Combined Footprint – the Effect of Collecting of the Solder Material

Valentin Tsenev and Valentin Videkov
Technical University of Sofia

***Abstract*—In the production of electronic modules through soldering the use of PiP (Pin in paste) technology is increasingly required. With this technology bulk components are soldered using the advantages of SMT (Surface mount technology). The article examines a non-standard way to obtain successful soldering with PiP. The process of applying solder paste to obtain a combined footprint is considered and studied. When using this footprint research has been done and the results of its application have been reported. A major drawback of PiP for the use of all solder material has been solved by its optimal collection at the soldering point. The aim is to gather enough statistics to use the results of the study to create a model for machine learning.**

***Keywords*—Pin in paste, soldering, combined footprint, machine learning, statistics.**

I. Introduction

Unlike standard surface mounting there are cases where the solder paste is applied outside the contact area for soldering. A typical example is the printing technology using soldering in PiP through holes [1].

Here, an additional amount of paste is applied to provide the required amount of solder filling the hole [2]. The use of a thicker layer is not always appropriate due to the thickness limit of the metal mask. An option is to use double printing or forced application. Another possibility is to apply the paste on an area outside the contact area and then suck it into the hole [3, 4].

The standard topology of a footprint with an increased area can be with an evenly increased area relative to the contact pad - fig. 1.

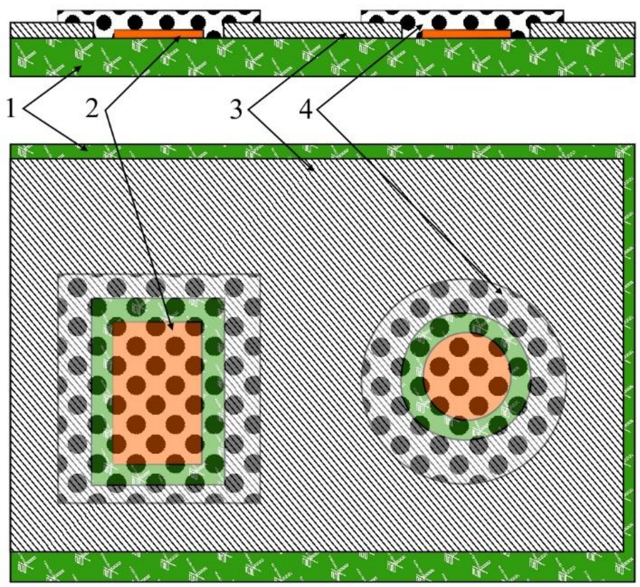


Figure 1. Overprinting footprint: 1- PCB, 2 - pad, 3 - solder mask, 4 – solder paste

In this way, the larger the area of the footprint, the greater the amount of solder material that is required for soldering the thru hole elements. This is possible because there is free space between the contact pads. Thus, electronic assembly of small and large elements can be performed. Small elements require imprints of small thickness, which is achieved with stencils of small height. Large elements are soldered with a large amount of solder material, which is possible thanks to the use of overprinting.

The second approach is asymmetric placement of the imprint in order to optimally use the free surface of the printed circuit board in accordance with the other contact pads - fig. 2.

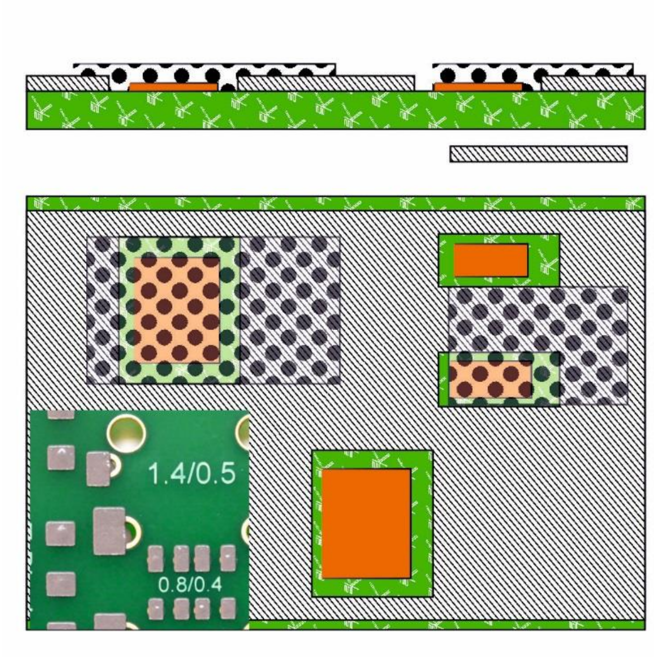


Figure 2. Overprinting footprint for PiP

II. Experiment

In the case of overprinting, a different shape of the footprint can be used depending on the topology - fig. 3.

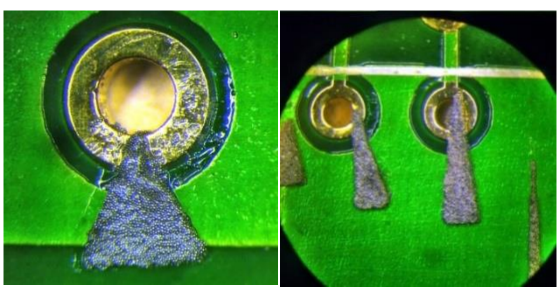


Figure 3. Non-standard overprinting

For the implementation of such footprints were designed, manufactured and used special stencils - fig. 4.

Figure 4. Stencils for nonstandard overprinting

Of particular interest is the process of melting and absorbing the paste. This process is timeless and differs from the melting process at a standard contact pad. Experiments have been conducted on the behavior of different footprint topologies in the fusion and drip mode.

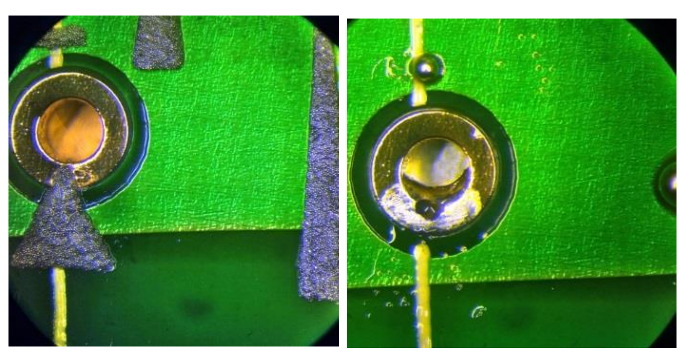
In the standard case we have melting and direct wetting by heat transfer through the contact area, the component outlet and heat transfer through the surface of the paste. In the cases under consideration, the process has two areas: a standard case area and a second area - heat supply through a printed circuit board and through the surface of the paste, which has a different heat transfer coefficient. The second difference is that the process of wetting consists in melting the paste - suction to the melt in the area of the solder - wetting. This leads to an increase in the time for complete wetting of the outlet and must be taken into account in the technological process.

Modeling the melting and suction process is possible but extremely complex. If you go down to the level of melting a single grain of solder, merging the individual grains, examining the shape of the melt and subsequent suction, it would be necessary to use a powerful calculation process, which is not justified.

A better option is to study the behavior of the process using basic forms of footprints and their subsequent application in machine learning.

The task can be divided into two:

1. Nature of melt formation in different imprint shapes as shown in Fig. 5.



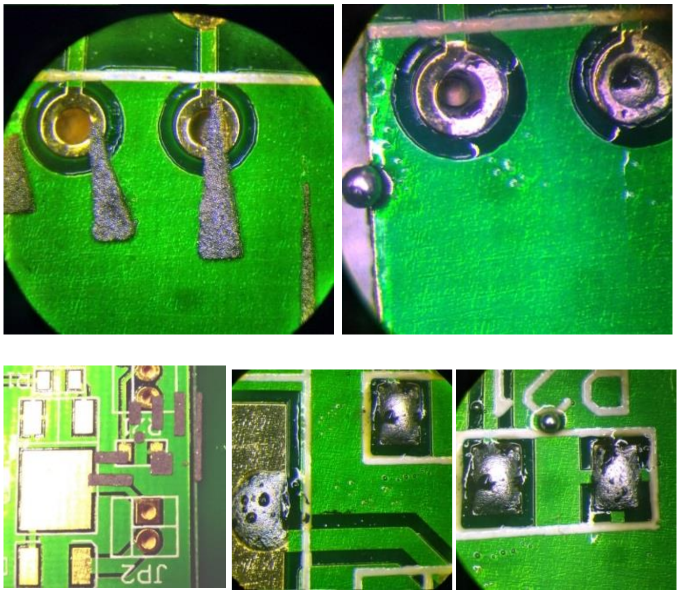
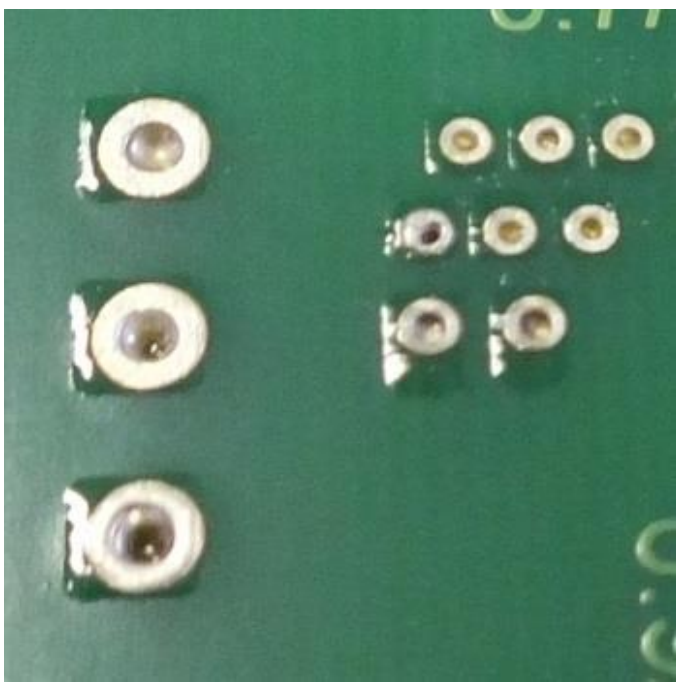


Figure 5. Solder with combined footprint and overprinting

2. Nature of suction at different contact of the imprint with the contact pad, as shown in fig. 6.



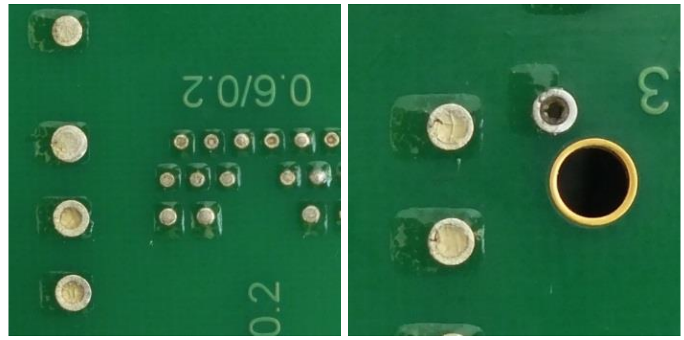


Figure 7. PCB – soldering with combined footprint and overprinting

For this purpose, conductive melting of various prints in shape and overlapping with a contact pad was performed. Conductive soldering was used to avoid the effect of fluid flow on droplet formation.

The temperature profile is controlled only until the moment of formation of the drop - wetting. Pastes with bismuth content are used for lower melting temperature.

The temperature profile used is shown in fig. 8.

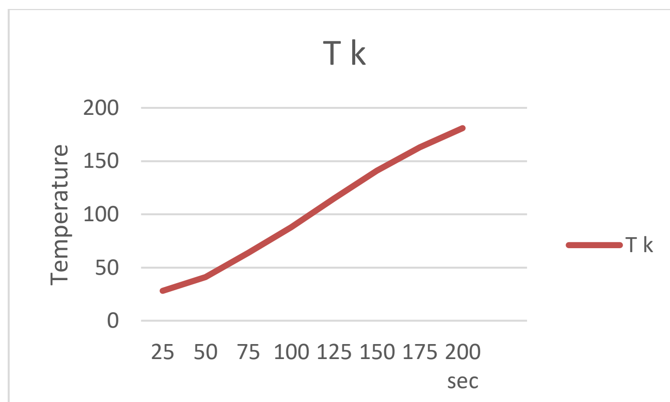


Figure 8. Temperature profile

III. Results

One of the obtained results is shown in fig. 9.

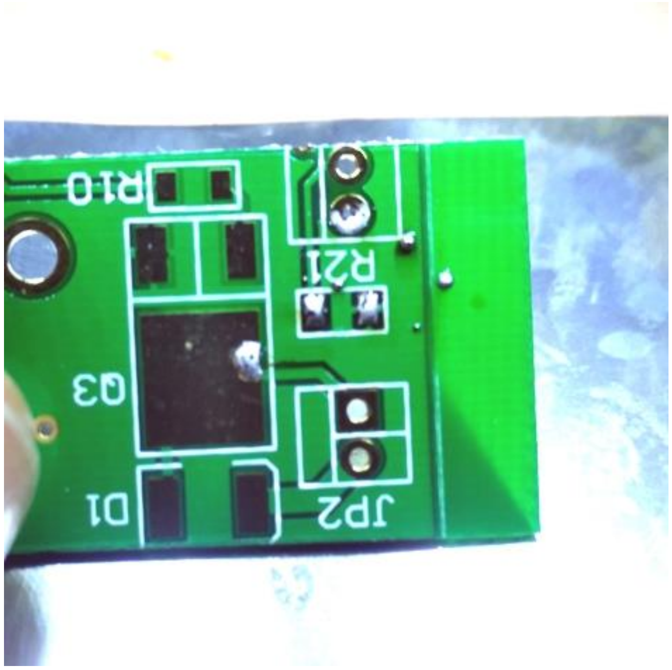


Figure 9. Collecting of soldering material

The obtained result shows excellent collection of the soldering material in different cases of combined footprint - only on solder mask, only on pad, on pad and solder mask.

The condition for good collection of the soldering material is that the combined footprint is well designed and dimensioned.

An even better result of collecting the solder material is obtained by making a combined footprint with two different pastes, as shown in fig. 10.

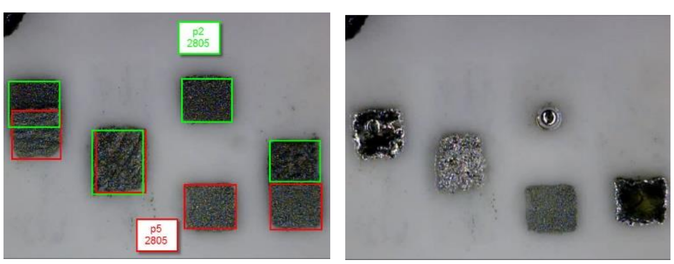


Figure 10. Combined footprint with two soldering pastes

Thanks to the ineffective soldering of the soldering materials (standard and low temperature) excellent collection of the melt in the place of the footprint is obtained.

IV. Conclusions

The obtained results are the basis for a more in-depth study of soldering using a combined footprint.

By collecting sufficient statistics on the results of soldering with different combined footprints, a machine learning model can be created. Thus, current standard printer machines can be easily reprogrammed to produce a combined footprint.

Soldering using a combined fingerprint will allow for:

- Stable and reusable PiP soldering
- No contamination with solder balls, which reduces the reliability of electronic products
- Avoid cleaning the electronic modules after soldering
- Creating strong solders
- Improved thermal conductivity of solders
- Low solder resistance
- Low cost due to the short production cycle

Acknowledgement

This research is supported by Bulgarian National Science Fund in the scope of the project "Exploration the application of statistics and machine learning in electronics" under contract number КП-06-Н42/1.

References

[1] Bob Willis, "PIHR technology", 2012.

[2] Oliver Krammer, New Method for Calculating the Necessary Amount of Solder Paste for Pin-in-paste Technology, 19th International Symposium for Design and Technology in Electronic Packaging (SIITME), 2013.

[3] Valentin Tsenev; Valentin Videkov; Aleksei Stratev; Mladen Mitov; Ana Bankova; Vasil Atanasov Influence of geometry on the applied solder paste in technology Pin in Paste(PiP) 2016 39th International Spring Seminar on Electronics Technology (ISSE) Year: 2016 Pages: 160 - 162, DOI: 10.1109/ISSE.2016.7563180.

[4] Valentin Tsenev ; Valentin Videkov, Experimental study of the melting of a solder paste, 2018 IEEE XXVII International Scientific Conference Electronics – ET, 13-15 Sept. 2018, DOI: 1109/ET.2018.8549606,

[5] R. B. Lima, V. F. Cardosoa and A. N. R. da Silva, Analysis of Pin in Paste Soldering on Printed Circuits Boards Assembly Using Lead Free Solder Paste, ECS Transactions, 39 (1) 417-422, 2011, DOI: 10.1149/1.3615221.

[6] Tony Lentz, Jasbir Bath, N. Pavithiran, How Does Printed Solder Paste Volume Affect Solder Joint Realiability, SMTAI, 2018.