ELECTRONICS MANUFACTURE-Intrusive reflow

The reaction of process engineers with a background in reflow soldering to any description of the many methods of applying liquid solder will probably be to throw up their hands in horror! Certainly all these techniques are expensive in equipment or jigs, or costly in labour. There is also some concern that they will lead to reduced yield and increased rework.

The alternative of using solder paste and reflowing all the components is attractive, and cost reduction and process simplification are major driving forces behind what is called 'intrusive reflow' or (more descriptively) 'pin-in-paste' or 'Pin-In-Hole Reflow' (PIHR). The process sequence for this is shown in Figure 1.

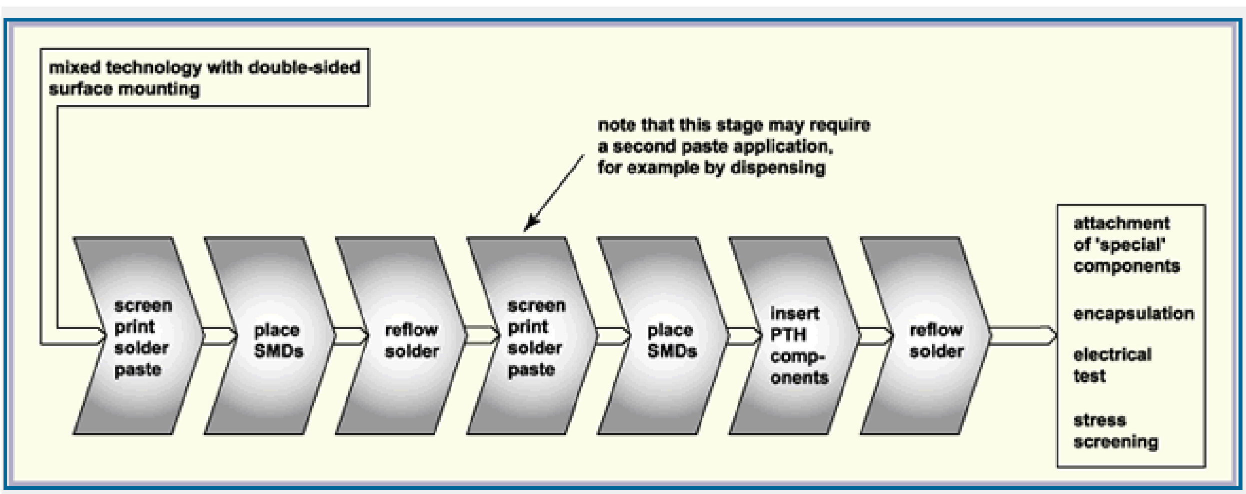


Figure 1: The intrusive reflow process route

The proponents of PIHR would claim that it reduces the number of manufacturing operations and the cost of assembly by replacing hand or wave soldering operations and the associated rework. It also involves just a single flux, which eliminates any problems of compatibility and cross-contamination.

The key prerequisite for PIHR is that components must be able to withstand the reflow conditions. Particularly as we move towards lead-free solders, whose higher melting points require increased zone temperatures, this can be a consideration. In fact some of the reluctance amongst assemblers to adopt PIHR is that connector housings made of materials that will withstand reflow conditions have to be made of more expensive base resin, and are consequently more expensive. Electrolytic capacitors also present a challenge.

Overall PIHR produces an effective solder joint. However, the solder volume is limited to that deposited as paste, and is consequently generally leaner than the normal wave-soldered through-hole pin (Figure 2). Visual standards may need to be reconsidered and staff retrained. An example of the results of the intrusive reflow process can be seen in the associated video. VideoClip1 Intrusive reflow of axial components.

Views from both sides of a Pin-In-Hole-Reflowed connector

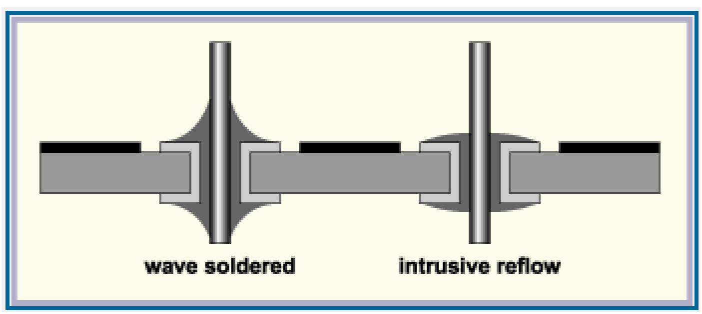


Figure 2: Different profiles of wave-soldered and PIHR joints

The main issue within intrusive reflow is getting enough solder volume. It is possible to calculate the amount needed from the dimensions of the aperture, making allowance for the fact that only 50% by volume of the paste converts to liquid solder, the balance being flux – though only 10% by weight, flux is substantially less dense than solder.

We can deposit more paste by allowing paste to penetrate into the hole during the printing process. The degree of penetration depends on the printing speed, the width and angle of the blade, and the squeegee pressure. With the correct settings penetration of 45–85% can be achieved, but the settings are important to ensure that there is not excessive bleed onto the underside of the board, which will cause contamination to spread from board to board (Figure 3).

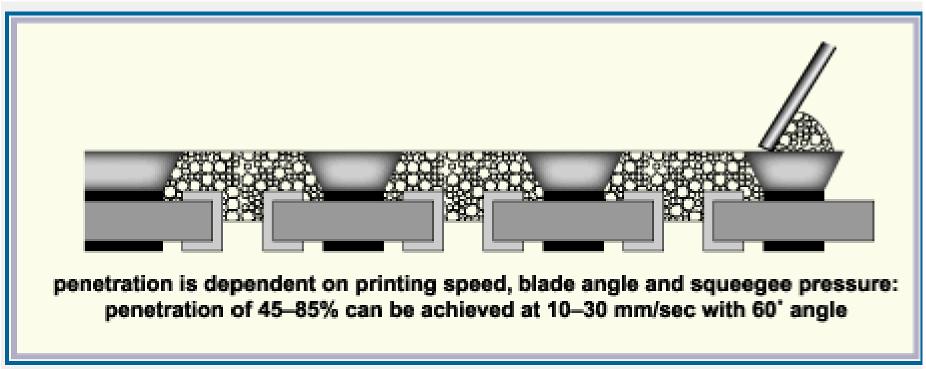
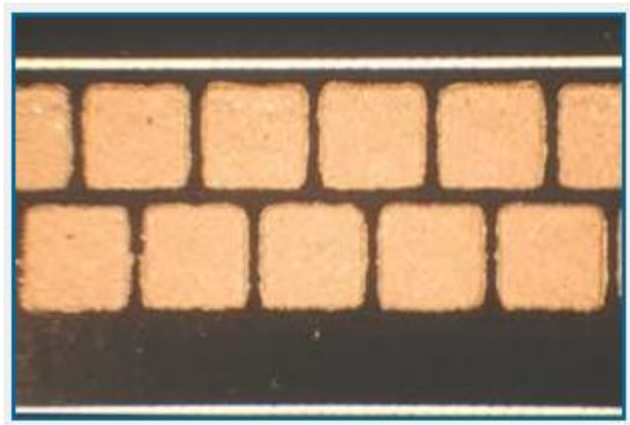


Figure 3: Penetration of paste into through holes

Given that just pasting into the hole will generally not give enough solder, we have to consider how else to create a greater solder volume. The simplest way is to expand the deposit beyond the periphery of the copper track, relying on the fact that, when melting, solder will withdraw from the solder mask and form part of a single joint. [Note that, if solder paste is to be printed on solder mask, then the correct material for the application must be selected] This pull-back is shown in the linked video clip, which also indicates that there is little margin between satisfactory joints and ones where bridging occurs. VideoClip2: Intrusive reflow of connectors, showing potential for bridging.

An alternative for increasing the deposit, shown in Figure 4, is to expand both diameter and length, and this is frequently done on connectors and for other applications where leads are closely spaced in one direction. Where yet more solder is required, but there are two rows of pins, then the ‘teardrop’ design is appropriate. Other assemblers prefer to use square deposits, seeking to maximise the area of the solder paste print.



Square paste deposits

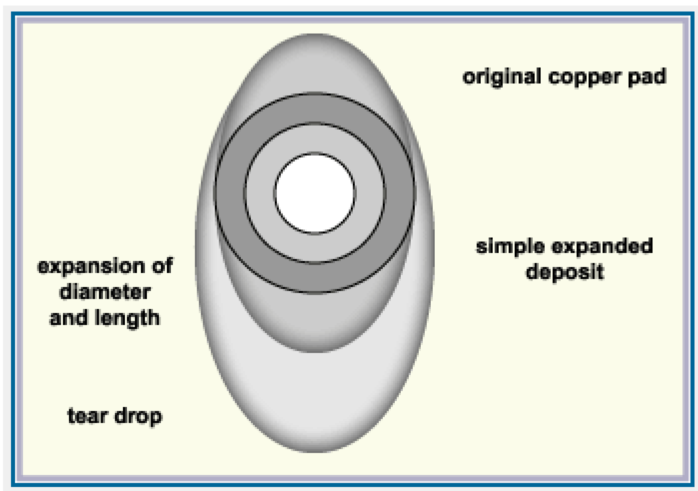
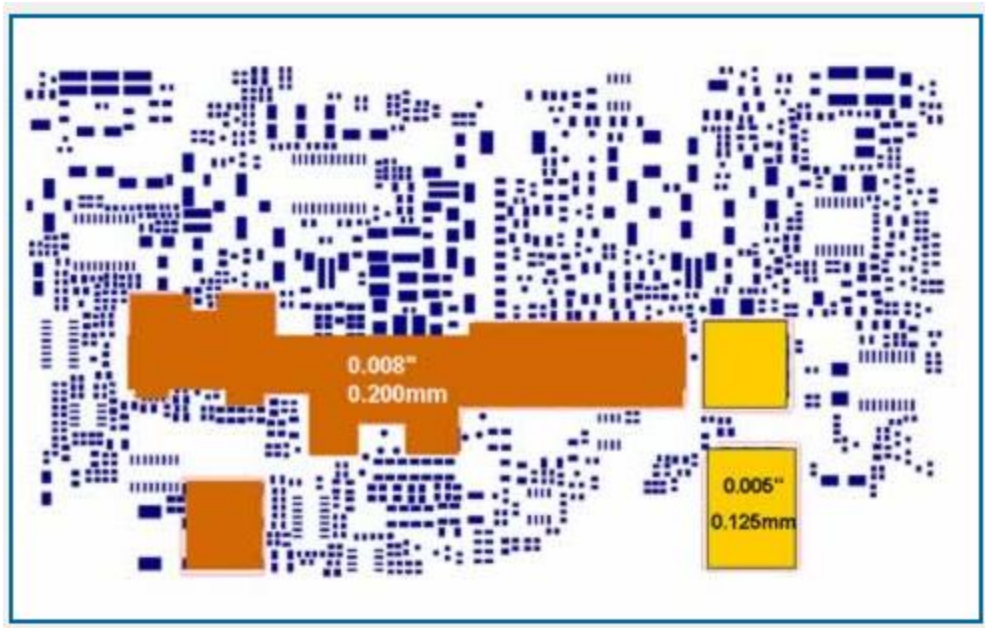


Figure 4: Options for different aperture designs

Of course there are some components, such as connectors with multiple rows of pins, where none of these approaches can be used to give sufficient solder paste to form satisfactory joints. The alternatives are then to dispense additional solder paste where required, as a separate operation, or to use a multilevel stencil as shown in Figure 5. The multilevel stencil, typically produced by electroforming, is particularly useful where there are both through-hole and fine pitch components, because the technique allows the fine-pitch areas to be created in a thinner foil, with PIHR components provided with a thicker deposit.



Multi-level stencil

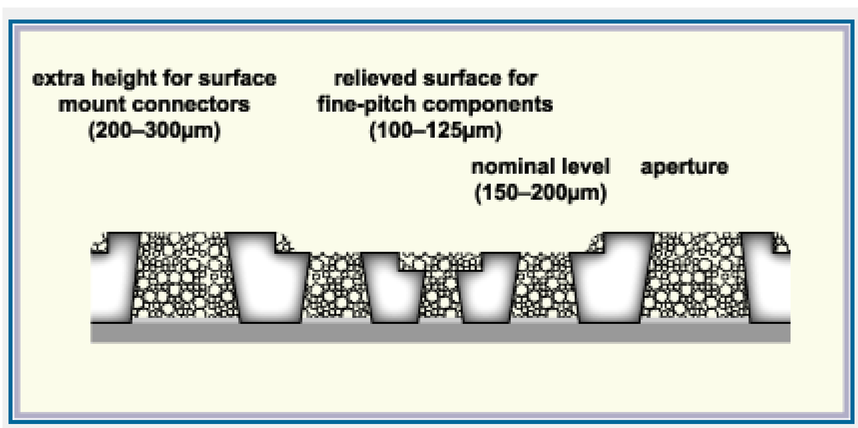


Figure 5: Schematic cross-section of a multi-level stencil

Another factor to be kept in mind is the pin projection. As suggested in Figure 6, the pin projection should be kept between 1.0–1.5mm, so that excessive paste is not displaced by the lead, to be lost to the final fillet.

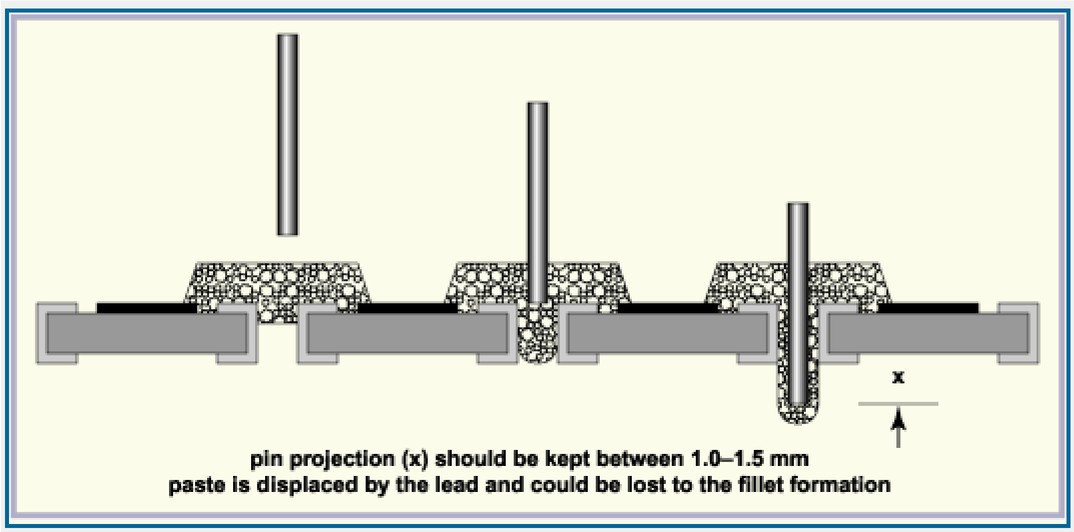
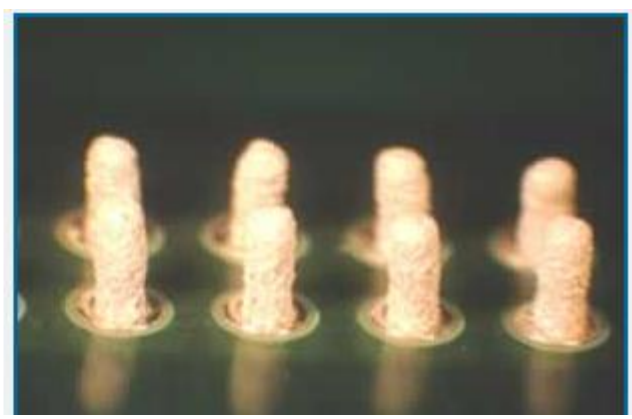


Figure 6: The impact of lead projection on the paste deposit

The problem with visual standards has already been noted, but there is also an additional problem at the test stage that the insertion process can lead to some build-up of solder on the pin tip and to flux residue on the pins, which means that it is recommended that pins should not be used as test points.



Hole paste displacement by inserted leads

Author: Martin Tarr

Source: http://www.ami.ac.uk/courses/topics/0226_pip/index.html