UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Faculty of Manufacturing Engineering

SOLDER PASTE PRINTING YIELD IMPROVEMENT FOR SMT ULTRA FINE PITCH PRODUCT BY USING SIX SIGMA TECHNIQUE

Lim Kean Teik

Master of Manufacturing Engineering (Manufacturing System Engineering)

2016

SOLDER PASTE PRINTING YIELD IMPROVEMENT FOR SMT ULTRA FINE PITCH PRODUCT BY USING SIX SIGMA TECHNIQUE

LIM KEAN TEIK

A thesis submitted
in fulfillment of the requirements for the degree of Master of Manufacturing Engineering (Manufacturing System Engineering)

Faculty of Manufacturing Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2016

DECLARATION

I declare that this thesis entitled "Solder paste printing yield improvement for SMT ultra fine pitch product by using six sigma technique" is the result of my own research except as cited in the references. This thesis has not yet being accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : Lim Kean Teik

Date : 03-June-2016

APPROVAL

I hereby declare that I have read this thesis and it is sufficient in terms of the scope and quality to gain the award of Master of Manufacturing Engineering (Manufacturing System Engineering).

Signature : ....

DR. MOHAMAD BIN MINHAT
Pensyarah
Fakulti Kejuruteraan Pembuatan
Universiti Teknikal Malaysia Melaka

Supervisor Name : ....

Date : 03-June-2016

DEDICATION

To my beloved mother and father

ABSTRACT

Surface mount technology (SMT) is a method for producing electronic circuits in which the components are mounted directly onto the surface of printed circuit boards (PCBs). It has been emerged in the 1960s, gained momentum in the early 1980s and became widely used by the mid-1990s and is continually becoming more important as PCB assemblies become more advanced and compact with componentry real estate savings provided with SMT become more valuable and necessary. At presents, in the industry it has largely replaced the through-hole technology construction method of fitting components with wire leads into holes in the PCB because it allows for better electrical performance when compared to through-hole via and offers a conventional reflow processes compared to conventional wave solder. The majority of defects in PCB assembly are caused due to problem in the solder paste printing process or due to defects in the solder paste. There are various types of defects such as excessive solder paste, and solder melts and connects too many solder pads, which causes a short circuit. Insufficient solder paste result in incomplete circuits. When making PCBs, manufacturers often test the solder paste deposits using SPI (solder paste inspection). 3D SPI systems measure the volume of the solder paste printed on solder pads before the components are applied and the solder reflow process. SPI systems can reduce the incidence of solder-related defects to statistically insignificant amounts. Yield improvement requires increased focus on stencil technology, printer capability, solder paste functionality and under stencil cleaning. The term of fine pitch is defining as 0.5mm in pitch and ultra fine pitch is 0.2mm – 0.4mm in pitch (Surface mount council, 1999).
The two major responses from the experimentation are printing quality and SPI yield.

ABSTRAK

Surface Mount Technology atau sering disingkat dengan sebutan SMT adalah satu kaedah untuk menghasilkan litar elektronik di mana komponen dipasang terus pada permukaan papan litar bercetak (PCB). Komponen elektronik yang dapat dipasangkan oleh mesin-mesin SMT adalah komponen khusus yang biasanya disebut dengan komponen Surface Mount Device (SMD). Ia telah muncul pada tahun 1960, bertambah pesat pada awal 1980-an dan menjadi applikasi secara meluas pada pertengahan 1990-an dan berterusan menjadi lebih penting kerana pemasangan PCB menjadi lebih maju dan padat dengan simpanan harta componentry sebenar disediakan dengan SMT menjadi lebih berharga dan perlu. Pada masa sekarang, dalam industri ini ia telah banyak menggantikan kaedah pembinaan teknologi melalui lubang komponen sesuai dengan dawai membawa ke dalam lubang pada PCB kerana ia membolehkan untuk prestasi elektrik yang lebih baik jika dibandingkan dengan cara lubang melalui dan menawarkan proses reflow konvensional berbanding solder gelombang konvensional. Majoriti kecacatan dalam pemasangan PCB àdalah disebabkan kerana masalah dalam proses percetakan pes pateri atau disebabkan oleh kecacatan dalam pes pateri. Terdapat pelbagai jenis kecacatan seperti pes pateri yang berlebihan, dan solder cair dan menghubungkan pad pateri terlalu banyak, yang menyebabkan litar pintas. Tidak mencukupi hasil timah dalam litar tidak lengkap. Apabila membuat PCB, pengeluar sering menguji deposit timah menggunakan SPI (pemeriksaan pes pateri). Sistem 3D SPI menyukat isipadu pes pateri yang dicetak pada pad pateri sebelum komponen yang digunakan dan proses reflow pateri. Sistem SPI boleh mengurangkan insiden kecacatan pateri yang berkaitan kepada jumlah statistik tidak penting. Peningkatan hasil memerlukan peningkatan fokus kepada teknologi stensil, keupayaan pencetak, fungsi pes pateri dan di bawah pembersihan stensil. Istilah padang halus mentakrifkan sebagai 0.5mm di padang dan padang halus ultra 0.2mm - 0.4mm di padang.

Kedua-dua jawapan utama dari uji kaji adalah kualiti pencetak dan hasil SPI.

ACKNOWLEDGEMENTS

I would firstly express my sincere acknowledgement to my supervisor, Dr. Mohamad Bin Minhat from Faculty of Manufacturing Engineering, Universiti Teknikal Malaysia Melaka (UTeM) for his essential supervision, support, and encouragement towards the completion of this thesis. Dr. Mohamad has been supervising me and providing a lot of advice and suggestions for me in completing this project.

I would like to thank all my peers, my mother, father, and siblings for their moral support in finishing this degree. Last but not the least; I would like to thank anyone who had played crucial parts in the realization of this project.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

FIGURE	TITLE	PAGE

FIGURE	TITLE	PAGE

LIST OF ABBREVIATIONS

2D	Two-dimensional
3D	Three-dimensional
7QC Tools	7 principal of quality control tools
Ag3.5	Silver 3.5%
ANOVA	Analysis of variance
BGA	Ball Grid Array
CAD	Computer-aided design
DI water	Deionized Water
DMAIC	Define, Measure, Analyze, Improve and Control
DOE	Design Of Experiment
Dppm	Defect points per million
GR&R	Gage repeatability and reproducibility
IC	Integrated Circuit
IPC	Institute for Printed Circuits
KPIV	Key factor input values
KPOV	Key factor output value
MSA	Measurement system analysis
PCB	Printed Circuit Board
QFP	Quad Flat Package
RPN	Risk priority numbers
SAC305	Tin 96.5%, Silver 3.0% and Copper 0.5%

SMD	Surface Mount Devices
SMT	Surface Mount Technology
Sn96.5	Tin 96.5%
SPI	Solder Paste Inspection

CHAPTER 1

INTRODUCTION

1.1 Overview

At present, with efforts towards tools that held smaller, lighter and more functional; any small pieces of PCB use, creating a high density board with mixture of the components. These components can be composed of smaller components such as 01005 passives to fine pitch QFP / BGA device to a larger power supply devices. Number of solder alloy required to produce a reliable combined with lead-free processing and high processing needs to be a daunting challenge for process engineers. As indicated by the fact that industry reports about 70% of fine pitch Surface Mount Technology defects are related to the solders paste stencil printing process (IPC-7526, 2007).

Solder paste printing operation is widely recognized as a major source of defects in surface mount assembly. One approach to improving outcomes associated with solder paste deposition process is to detect print defects immediately after print operation and the reject defective board before component placement. This allows manufacturers to save time of wasted assembly defective board and avoid costly rework. Whether or not the reported defects in any particular site, SPC data can be collected and used for monitoring and correcting undesirable trends before they become critical to the process. 3D SPI system can be a tool to provide instant detection (Paul Haugen and Rita Mohanty, 2008).

The stencil itself is becoming increasingly critical as component size and density of placement increased. Main stencil factor could be material, fabrication methods, layout of the aperture, thickness, aperture geometry, aspect ratio, the size of the opening, the area ratio, taper and polish. Electroformed polished stencil with smooth aperture walls increases the paste release. The recent innovation of stencil coating to improve repellence of solders paste from the stencil wall is gaining significant interest.

In order to secure a sufficient volume of solder paste, it is desirable to print solder paste by considering the process parameters need to be established for the assembly concerned. Stroke length, print pressure, print speed and print release from the stencil that is usually dialed in using automated inspection operations. Cleaning frequency is established based on the board design and manufacturing environment or based on how well the board, stencil and paste are interacting within the print process. Printing consistency is crucial to achieve printing repeatability and reproducibility.

1.2 Problem Statement

The solder paste printing is known to be one of the most critical properties in Surface Mount Technology (SMT) to assure quality in the manufacturing of electronics. The challenge increases as the technology development moves toward a smaller chip components and finer pitch size of the solder pad on smaller and densely populated printed circuit boards. This study investigated the effects of printing critical factor that affected on solder print efficiency, defined as the volume ratio of printed solder to the solder pad size. Printed solder paste volume is measured by solder paste inspection (SPI). The test printed circuit board (PCB) in this study possessed 01005 and fine pitch package, and printability test pads printed using type 6 solder paste (Sn 96.5%, Ag 3.0%, Cu 0.5%, by weight). Various studies conducted to understand the behavior of solder paste stencil printing process. Solder paste has a range of factors that influence such as printing machine parameter, environment condition, stencil design and material, solder paste type and so on.

Thus intensive research is needed to understand the factors that most influence and how to overcome them. There is no any standard method for stencil printing process simulation using virtual flow. As such, this article can be the one to help researchers to simulate the solder paste printability in the future.

1.3 Objectives

Regarding to the problems stated in the section 1.2, a few studies have been done to improve the solder paste printability to get better yield. The following objectives have been set to overcome the issues.

a) To investigate the printing parameter setting such as printing speed, snap-off, separation speed and printing force and investigate its effectiveness in the printing process.
b) To investigate the performance of two types of stencil, which are stainless steel laser cut and electro-form stencil, in stencil printing.
c) To evaluate the Nimfron squeegee blade will be use to replace the conventional stainless squeegee blade.

1.4 Outline of Thesis

The organization of this thesis is dividing into three main chapters. First chapter presents the background of the project and declares the objectives of the study. This chapter is concluding with the outline of the thesis.

Chapter Two reviews the basic and technology of solder paste printing operation concept. It also investigates the critical parameter that influence the solder printing process which involving of printing machine, type of stencil, solder paste, and the squeegee setting, squeegee blade type and important of solder paste transfer efficiency. Furthermore, chapter two reviews the applications of 3-Dimensional Solder Paste Inspection (3D SPI) machine use to inspect the solder volume.

In Chapter Three, the methodology to improve the solder paste printing yield is presented by using Six Sigma Technique. It describes the study and development of printing parameter, stencil type and squeegee blade.

CHAPTER 2

LITERATURE REVIEW

This chapter describes the basic operation of a solder paste printing process. The typical important printing parameter that influence the solder paste printing performance has been identify and Electroform stencil and Nimfron squeegee blade is investigated. It also discusses the important of solder paste selection and transfer efficiency that influence the solder paste print onto the Printed Circuit Board (PCB). The 3-Dimensional Solder Paste Inspection (3D SPI) machine use to inspect the solder volume.

2.1 Solder paste printing process overview

Circuit board manufacturing operations involve a stencil printer used to print solder paste onto a circuit board. A circuit board having a pattern of pads or other conductive surfaces onto which solder paste will be deposited is delivered into the stencil printer and one or more small holes or marks on the circuit board, called fiducial marks, is used to align the circuit board with a stencil or screen of the printer prior to the printing of solder paste onto the circuit board. After the circuit board is aligned, the board is raised to the stencil or in some configurations the stencil is lowered to the circuit board. Solder paste is dispensed onto the stencil, and a wiper blade/squeegee traverses the stencil to force the solder paste through apertures formed in the stencil and onto the board.

In some art stencil printers, a dispensing head delivers solder paste between first and second wiper blades, wherein during a print stroke one of the wiper blades is used to move or roll solder paste across the stencil. The first and second wiper blades are used on alternating boards to continually pass the roll of solder paste over the apertures of a stencil to print each successive circuit board. The wiper blades are typically at an angle with respect to the stencil to apply downward pressure on the solder paste to force the solder paste through the apertures of the stencil. While in some other art stencil printers, the dispensing head is pressurized to force solder paste through the apertures, and the wiper blades are employed to scrape excess solder paste from the stencil during a print stroke.

Printer must be able to carry out several other operations to complete the print cycle, which comprises; loading with transport and mounting – Alignment (stencil to board) – The printing process (Squeegee) – Cleaning of the underside of the stencil (after some cycles) – unloading

The sequence of operations is as follows:

i. Board moves to board stop (Conveyor system)

ii. Board is clamped with board support (Snugger System)

iii. Camera moves to fiducial 1/fiducial 2 and locates its position for board and stencil

iv. Camera moves to rest position

v. Board or stencil is aligned

vi. Board moves up to stencil height with zero snap off

vii. Squeegee operation commences with it pre-set Squeegee Pressure (Printing)

viii. Board support moves down

ix. Board is unclamped and moves out of machine.

The solder paste printing process illustration as shown in Figure 2.1 which begin with the stencil close to the board, squeegee move from left to right to make the solder paste rolling and at the same time allow the solder paste filling up the aperture. The solder paste printing process finally will complete with board off from the stencil and make the aperture emptying to allow the solder paste completely transfers from the aperture.

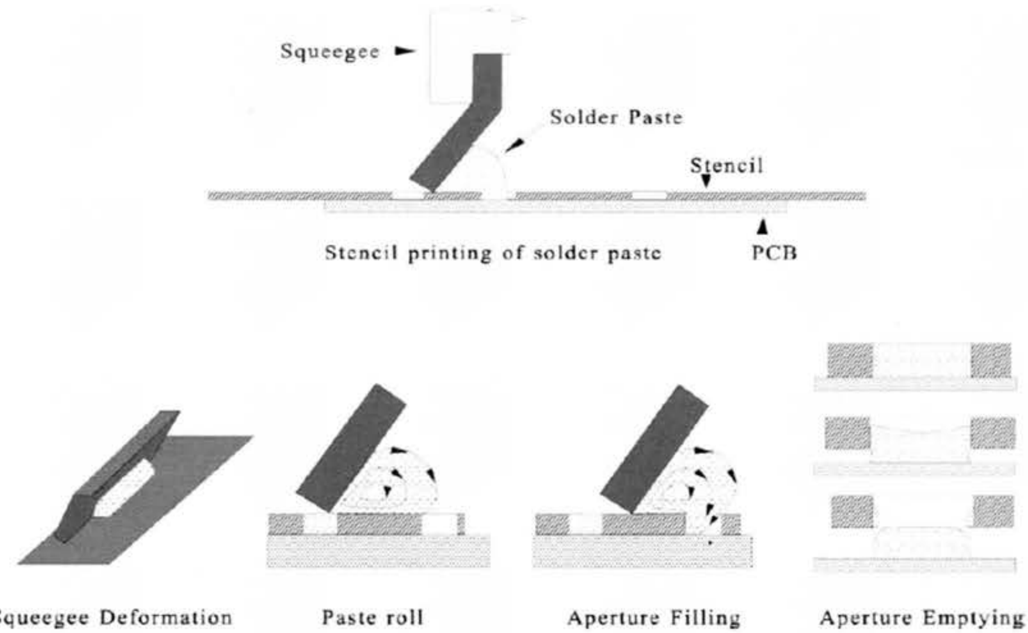

Figure 2.1: Solder Paste Printing Process (R. Durairaj [a],*, S. Ramesh [a], S. Mallik [b,1], A. Seman [b], N. Ekere [b], 2009)

2.2 Printing machine

As the electronics industry develops, there is a growing demand to increase the package density of components on a substrate area. Along with this, performance and quality requirements for solder pastes have become more demanding. With regard to fine pitch printing, what type of printing equipment and parameters to apply are one of the most crucial factors as well as the performance of the solder paste in order to achieve a high level of printability. This means that even if you have chosen the potentially high performance paste, it is very likely that you will end up with poor printing results because of inadequate parameter