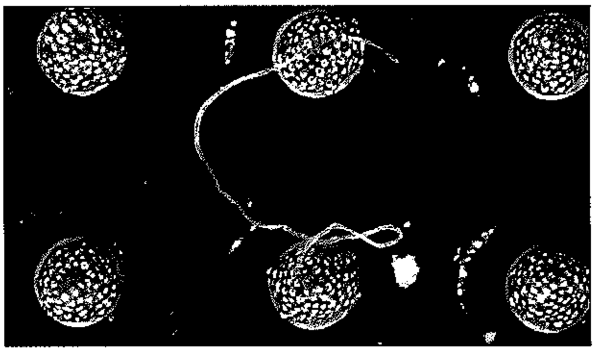


*Figure 3 – BGA with fibre and dust*

- photography: unexposed photographic film, taken straight from the box can be contaminated with particles generated during the slitting operation which converts the rolls of film into sheets prior to boxing. Cleaning all film, exposed and unexposed, prior to contacting will eliminate pinholes caused by this loose surface contamination. In addition, cleaning of diazo or silver halide films should be undertaken each time a phototool is used to reduce touch-up time and improve the quality of fine line circuit boards. Phototools which are to be laminated with an emulsion protection film such as 'Protek', should be cleaned prior to lamination. Combination cleaner/laminator units are available for this application;
- resists: whether dry film, screen printed or curtain coated, all resists require the copper panels or inner layers to be cleaned to prevent pinholes or blistering of the resist. Specially designed units are available which can be retrofitted to most models of dry film laminator. Stand-alone units, located beside the laminator, can also be used. The resist coated panel should be cleaned again immediately prior to exposure;
- screen printing: all screen printing operations result in better quality parts if the substrate is cleaned prior to printing, as dust on a part can either be encapsulated in the printing ink, causing rejection of the part, or can clog the mesh. Should the mesh become clogged, several parts may be printed before the fault is noticed and these boards may have to be scrapped;
- mutlilayer build: the cleaning of each individual layer of a multilayer package prior to assembly will prevent particles of contamination being compacted into the copper tracks of inner layers during pressing. Caul plates and other tooling ancillaries should also be cleaned;
- automatic optical inspection: the use of automatic optical inspection equipment to locate defects in PCBs is becoming more prevalent. However, the inability of many of these units to distinguish between a defect and a particle of dust will cause false readings. Cleaning the board before scanning can save operators spending substantial amounts of time investigating these phantom defects.

**Conclusions**

Substantial yield improvements can be gained by adopting a systematic and proactive approach to contamination monitoring and control. The Contamination Audit and its related matrices provide a flexible framework with which to assess the contamination levels throughout the production facility and identify the most beneficial contamination control methods. Pre-cleaning of PCBs before they enter the production process is essential. One of the most effective methods of doing this is to pass the boards through contact cleaning and static neutralisation equipment prior to the production line in order to remove surface contamination on the substrate.