ELECTRONICS MANUFACTURE-Selective machine soldering

There are a number of designs when one wants to use a machine soldering process, but needs to confine it to just a section of the board. For example:

to keep specific through-holes clear, so that post wave-solder assembly operations can be carried out

to keep surfaces like gold-plated edge connectors free of solder, so that they function properly

increasingly, for designs which use wave soldering only for specific components, such as connectors, and the remainder of the underside carries complex surface mount components that would be difficult or impossible to wave solder.

The first two of these requirements are normally met by temporary solder masks or physical barriers, both of which mask off areas of tracking and pads from the solder wave. For the third, custom pallets are commonly used, but there are an increasing number of selective soldering solutions.

Temporary solder mask

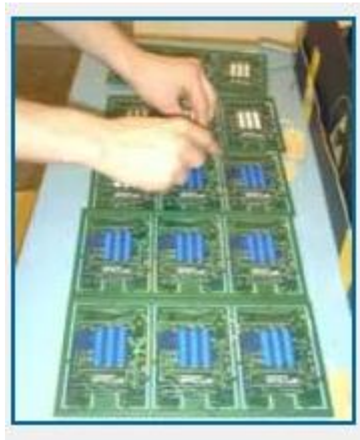
There are three approaches to providing a temporary solder mask:

Soluble coatings. These are usually applied by painting or dabbing onto the surfaces to be protected. The materials can be selected to be water washable or solvent soluble, so that they are able to be removed during the cleaning process.

Peelable tapes, such as self-adhesive polyimide materials, which are pressed into position. These can be applied manually or by machine, and by the board manufacturer or the assembler. Often used for protecting gold fingers, they are most useful when the areas to be masked are few and rectangular.

Peelable liquid masks, which are cured after they have been applied, typically by screen printing at the board manufacturer1, or by dispensing at the assembler. The majority of peelable masks are latex-based, but the range also includes vinyl, silicone and urethane based chemistries.

1 Board fabricators sometimes use peelable solder masks during manufacture. Examples are: in Hot Air Solder Level to keep solder away from areas such as gold contacts; in selective plating applications such as protecting gold electroplated areas during electroless gold plating.



Removing peelable solder mask from boards

Peelable masks should peel off easily from the surface and the holes, but there is often a problem in achieving complete removal: a dotted line of residual mask may be seen around the perimeter of the peeled area; material may be left in via holes. The nature of such adhesive residue is critical: it must either not be detrimental to circuit function, or else be dissolved completely in a cleaning process. Inspectors are rightly suspicious of residues in plated holes: even if non-conductive, they may trap flux and contamination during subsequent assembly. The greatest problems are seen with liquid masks, as these penetrate the holes and vias to give a mechanical key.

Unfortunately, some resins can become very difficult to remove once they have been subjected to reflow temperatures, so it is important to make sure that the mask material has been designed to cope with the double soldering process. Improved results have also been reported from buying boards with the peelable mask a little under-polymerised, so that it reached the right physical properties only after assembly.

An alternative is to apply a selective mask post reflow/pre wave. This can be done by hand, or in volume by using selective coating equipment (dispensing or spray).

Peelable masks vary widely in their ionic properties and flux absorption characteristics, but should always be removed either before or during the cleaning operation. Not only may the mask itself degrade, but flux tends to become trapped underneath. Whether the resulting contamination originates from the mask or the flux, corrosion during life may be the outcome.

Physical barriers

There are many mechanical ways of restricting solder flow. Where only small areas of the board are to be screened off, three solutions are:

Covering gold plated edge connectors with a non-wettable channel made of titanium, or high temperature plastic composites. These channels have the added advantage of helping to keep the board rigid

Using dummy components with plastic leads to block out via holes as required. These can be manually placed or sometimes even auto-loaded, although this can take up valuable cassette loading space

Building mechanical guides into the solder wave itself, to prevent the flow of solder from reaching connectors and gold plated fingers on the board in specific locations. This method makes the solder wave equipment specific to a board type.

However, the custom pallet may well be the preferred way to assemble the increasingly common designs where only a few areas are to be wave soldered. The pallet screens most of the underside, with apertures for the solder areas, and internal relief to allow flush fitting on top of SM components. The windows need careful design, with chamfered edges to aid solder access.

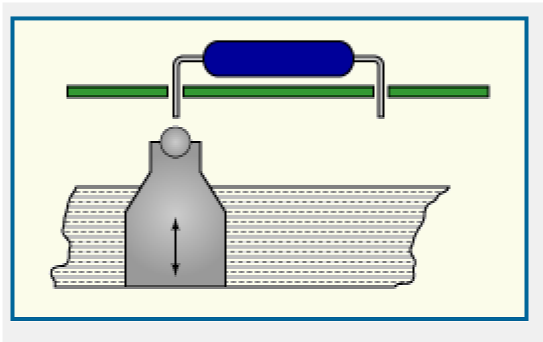
Selective soldering equipment

There are a number of ways of automating selective soldering, the simplest of which (at least in concept) is to use a robotically-controlled soldering iron, where solder and iron are applied sequentially to the joints to be attached. This is most suitable for applications where only a few joints are needed. Where more joints are to be made, some alternatives are illustrated below:

Pin transfer

An array of transfer pins with a concave top surface, mounted on a carrier plate, is submerged in a solder bath. When raised, each pin holds a solder ball, which is applied to the joint from below, so that all joints are made simultaneously in one cycle. The transfer pin remains in contact with the solder bath, and this ensures adequate heat transfer to the joint. Flux is applied by transfer pin or spray. This process is relatively fast, but needs custom jigging, so is best suited for high volume applications.

Figure 2: Pin transfer method for selective soldering



Mini-wave

This is a more flexible process, based on a CNC controlled transport system which moves the board sequentially over preheat, spray fluxing and solder stations. Flux is sprayed selectively, but the whole board is preheated, to prevent warping. A nozzle directs a fountain of molten solder onto the joint area, and the overflow falls back into the bath along the nozzle sides. Given a programmable nozzle and conveyers, product changes are easy to make, requiring only a new program.

'Solder fountain' used for selective soldering

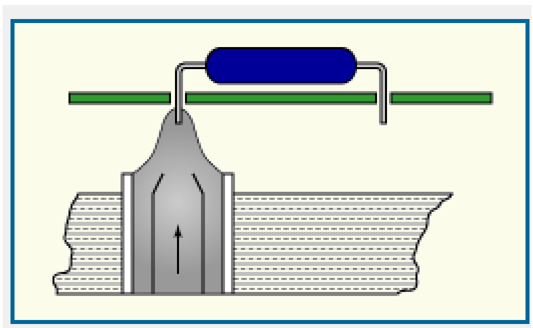


Figure 3: Mini-wave method for selective soldering

Stencil dip

This process is a variant of the custom pallet system, but requires just two stencils, one for flux spraying and the other for soldering. The soldering stencil is made in heat resistant material and completely covers the circuit board, with apertures corresponding to the solder joint locations on the board. The assembly is lowered onto the soldering stencil, which is then lowered onto a soldering area created in a solder bath by a pump and solder nozzle. Simultaneously, flux is sprayed on the next board. This system can be made highly flexible, provided stencils can be changed quickly.

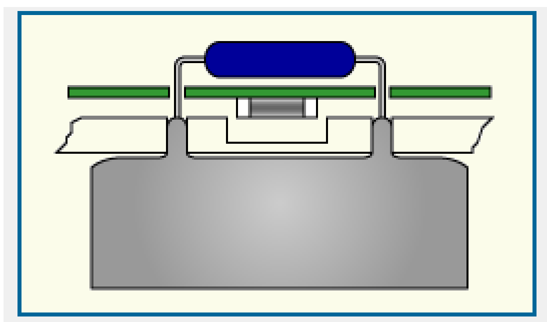


Figure 4: Stencil dip method for selective soldering

Although wiper blades can be used to skim oxide from the solder surface, most of these processes benefit from the use of a local nitrogen atmosphere to minimise dross formation.

Author: Martin Tarr

Source: http://www.ami.ac.uk/courses/topics/0227_sms/index.html