



www.singhamarpreet.com



SMT – Solder Paste Printing Process

Zenaca Consulting

Amarpreet Singh

Revision 0

Dec 7, 2023

Agenda

- Purpose
- Introduction to solder paste printing process
- Stencil design
- Stencil manufacturing methods
- Solder printing process
- Stencil cleaning and Inspection criteria
- Misprint board handling
- Solder Paste Inspection

Purpose

“ The purpose of this document is to provide a comprehensive overview of the solder paste printing and inspection processes in the context of printed circuit board assembly (PCBA).

It covers key aspects such as the importance of stencil design, various methods of stencil manufacturing, and the role of solder paste in accurate deposition on PCB pads.

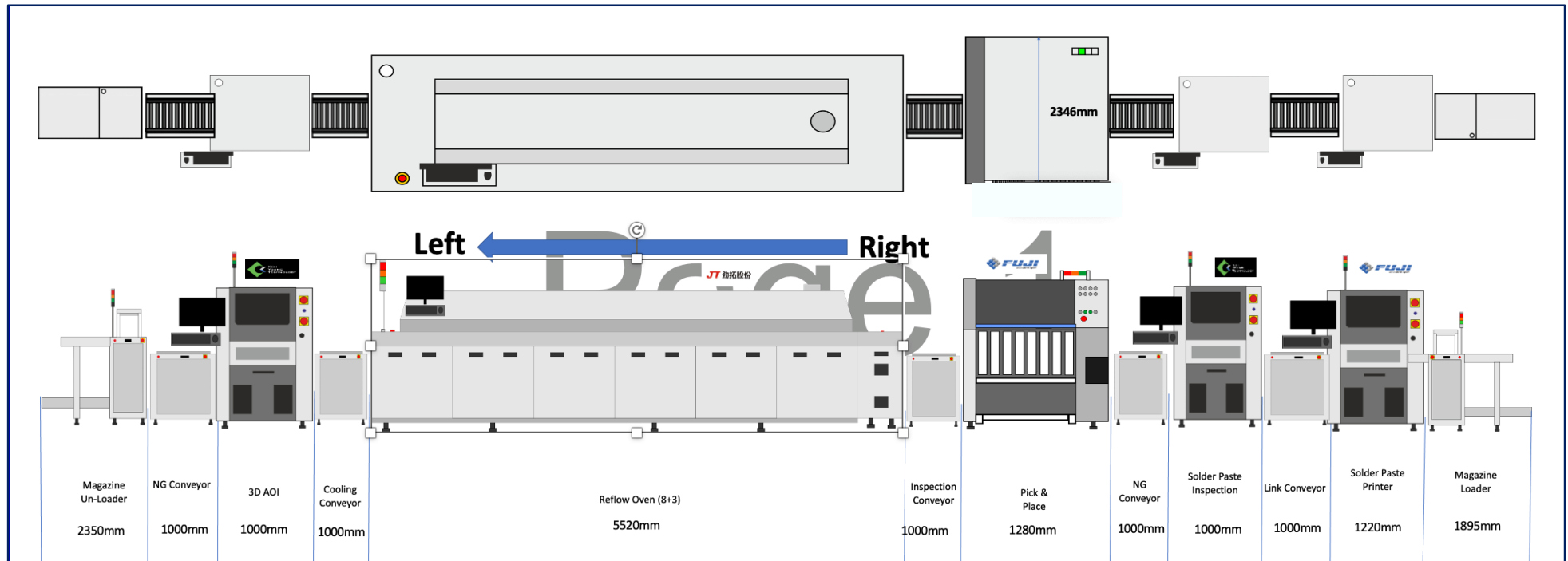
The goal is to equip readers with a thorough understanding of the critical factors and procedures involved in achieving optimal solder paste application and quality control in PCBA.”

SMT Line Details



- All the equipment's are connected using conveyors and there is no human involvement for board movement on the line.
- Considering the number of line items to be mounted, there might be more than one mounting machine (Pick n place) on the line.
- The line can be configured Right to left or left to right.

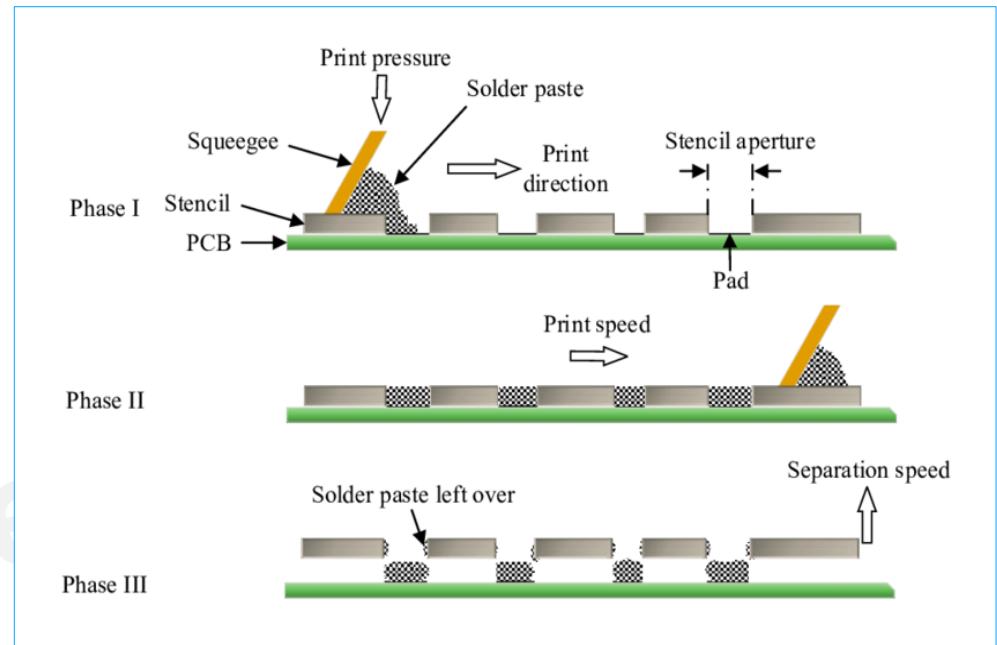
SMT Equipment Details



- A reference line with Fuji Pick & place machine.

Solder Paste - Introduction

- The application of solder paste to the printed circuit board (PCB) is a crucial step in the surface mount assembly process.
- The objective of this step is to precisely deposit the appropriate amount of solder paste onto each of the soldering pads.
- This is accomplished by screen-printing the solder paste through a stencil or foil using a fully automatic printer.

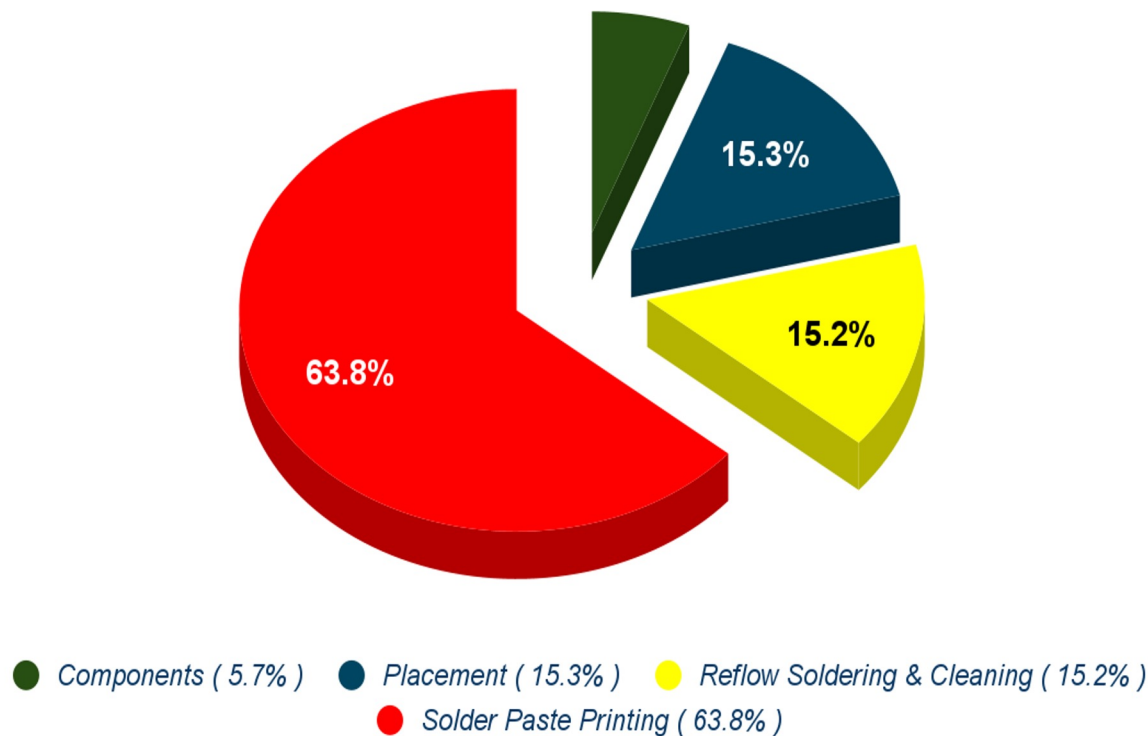


Key Factors

- Stencil design
- Stencil cleaning process
- Stencil Identification & storage recommendation
- Stencil maintenance and life cycle management
- Misprint board cleaning process
- Solder paste storage and handling
- Solder paste inspection process

SMT Process Defect Contribution

SMT Process Defect Contribution



- Solder paste printing is the most critical process in an SMT line, contributing to 64% of the total defects.
- A lot of focus, expertise and process controls are required to execute a flawless solder paste printing process.

Source – Defect distribution data from an EMS Industry

Factors Influencing

Stencil

- Materials
- Manufacturing techniques
- Dimensional accuracy
- Surface and wall finish
- Area aspect ratio
- Solder paste transfer accuracy

Equipment

- Print head
- Squeegee materials
- Printing table
- PCB Support
- Vision system
- Accuracy
- Printing repeatability

Materials

- Solder paste
- Alloy composition
- Alloy particle size
- PCB Flatness
- PCB Surface finish
- Solder flatness

Parameters

- Squeegee pressure
- Squeegee speed
- Stencil Separation speed
- Stencil cleaning frequency

Environment

- Production area cleanliness
- Air humidity
- Temperature

- Operators : Training, Experience and knowledge, Passion and authority

Stencil Design

- Designing the stencil is a crucial step in the PCBA process setup, directly impacting product yield, quality, and reliability.
- The stencil must be designed to apply the correct amount of solder paste to the PCBA pads, meeting specified requirements. Additionally, the stencil design should prevent any excess solder paste on the PCB.
- Many companies maintain evolving stencil design guidelines based on continuous learning and experience.

Stencil Design – Key Considerations

Key considerations for a good stencil design

- Aperture Vs Pad size
- Aperture shape
- Aperture design
- Stencil thickness
- Stencil manufacturing method
- Type of solder paste to be used
- Lead free or leaded solder paste

Stencil Design – Aperture Size

It is recommended to have smaller openings for fine pitch components, this is mainly for achieving

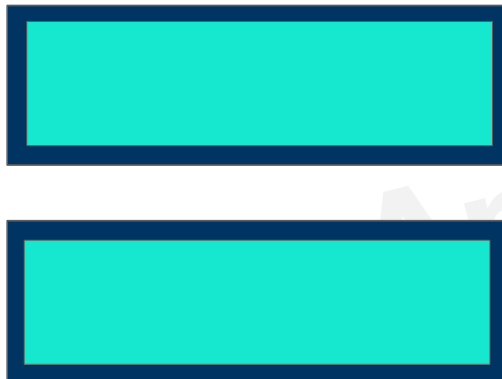
- Improved gasketing between the landing pad and stencil.
- Avoid bridging on the fine pitch components

CBGA will requires bigger apertures compared to the pad design so that the required solder volume can be achieved.

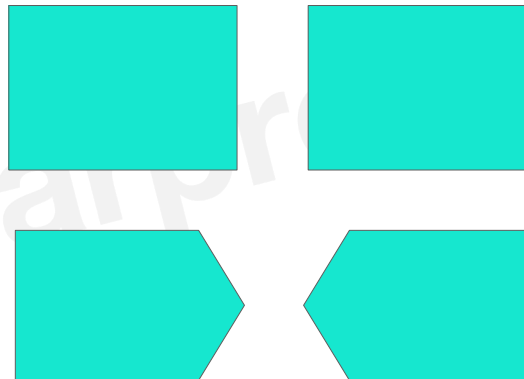
Stencil Design – Aperture Size Vs Pad Size



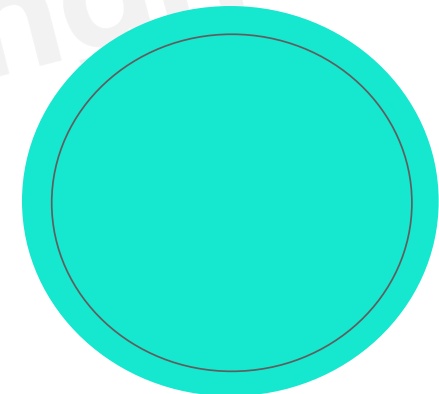
Reduced Aperture Size



1:1 Aperture Size



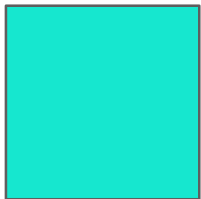
Oversize Apertures



Stencil Design - Aperture Shapes

Different aperture shapes have been found to offer the benefit of optimum solder paste release, reduce solder balls, bridging, solder voids and other soldering related defects.

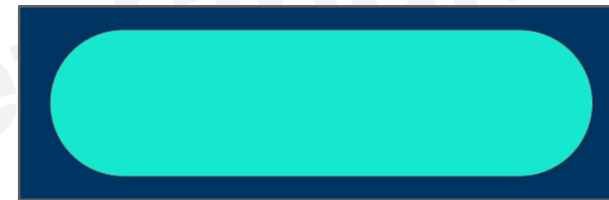
Square



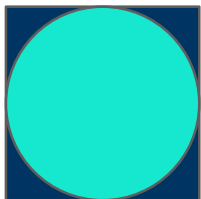
Rectangle



Oblong



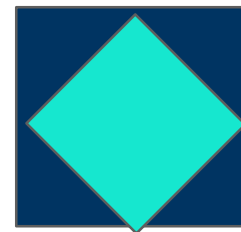
Circle



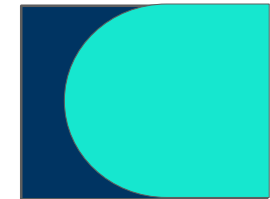
Home Plate



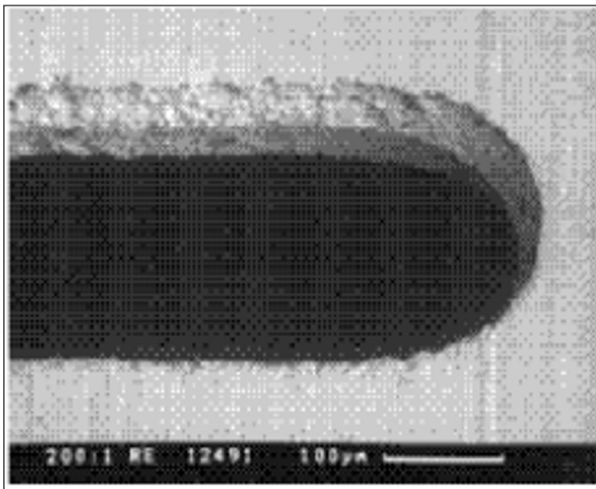
Diamond



D-Shape



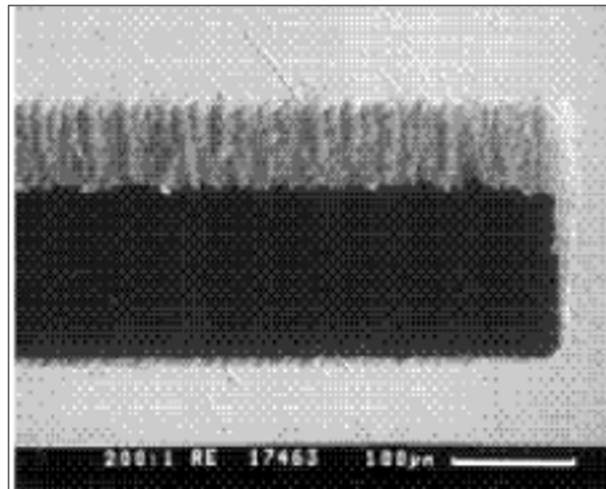
Stencil Manufacturing Method



Process - **Etching**

Materials - CrNi Steel

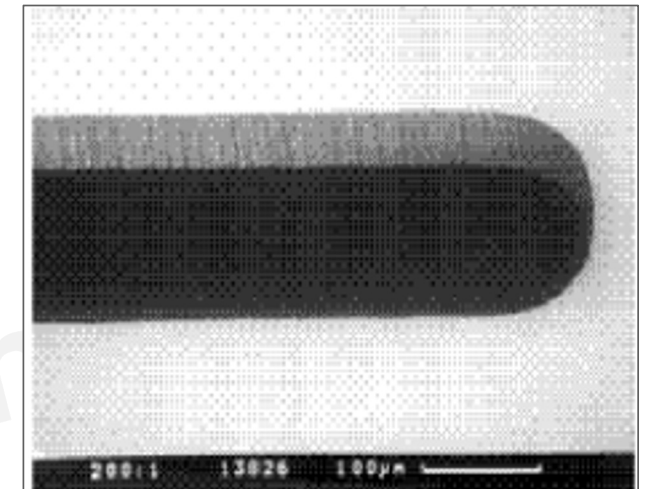
Cost factor - 1.0



Process - **Laser Cut**

Materials - CrNi Steel

Cost factor - 1.2



Process - **Electroformed**

Materials - Nickel

Cost factor - 1.8

Stencil Manufacturing Method

- Step Down Stencils - It does happens sometime that some of the components like 0201 or CSP on the PCBA needs less solder compared to the generic population of the components.
 - Step Up Stencils - Some of the components like CBGA or PTH PIP (Paste in Pin) components needs more solder compared to the other components. It is required to have step up stencils for these components.
-

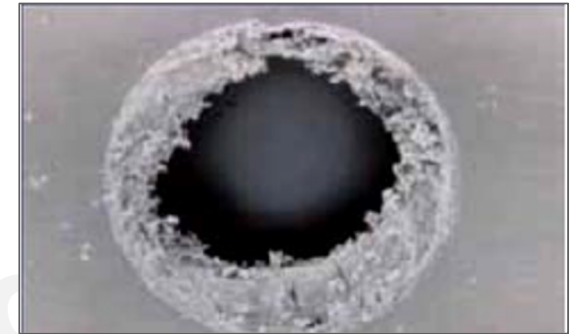
Solder Printing Process

Following are some of the variables those affect quality of the printing process and we must needs to look into these while setting up printing process

- Solder paste viscosity
 - Squeegee health, cleanliness, pressure and hardness.
 - Print speed
 - Separation speed
 - Height and volume of the solder paste
-

Stencil Cleaning

- All the printers these days have in build cleaning functionality, where the printer will clean the stencil after 2-3 cycles.
- However it is important to clean the stencils in a stencil cleaning machine at the end of the shift or at the end of the build before putting into storage.
- A uncleaned stencil stored cannot be used again during the next build, as the solder paste deposited on the inner walls will permanently damage it.



Solder Residue Left in Apertures

Stencil Inspection

- All stencils should be inspected under magnifying glass and back light to ensure cleaning of each and every aperture.
- A simple in house setup can be developed with a provision of backlight under the stencil and a movable magnifying glass on the top side.



Stencil Inspection Set-Up

Stencil Tension Verification

- During the usage and cleaning of stencils, it is a good practice to record the stencil surface tension. If the surface tension is not as per the specification or requirements, will cause poor solder paste printing and ultimately defects on SMT line.
- As a thumb rule, it is recommended to replace the stencils after 40K cycles.
- It is also important to check for any damages etc. during the inspection process.



Stencil Tension Verification

Stencil Storage

- It is important to store the stencils in a way that those are not touching each other and there is no pressure on the screen of the stencils during the storage.
- There are racks available in the market, those can be procured for stencil storage.
- Each stencil should also be identified with basic information like product revision, Mfg. date, Thickness, Lead or lead free process etc.
- The stencils are barcoded and most of the stencil printers have set up verification process and stencils are verified in the system before starting the line.



Stencil Storage Racks

Squeegee Storage

Like stencils different Squeegee should also be stored carefully to prevent any damages during the storage.

Any damage to the squeegee will be impact on the quality of solder paste printing.

Every squeegee should carry basic information like Blade Size, Material and Leaded or Lead free. We cannot mix leaded and lead free squeegees.

The squeeze are barcoded and most of the stencil printers have set up verification process and squeeze are verified in the system before starting the line.

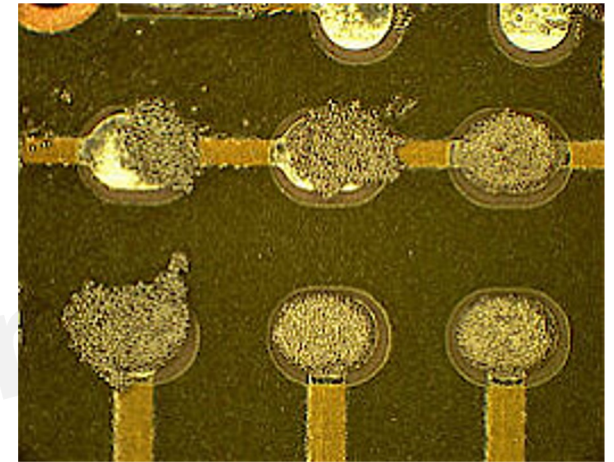


Squeegee Storage Racks

Misprint Board Cleaning

Misprint boards must be cleaned by a machine as manual cleaning will never be enough to remove all the residual solder from the pads of the PCB.

Manual cleaning actually pushes the solder into the Via's, PTH or under the solder mask opening making it even worse.



- ✓ Misprint board cleaning process should be even more carefully done if one side of the PCB is already populated with the components.
- ✓ The subject is even more severe if the components mounted on bottom of the PCBA are MSD or fine pitch components.

Misprint Board Cleaning

The cleaning process must be compatible with the PCB surface finish. It is important for the assembly houses to qualify the cleaning process during the qualification of the products as there will be a requirement to clean the misprint PCBA's during the volume production.

The process will be different for each PCBA, but some of the steps will be common or minimum required while setting up the process. The steps are

- ★ Cleaning process must have a rinsing step.
 - ★ Deionised (DI) water must be used for cleaning the PCBA's.
 - ★ There must be a detailed inspection of the cleaned boards.
-

Misprint Board Cleaning

In case the PCB surface finish is OSP (Organically Soluble Preservatives), some special considerations like below must be taken while setting up the cleaning process for these PCBA's

- ❖ Chemical removal of the solder degrades OSP coating.
 - ❖ Alcohol based solvents dissolve up to 75% of the OSP coating.
 - ❖ Water cleaning is recommended as it dissolves approx. 15% of the OSP coating.
 - ❖ OSP Board those have undergone cleaning process, must be assembled on priority at SMT lines to avoid solderability issues resulted from oxidized copper exposed during the cleaning process.
 - ❖ Minimum handling of the PCB's to avoid any kind of oxidation of the copper pads after cleaning.
-



Solder Paste Storage

- Solder paste needs to be stored in a refrigerator as per solder paste supplier recommendations.
- Before usage solder paste needs to be normalized at room temperature as per the solder paste supplier recommendations.
- We can also use a machine to control the normalization time.



Solder Paste Storage - Good Practices

- Do not rapidly warm up solder paste in the hot sun or by keeping it on top of the reflow oven.
 - Do not try using solder paste directly from the refrigerator.
 - Do not store leaded and lead free solder paste in the same storage location.
 - Do not mix the solder paste removed from the stencil with the fresh solder paste
 - Do not use the solder paste which appears to be separated.
 - Always try to buy solder paste in the minimum container size to reduce the wastages etc.
-

Solder Paste Storage - Good Practices

- After procurement of solder paste, make sure to add FIFO Label having the receipt date of the solder paste.
 - While removing solder paste from the refrigerator record the removal date and time on the solder paste label.
 - Date and time should also be recorded at the time of opening the solder paste on the same FIFO Label.
 - Solder paste should not be left on the stencil for more than recommended stencil life of the solder paste.
 - The MES (Manufacturing Execution System) also can track the various timings of the solder paste.
-



Solder Paste Storage - Good Practices

| | |
|-------------------|--|
| Part # | |
| Description | |
| Alloy Composition | |
| Clean or No Clean | |
| Powder Size | |
| Batch Code | |
| Receipt Date | |
| Shelf Life | |

Solder Paste Tracking Label - 1

| | |
|------------------------------------|--|
| Part # | |
| Description | |
| Time to room temp (Before use) | |
| Refrigerator removal date and time | |
| Open date and time | |
| Stencil life | |
| Can be used till (Date and time) | |

Solder Paste Tracking Label - 2

Solder Paste Inspection

- ❖ Solder paste inspection is required to ensure the required amount of solder deposited on the pads. Volume of the solder paste deposited is a direct representation of the yield of the product.
 - ❖ Solder paste inspection can be done manually or using Solder Paste Inspection machines (SPI). The SPI machines needs to be programmed accurately and fed with a lot of data for predictable results.
-

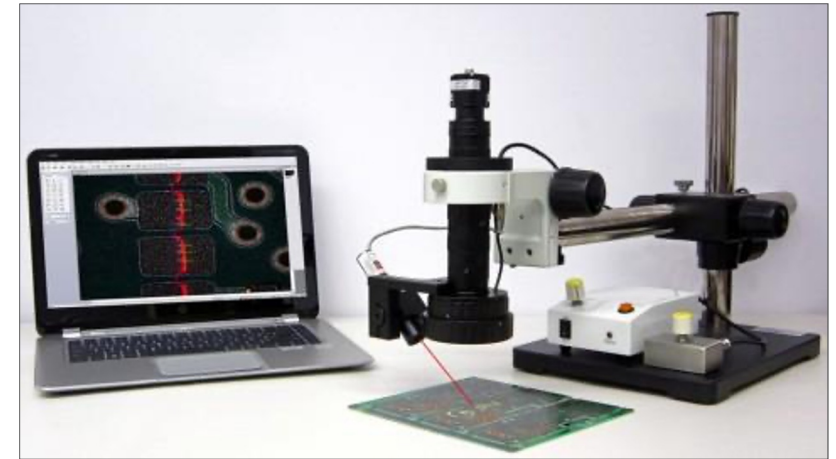
Solder Paste Inspection - Automated

- SPI Machines needs to be loaded with stencil check plots for developing a program, where the required height and volume of each of the pad is defined by the software.
- Each board when inspected through the machine will compare the actual measurements w.r.t. The required measurements for defining the solder paste printing process.
- As it is an automated process, 100% of the components are usually inspected.



Solder Paste Inspection - Manual

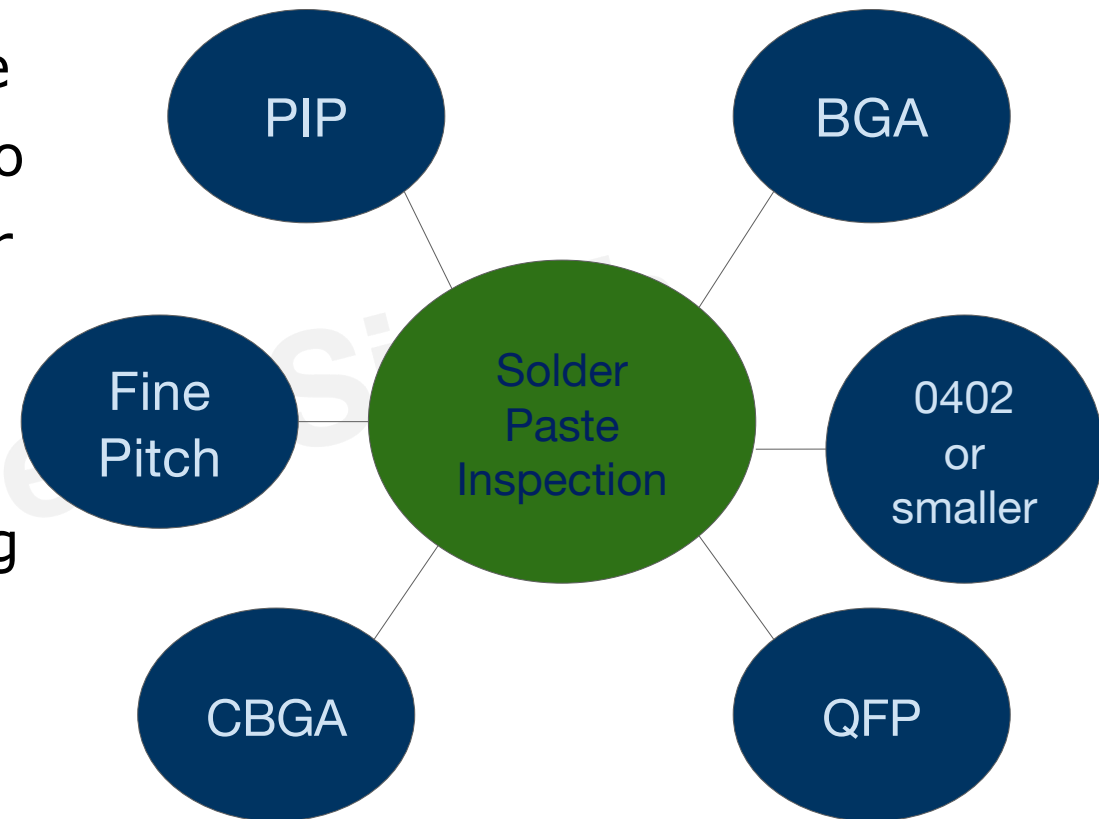
In the absence of fully automated SPI machines, manual inspection and measurements are performed on the PCBA's after printing process.



- ❖ Manual inspection needs to be done very carefully, not to disturb the solder paste during the inspection process.
- ❖ There are some semi automatic scopes are also available in the market to measure the height of the solder paste.

Solder Paste Inspection - Manual

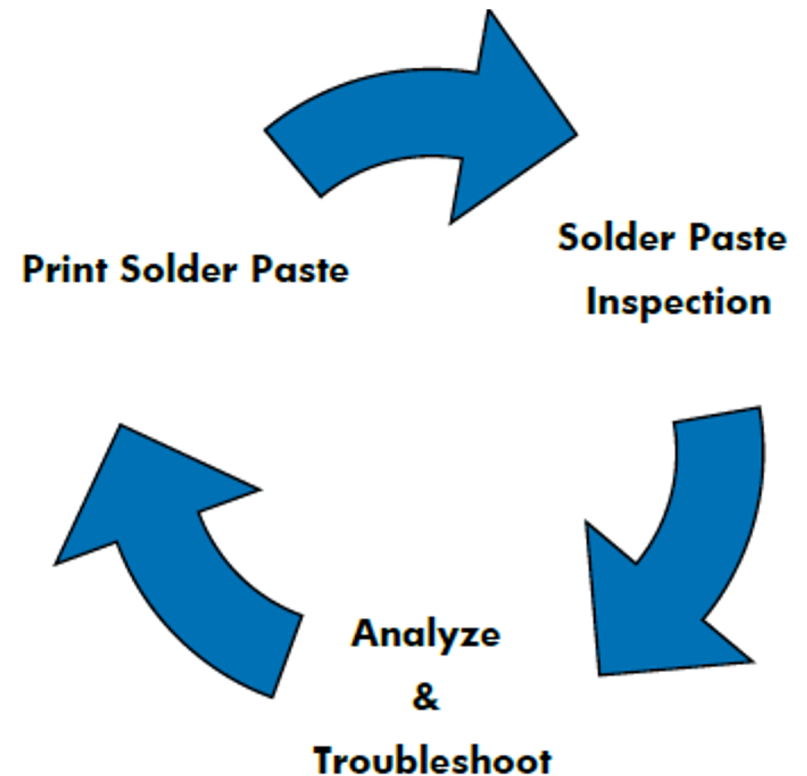
- As the manual inspection is done offline, as a minimum we need to check the critical components for the presence and volume of the solder paste.
- Whether to do 100% or sampling ? It can be decided based on the quality of the PCBA produced from the line.



Solder Paste Inspection - Feedback



- 100% Inspection using SPI machines can help to eliminate most of the soldering defects on the PCBA line.
- The process needs to be analyzed after every printing cycle, followed by the corrective actions out of the results.



Key to Successful Solder Paste Printing Process



Knowledgeable, experienced and passionate process engineers and SMT operators.

Process controls in place and discipline to follow those controls like DfX, Stencil design, Machine set up, Board support, Solder paste Inspection and feedback mechanism, Solder paste management etc.

Regular RCA and monitoring of the processes.

Do you have more questions on topic ?

Please reach me at

Email : amar@singhamarpreet.com

Contact : +91 96866 83783

www.singhamarpreet.com

Thank You

Amarpreet Singh