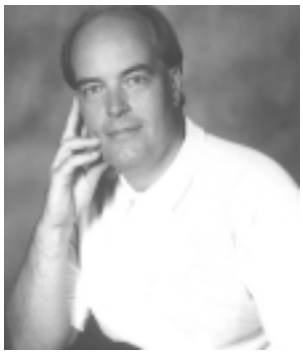
Solder resist does it perform as well as it could during assembly?

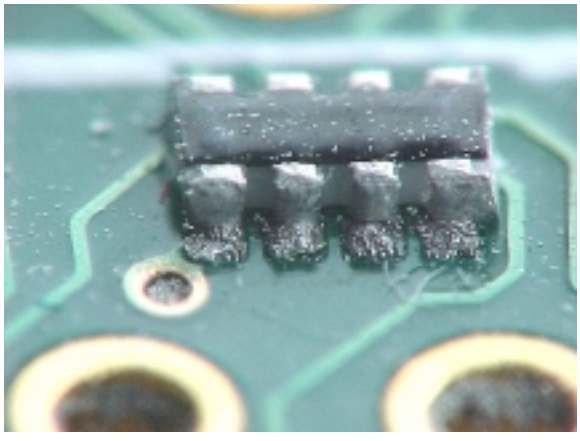


Practically all printed circuit boards are coated with a thin layer of material which provides mechanical protection to the circuitry, insulating properties and reduces the possibility of soldering defects during assembly. The material is typically green in colour and referred to as solder resist or mask, but does it meet the requirements of modern assembly or worse still does it contribute to defects ?

Solder mask has been used in the industry for over 20 years and may be applied to the circuit board by screen printing, spray, curtain coating or in the past by hot roll lamination as a dry film. Today there are probably six key suppliers in the industry offering a range of products all complying to either European or IPC standards. But how do products perform in the ever changing assembly process. This is the subject of a new project by the National Physical Laboratory (NPL).

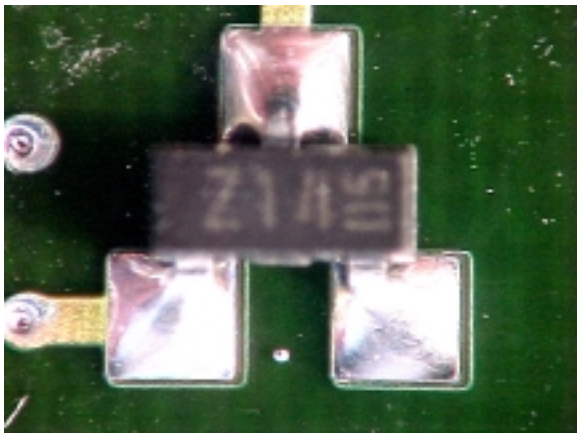
There are a number of process issues which can be directly effected by the type of mask used or the way it is processed and needs to be considered by production and design engineers.. Each of these areas are being examined as part of a new NPL project to identify a test method for masks. The key issue is considered to be directly related to the degree of solder mask cure which has been an issue for many years. Existing measuring techniques do not give the confidence needed by the industry and variability can contribute to defects. The test method should be as simple as existing techniques but provide a useful measure of performance. This would allow both fabricators and assembly companies to perform the test.

Outgassing from the solder mask can cause voiding in underfills during normal cure or in the new no flow underfill materials during reflow. Underfill is used on flip chip assembly to reduce thermal differential expansion between chip and board. In many applications to day like uBGA underfill is simply used to provide mechanical support on this boards. Underfills on the surface of the board can exhibit voids which reduces the benefits of the process. Currently boards can be baked to eliminate the potential voiding problem but this is a no value added process particularly on low margin products.



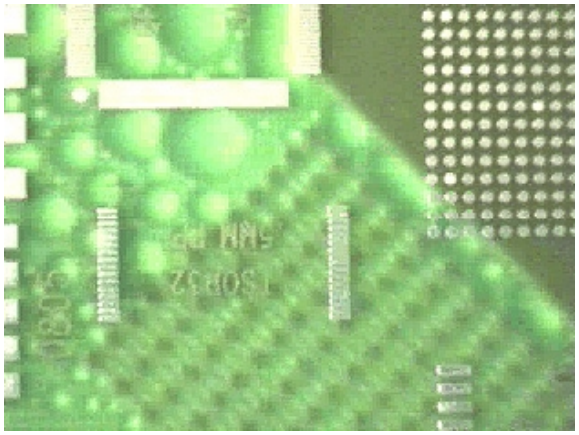
Solder balling or incomplete reflow of solder pate can result from solvents coming out from masks during reflow. In this case the solvent does not come from the printed board manufacturing process but can be trapped in the mask. Often boards are washed off if the quality of the solder paste print is considered to be poor. There are companies who clean boards prior to printing to improve printing yields, both processes can cause the mask to have a small amount of solvent in the surface layer.

During reflow soldering the solvent from the cleaning process will evaporate from the surface of the mask making it difficult for the paste to reflow completely. This leads to non reflow or solder balls and the paste manufacturer often gets the blame incorrectly. The example resistor network is a typical of paste reflowed on a board that has been recently cleaned prior to paste application.



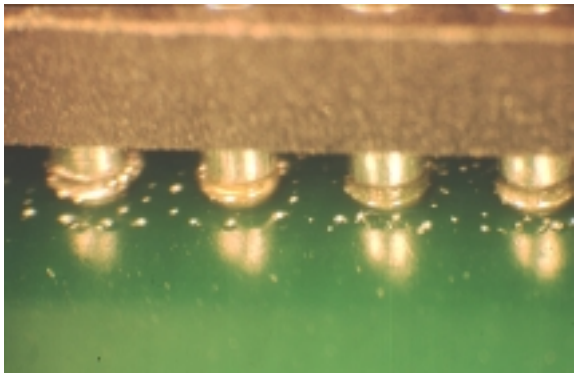
The solder mask surface has been shown to pick up solder balls during wave soldering as the board exits from the wave. The balls can be random due to spitback from the surface of the solder bath and wave or repeatable appearing only between groups of termination points. The key repeatable feature is the balls adhering to the mask. Trials have demonstrated that the surface of the mask can have a role in solder adhesion regardless of where the balls adhere to the surface.

Surface roughness has been shown to effect masks, taking a mask and putting it through a mechanic cleaning process has been seen to reduce the incidence of ball adhesion. Degree of mask cure has also been suggested as a potential cause.



Another issue highlighted by engineers is the outgassing of the mask when in contact with the surface of the wave. Many people assume that this is related to the volatile content of the flux still on the base of the board. Running test boards through the wave can often demonstrate that this is not the case and the outgassing is purely from the mask surface. Some years back a solder equipment supplier in conjunction with a German University undertook trials with high speed cameras to demonstrate the problem.

The growing interest in Pin-in-Hole/intrusive reflow has shown some issues with solder paste reflowing on the surface of the mask and leaving solder balls after reflow. It is often necessary to print paste on the surface of the solder mask and through hole to increase the volume to fill the through hole. In some case the component may dictate that the design of the solder paste print. Many trials conducted by this columnist with different solder masks and paste have only shown a limited problem but some masks are an issue. It should be born in mind that solder resist was never designed or specified to compatible with this process, thankfully many materials are.



There are a range of process problems or issues directly relating to the mask but do we specify it correctly from our suppliers ? Often not. Many company drawings, procurement documents and specifications simply say solder mask and normally state the colour green, not very specific is it. It is hoped that with the help of the NPL project we will be able to understand the true cause of some of these problems and find a test method which works effetely on materials. The ultimate would be to have a test method that is meaningful and could one day feature in international specifications.

We have highlighted some specific process problems with assembly and the selection of solder masks, a product that is often taken for granted as the green coating. If any reader has encountered process problems possibly related to solder masks let us know as it may be useful to understand other issues seen during assembly. The more process problems considered during the project will help make the results more meaningful to the industry when completed.

Bob Willis is also a process engineer providing engineering support in conventional and surface mount assembly processes. He runs production lines for suppliers at exhibitions and also provides seminar and workshops world wide. Bob has one of the largest collection of training videos, interactive CD-ROMs and training material in the industry. Bob will be presenting four Master Classes at APEX in California, he will also be presenting classes at SMT Nuremberg in Germany for those engineers visiting the show. For further information on how Bob may be able to support your staff contact him via his web site **www.bobwillis.co.uk**