INFLUENCE OF MATERIAL PROPERTIES ON THE APERTURE FILLING DURING STENCIL PRINTING PROCESS

By:

MUHAMMAD FARID BIN MOHAMAD RAFDZI

(Matric no.: 125031)

Supervisor:

Dr. Mohd Sharizal Bin Abdul Aziz

June 2018

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfilment of the requirement to graduate with honors degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This thesis is the result of my own investigation, except where otherwise stated and has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any other degree.

(Muhammad Farid B Mohamad Rafdzi)
(125031)
Date:

(Dr Mohd Sharizal B Abdul Aziz)
Date:

ACKNOWLEDGEMENTS

The highest gratitude goes to the God for giving me a good health and wellbeing along the venture of final year project completion. It was a tough journey and required a lot of sacrifices but by the wisdom from Him, I managed to withstand through it successfully.

I also would like to express my deepest appreciation to my final year project supervisor, Dr. Mohd Sharizal Bin Abdul Aziz, for giving me the opportunity and encouragement to conduct project under his supervision. Persistent help and guidance from him have given me a clear vision and solution on the completion of the project, thus, made it possible for me to achieve as best as I could in the project.

In addition, my sincere thank you to En, Mohd Syakirin Universiti Sains Malaysia (USM) postgraduate students for assisting me through the project completion. Lots of knowledge was gained by the sharing of expertise regarding to the project in the solution findings to tackle the occurred problems.

I would like to thank all the staff in the Celestica Malaysia Sdn Bhd especially Mr. Siva for their time and technical support during the experiment by using their facilities to run the research.

Lastly but never be the least, my best appreciation to my parents and families for all the uncountable support and prayers, their patience, understandings and for providing the perfect environment for me to focus on my study.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATION

Nomenclatures	Descriptions
PCB	Printed Circuit Board
SMT	Surface Mount Technology
SMD	Surface Mount Device
SMA	Surface Mount Assembly
SPI	Solder Paste Inspection
SAC105	$Sn_{98.5\%}Ag_{1.0\%}Cu_{0.5\%}$
SAC305	$Sn_{96.5\%}Ag_{3.0\%}Cu_{0.5\%}$
SAC307	$Sn_{96.3\%}Ag_{3.0\%}Cu_{0.7\%}$
SAC405	$Sn_{95.5\%}Ag_{4.0\%}Cu_{0.5\%}$
Sn100C	$Sn_{99.3\%}Cu_{0.7\%+Ni+Ge}$
SnPb	$Sn_{63.0\%}Pb_{37.0\%}$
RSM	Response Surface Methodology
DPPMO	Defect-Per-Million Opportunities
ORP	Optimal Reflow Profile
SDV	Solder Deposit Volume
IMC	Intermetallic Compound
SOT	Small Outlet Transistor
MELF	Metal Electrode Leadless Face
FSI	Fluid Structure Interaction
FVM	Finite Volume Method
FEM	Finite Element Method

PENGARUH SIFAT-SIFAT BAHAN TERHADAP PENGISIAN BUKAAN PADA PROSES PERCETAKAN STENCIL

ABSTRAK

Peningkatan keperluan terhadap Papan Litar Bercetak (PCB) yang lebih kecil, lebih ringan, dan berprestasi tinggi dalam pempakejan elektronik telah menyebabkan penggunaan percetakan stensil secara meluas dalam proses pematerian. Percetakan stensil menawarkan prestasi proses pematerian yang baik dan konsisten serta menghasilkan ouput yang lebih besar pada masa yang singkat sehingga menjadikannya sebagai salah satu pilihan yang terbaik untuk pengeluaran berskala besar dalam Teknologi Pelekapan Permukaan (SMT). Namun, kaedah pematerian ini turut menjadi penyumbang utama kepada kecacatan pematerian berbanding dengan kaedah lain dan perlu ditangani melalui penyelidikan dan pembangunan. Salah satu daripada kecacatan biasa yang terjadi dalam proses pematerian melalui kaedah ini ialah berkaitan dengan kualiti percetakan pes pateri di atas substrat seperti Papan Litar Bercetak disebabkan perubahan parameter proses. Parameter proses yang tidak terkawal mengakibatkan kecacatan pematerian seperti penyambungan pateri yang boleh membawa kepada kegagalan produk dalam proses seterusnya di barisan pengeluaran.

Eksperimen telah dijalankan untuk menganalisis pengaruh sifat bahan pada pengisian bukaan semasa proses percetakan pateri. Parameter kajian ialah kelajuan percetakan, beban percetakan dan arah percetakan. Kajian itu mendapati bahawa kelajuan percetakan 35mm/s hingga 45mm/s berpotensi untuk mendapatkan kualiti cetak yang baik untuk mengisi pateri. Beban cetakan 8kg hingga 9kg menunjukkan pilihan beban percetakan yang baik untuk mencapai kualiti cetak yang baik dalam proses percetakan stensil. Akhir sekali, cetakan dari arah belakang menunjukkan kualiti yang lebih baik berbanding arah hadapan mod percetakan.

INFLUENCE OF MATERIAL PROPERTIES ON THE APERTURE FILLING DURING STENCIL PRINTING PROCESS

ABSTRACT

High requirement of smaller size, lighter weight, and high performance Printed Circuit Board (PCB) in electronic packaging has contributed to the wide application of stencil printing for soldering process. Stencil printing offers good consistency of soldering performance as well as produce larger process output at short time that make it one of the best option for high volume application in Surface Mount Technology (SMT). However, the soldering method also contributes to major percentage of soldering defect compared to other methods which is necessary to be addressed through research and development. One of the common defects due to the stencil printing is regarding to the printing quality of solder paste on substrate such as PCB with respect to the variation of process parameters. Uncontrolled process parameters cause the soldering defects such as solder bridging that can lead to product failure in further processes in production line.

An experiment has been conducted to analyse the influence of material properties on the aperture filling during solder printing process. The studied parameters are the printing speed, printing load and the printing direction. The study had found that the printing speed 35mm/s to 45mm/s has potential to obtain good print quality of solder filling. Printing load of 8kg to 9kg shows the good printing load option to achieve good print quality in stencil printing process. Finally, the reverse direction of printing mode shows a better quality compared to forward direction of printing mode.

CHAPTER 1

INTRODUCTION

1.0 Introduction

Surface-mount technology (SMT) is a technique for producing electronic circuits where the components are mounted on the surface of printed circuit boards (PCBs). A tiny electronic device such as capacitor and inductor that had been mounted in SMT is called a surface-mount device (SMD). The process in SMT line start with solder printing with is the PCB is deposited by solder paste before the next step which is component placement. Then the component will be inspected before it go through reflow oven for reflow solder phase. Then, the PCB will go through cleaning process to clean the excess flux on the PCB. As it finish, the solder joint will be inspected before it go through circuit test phase and end with packaging. Figure 1.1 shows the typical electronics manufacturing printed circuit board (PCB) in SMT line.

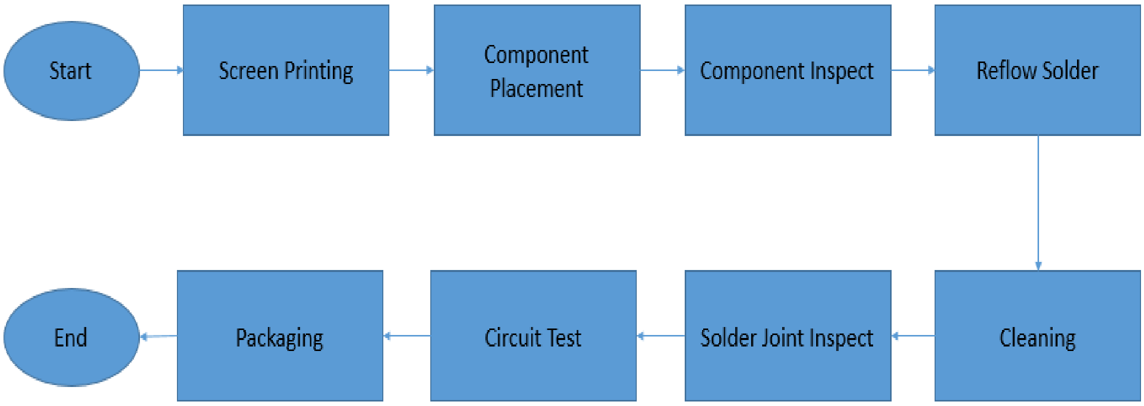


Figure 1.1: Typical electronics manufacturing printed circuit board (PCB) in SMT line

On the component placement phase, pick-and-place robotic machines are used to place surface-mount devices (SMDs) onto a printed circuit board (PCB). They are commonly used robotic machine for high speed, high precision placing of broad range of electronic components like capacitors, resistors and inductors onto the PCBs which are in turn used as an controller in the electronic device.

There are some advantages by using SMT on producing electronic circuit such as can speed up production process[1]. Although the production of the device will be faster, there are still have some limitation in term of defect on the electronic circuit. There might be some risk on the failure detection in the electronic circuit that have to overcome for any SMT manufacturing process.

Stencil printing is one of the process in SMT where it is depositing solder paste on the aperture filling to provide an electrical connections on the printed circuit boards (PCB). It is instantly followed by the component placement stage. There are some equipment and materials used in this stage such as solder printing machine, stencil and solder paste. The use of stencil printing was developed during the late 1960s for the deposition of conductive interconnects as companies looking forward to increase their production rate and reduce the cycle time for assembly the product as it can reduce the production [1]. This process is called as Surface Mount Assembly (SMA) and is currently used today in almost electronic industries. The stencil printing process is commonly known in SMA as the major contributor to solder defect by 60% averagely[2].

The stencil printing quality determine the performance of the electronic device. The main material that had been used in the process is solder paste which consists of solder metal and flux. It acts as an adhesive during component placement and solder reflow stages[3]. The solder paste enables the components to stick in place by the tackiness behaviour. A good solder joint is when the solder paste has melted well and flowed and wetted the lead on the component and the pad on the circuit board.

There are some parameter of the solder paste that can determine a good solder joint. Firstly, the right solder paste volume need to be applied in order to prevent the insufficient filling of solder in PCB. Besides, the height and area of filling of solder also determine the quality of the solder printing process[4]. So the area on stencil opening on the solder pad have to be optimize in order to prevent the quality issue of solder filling. The optimum squeegee angle and speed also can improve the quality of stencil printing process[5].

Basically, the process of stencil printing begins with loading the PCB into the stencil printing machine. The internal vision system in the machine will aligns the stencil to the PCB then the squeegee will print the solder on the PCB. Next, the stencil is unloaded from the PCB and it will go through to the next stage. Figure 1.2 shows the schematic diagram of stencil printing process[6].

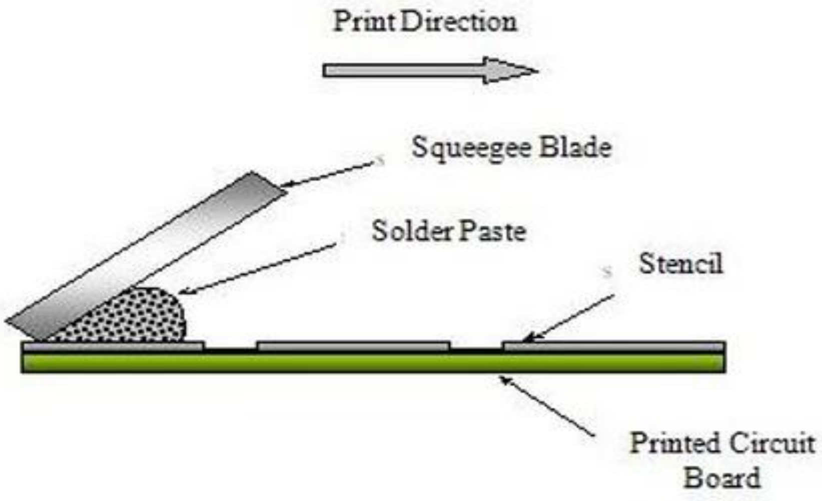


Figure 1.2: Schematic diagram of stencil printing process[7]

The most important material in stencil printing is solder paste. Solder paste is a material used in the manufacture to connect the surface mount device (SMD) on the of printed circuit boards (PCBs) on the board pad. The behaviour of solder paste initially adheres components in place by being sticky, it is then heated along with the rest of the PCB to melt the solder paste and forming a mechanical connection as well as an electrical connection[5]. In this project, 5 different type of solder paste were used that consist different of material composition and mixture. The 5 type of solder paste that we use in this project is SAC 105, SAC 305, SAC 307 and SAC 405, Sn100C and SnPb. Figure 1.3 shows the solder paste sample for solder printing process.

Figure 1.3: Sample of solder paste SAC305[8]

1.1 Overview

A solder paste is essentially a viscous non-Newtonian fluid where the resistance to flow is not constant. It is consists of powder metal suspended in a thick medium called flux. Flux is added to act as a temporary adhesive, holding the components until the soldering process melts the solder due to oven reflow process and makes a stronger physical connection. The solder paste have many type of composition depend to the intended use. For example, the solder compositions is eutectic Sn-Pb (63% tin, 37% lead) or SAC alloys (tin/silver/copper, named for the elemental symbols Sn/Ag/Cu)[9].

The solder paste is an essential requirement for the stencil printing process as the solder paste must flow in and out of the stencil apertures during the printing and remain the position after the process without slumping. The solder paste is called as thixotropic as the viscosity of solder paste changes over time with applied shear force. After the stirring or the movement during the stencil printing, the paste become less viscous[10]. As a thixotropic, solder paste have a yield point which indicates the minimum shear stress where it does not move. The solder paste viscosity also depend on temperature as the viscosity of paste will fall off at the high temperature[11]. It is because of the change of state of the material in that composition.

The rheology of solder paste is the identification of the flow and deformation under a certain condition such as temperature and force[12]. The rheology of solder paste can affect stencil printing quality due to it deformation and flow and the characteristic have to be matched to the application.

The most of the defects in electronic board assembly are caused due to issues in the solder-paste printing process and due to defects in the solder paste. There are some different types of defects possible such as too much solder, the solder melts and connects too many wires (bridging effect) resulting in a short circuit[9]. Besides, Insufficient amounts of paste that can result in incomplete circuits[13]. The solder paste distribution is always being analyse using Solder Paste Inspection (SPI). It is to determine the quality of solder paste that had been deposited on the PCB during stencil printing process.

1.2 Problem Statement

Nowadays, solder-paste printing process caused the most defects in circuit-board assembly in SMT lines due to defects in the solder paste. There are many different types of defects possible such as too much solder, solder melts and connects too many wires (bridging effect) resulting in a short circuit. Besides, insufficient amounts of solder paste will cause in incomplete circuits. Intermetallic compound issue also can cause the defect in circuit board assembly. The solder paste's physical characteristics like viscosity need to be monitored by performing an experiment to get a better understanding.

The presence of the defect is lowering the performance of stencil printing process. To overcome the problem, we have to control some parameter such as printing speed and printing load on a different type of solder paste. Besides, the distribution of solder paste deposited on the PCB or copper substrate have to be analyse to determine the quality of aperture filling of solder paste. The better understanding of this issue can be gained as we doing an experimental on the different solder paste distribution on PCB under 3D inspection machine.

Due to this problem, there is some parameter that have to be control to have a good solder printing which is the height, volume and the area of the solder paste. The are some of sample of solder paste that have to be study which is SAC 105($Sn_{98.5}Ag_{1.0}Cu_{0.5}$), SAC 305($Sn_{96.5}Ag_{3.0}Cu_{0.5}$), SAC 307($Sn_{96.3}Ag_{3.0}Cu_{0.7}$), SAC 405($Sn_{95.5}Ag_{4.0}Cu_{0.5}$), Sn100C($Sn_{99.3}Cu_{0.7+Ni+Ge}$) and SnPb($Sn_{63}Pb_{37}$).

1.3 Objective

The objective of this study are:

- I. To analyse the effect of printing direction on the aperture filling quality.
- II. To investigate the effect of the printing speed on the stencil printing performance.
- III. To study the effect of the printing load on the printing quality.

1.4 Scope of Study

The scope of work is to study the material properties of the solder paste on the different solder paste which is SAC 105, SAC 305, SAC 307, SAC 405, Sn100C and SnPb. The physical characteristic of these type of solder paste can be determined by the solder paste distribution on PCB and copper substrate the can be view under 3D inspection machine.

Besides, the experiment is carried out to investigate the effect of stencil printing performance on different condition due to different type of solder paste such as the printing speed, printing load and printing direction. This will help to get a better understanding for optimizing the process. The investigation on the quality of aperture filling of solder paste have to be done by analysing the solder paste distribution on PCB and copper substrate by using Koh Young 3D inspection machine.

CHAPTER 2

LITERATURE REVIEW

2.1 Optimizing stencil printing performance

This study has stated that stencil printing process contribute an average 60% of soldering defects are attributed in surface mount assembly (SMA)[2].Due to this issue, Tsai et al. using hybrid intelligence technique and response surface methodology (RSM). The purpose of their study is to compares two hybrid intelligence approaches with RSM as methods to breakdown the stencil printing optimization problem that involves variety performance characteristics.

A set of data can be obtained from an experimental approach. Figure 2.1 shows the experimental procedure for the overall stencil printing process[2]. The experimental result will be validate with another experiment to provide an accurate source of data for RSM study and training neural networks to formulate the nonlinear model of the stencil printing process with or without combining multiple performance characteristics into a single desirability value, followed by a genetic algorithm searching the trained neural networks for obtaining the optimal parameter sets. The finding of this project shows that the empirical defect-per-million-opportunities (DPMO) measurements demonstrate that the two hybrid intelligence methods can provide a better performance for stencil printing process[2].

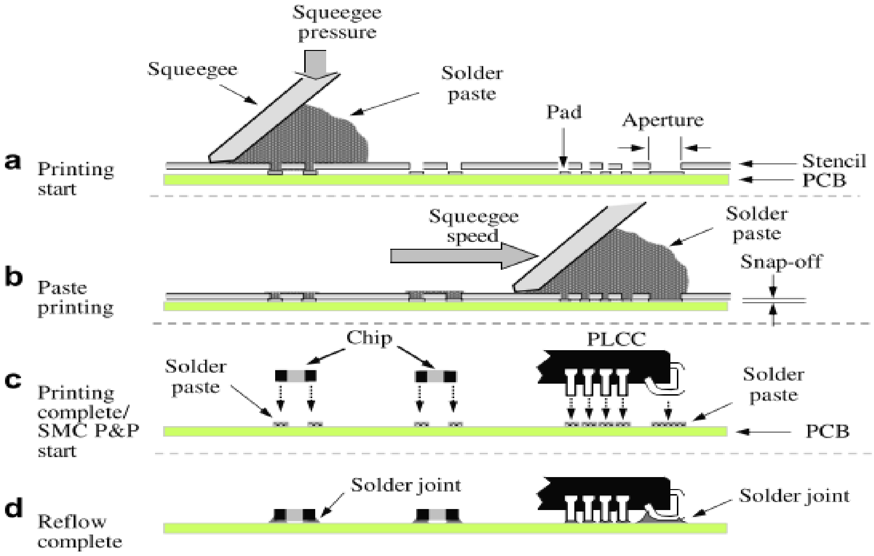


Figure 2.1: Surface mount assembly (SMA) processes[2]

2.2 Effect of Optimal Reflow Profile(ORF)

The good stencil printing and quality reflow solder printing enhanced the reliability of solder joints in electronic. The key factor which affects the quality of solder printing is when the solder paste release from stencil during printing. Thus, the main concern of SMT production process is the efficient transfer rate of paste from stencil through aperture to pad. Based on Amalu et al., the recent trends on further miniaturisation of electronic products have introduced more process challenges in the SMT assembly area[14].

A good transfer efficiency of solder paste through small stencil aperture is required for the effective mounting of these electronic device. Amalu et al. had done an investigation of the transfer efficiency of type 3, 96.5Sn3.0Ag0.5Cu solder paste through linearly decreasing rectangular stencil aperture sizes typically used in PCB assembly[14]. Moreover, the objective of this study is to analyse he effects of optimal reflow profile (ORP) on the deposited solder paste volumes. The finding of the study shows a power law relationship between actual solder deposit volume (SDV) and aperture cavity. The result of the experiment shows the effect of ORP on actual SDVs was a 46% volume change which was fairly constant across the pad geometries[14].The result shows that the transfer efficiency of solder paste decreases with decreasing stencil aperture size.

2.3 Lead free solder pastes composition for ultra-fine pitch

The stencil printing issues in SMT however still have some defects such as bridging issue and insufficient solder paste deposited on the solder pad which could lead to major reliability issues. It can be controlled and or minimised through proper investigation and control of the flow and deformation behaviour of solder pastes[15]. Amalu et al. study about characteristic of three different Pb-free solder pastes composition that used for ultra-fine pitch assembly applications[15]. This study also concern on the paste transfer efficiency through linearly varying stencil apertures filling sizes, and correlation between the paste rheological properties with the transfer efficiency[15].

The purpose of the experiment is to provide better understanding of the effects and interactions of stencil printing process variables on the solder paste transfer efficiency. The experiment consist of three rheological tests which is the viscosity, thixotropic and oscillatory amplitude sweep to employ the characterisation of the solder paste. The experiment begin by printing solder paste samples on copper substrates using stencil printing with varying aperture cavity dimensions in the range 0.79 mm^3 –1.62 mm^3[15]. The finding of the study shows that the solder paste type and volume of stencil aperture interact during printing and the solder paste with unique rheological properties produced distinctive transfer pattern[15].

2.4 Rheological behaviour and stencil printing performance

Durairaj et al. run an investigation on rheological behaviour and stencil printing performance of the lead-free solder pastes (Sn/Ag/Cu) to improve the rate of production in SMT production line[16]. The study of visco-elastic behaviour of the lead-free solder pastes can be carried out with oscillatory stress sweep test. The visco-elastic behaviour of the paste encompasses solid and liquid characteristic of the solder paste, the purpose of study is to get the proper understanding of the flow behaviour experienced by the pastes during the stencil printing process. The finding of the study found that the solid characteristics (G') is higher than the liquid characteristic (G'') for the pastes material[16]. The results from the study shows that the solder paste with a large $G' = G''$ has a higher cohesiveness resulting in poor withdrawal of the paste during the stencil printing process[16]. The phase angles (δ) was used to correlate the quality of the dense suspensions to the formulation of solder paste materials[16].

2.5 Influence of the solder paste formulation on wall-slip formation

Besides, the flow behaviour of solder paste materials also effected by wall-slip effect. The wall-slip happened due to the various attractive and repulsive forces acting between the solder particles and the walls of the measuring geometry. Durairaj et al. state that the interactions could lead to the presence of a thin liquid layer adjacent to the wall, which causes slippages[17]. They are doing an investigation to study the influence of the solder paste formulation on wall-slip formation and its effect on these pastes material printability[17]. The calculation of true viscosity and slip velocity for the lead-free solder pastes samples can be utilised by wall-slip model. Based from their finding, the wall-slip formation between the solder pastes and the parallel plate indicate the difference in the measured viscosity and the true viscosity[17]. The slip velocity calculated for the solder pastes could be used as a performance indicator to get a better understanding on the paste release characteristics in the stencil printing process.

2.6 Effects of diamond nanoparticles reinforcement on lead-free SAC 305 solder paste

Furthermore, there are some improvement that improve the solder paste rheology which is presents the effects of diamond nanoparticles reinforcement on lead-free SAC 305 solder paste after the reflow stencil printing process[18]. Chellvarajoo et al. study the different diamond nanoparticles amounts (0.5, 1.5, and 2.5 wt.%) mechanically mixed with SAC 305 to produce a new form of nano-composite solder paste[18]. The characteristics of the nano-composite solder, such as melting point, morphology and thickness of the intermetallic compound (IMC), agglomeration of diamond nanoparticles, and hardness, were investigated by doing an experiment on stencil printing process to have better understanding on this type of solder paste[18].

The finding of this study shows that the addition of diamond nanoparticles slightly decreases the melting point after the result of the experiment revealed but significantly reduces the IMC thickness[18]. The result of the experiment shows morphologies of the nano-reinforced solder paste showed the agglomeration of nanoparticles on the surface of the solder paste with increasing diamond nanoparticles percentage[18]. Nanoindentation technique had been used to evaluate the hardness of the nano-reinforced solder paste. The addition of 0.5 wt.% diamond nanoparticles improved the hardness of SAC 305 by 77.5% and increasing the nanoparticles amount by 1.5 and 2.5 wt.% in SAC 305 enhanced the hardness of SAC 305–0.5 wt.% by 6.3% and 17.8%, respectively[18].

CHAPTER 3

METHODOLOGY

3.1 Experiment Plan

This project can be obtained by achieving the objective the project. Before start the experiment, the proper understanding on the material properties of solder paste. The experiment start with the experiment setup. The PCB dimension and the solder pad location had to be teach in the DEK 265 stencil printing machine. The stencil had to be prepared for the substrate and the solder paste is ready to be used. The experiment was performed at ambient temperature and pressure which was approximately 25℃ and 1 atm. Figure 3.1 shows the stencil printing experiment’s flow chart.

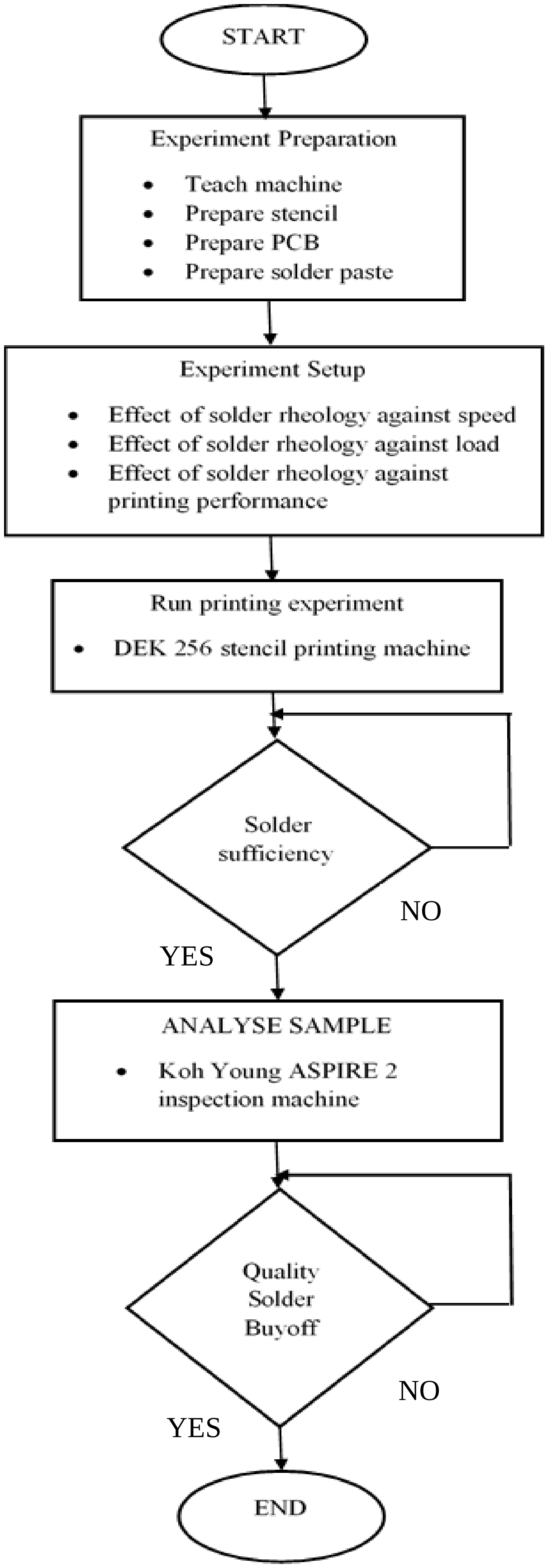


Figure 3.1: Solder printing’s experiment flow chart

The experiment is carried out by two different condition which is firstly run the experiment in different printing which is 15mm/s, 25mm/s, 35mm/s, 45mm/s and 55mm/s with constant printing load at 9 kg. The next experiment is to taking the result on the rheology of the different solder paste type and recorded in forward mode of printing movement. The experiment then repeated with different printing load which is at 7 kg, 8 kg, 10 kg and 11 kg with constant printing speed which is 35mm/s. Next, the experiment is taking the result on the rheology of the different solder paste type and recorded in forward mode of printing movement. The experiment is then repeated at the same parameter tested with the different printing movement which is reverse mode. The solder filling is sufficient on the substrate then we can move to the next step which is analysing the solder paste distribution on the substrate using Koh Young 3D inspection machine.

The setup of experiment will be held on January 2018 at Celestica Malaysia at Kulim, Kedah due to the stencil printing experiment at USM Mechanical Engineering School cannot be followed by inspection process because of Alicona Infinite Focus is still under maintenance issue. For semester 1, the experiment setup on stencil printing machine has been done to ensure the project can be continue and check the suitability of the experiment for this project. For the next semester, the data collection have to be taken to provide the result for the project.

3.2 Experiment Preparation

The experiment is firstly done by measuring height, length and width of Printed Circuit Board (PCB). Figure 3.2 shows the measurement of PCB. Then, the experiment is then followed by measuring the component configuration in the PCB. In the PCB that has been tested consist of 5 different component configuration which is have the different size of solder pad. The 5 different configuration is 0603 component, Small Outlet Transistor(SOT) component, Metal Electrode Leadless Face(MELF) component, 1210 component and Tantalum-D component. The 5 different size in the 5 different component configuration is depend on the size and type of component that will be mounted in the PCB. Table 3.1 shows the type of component configuration and the area of the solder pad. Note that 1mills equals to 0.0254mm.

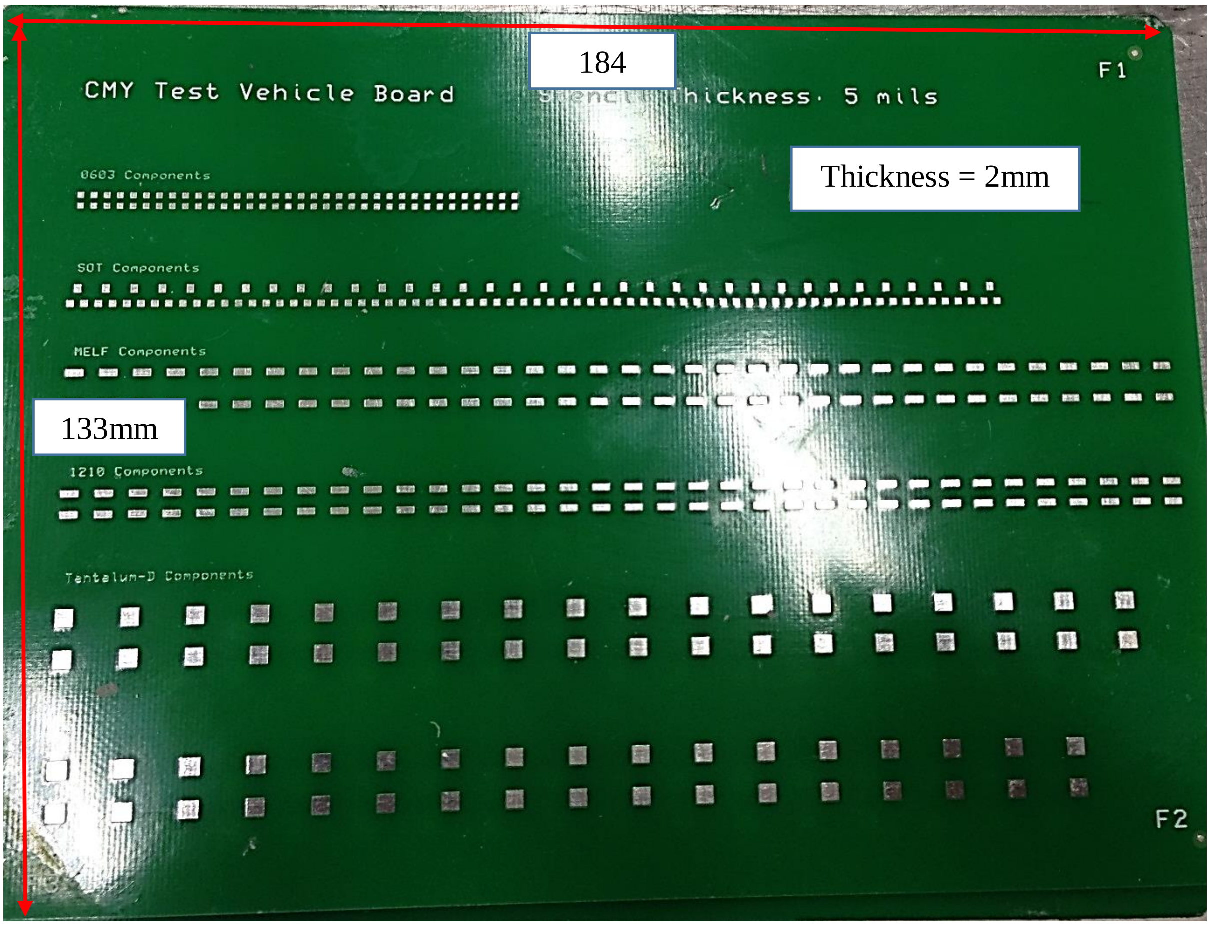


Figure 3.2: The measurement of PCB

Table 3.1: Type of component configuration and the size of the solder pad.

Configuration	**Component**	**Length(mil)**	**Width(mil)**	**Area(mil²)**
1	Tantalum-D component	100	100	10000
2	MELF component	100	43	4300
3	1210 component	100	40	4000
4	SOT component	40	40 & 50	1600 & 2000
5	0603 component	31	35	1085

3.3 Experiment Setup

The experiment setup is done by preparing the materials and tools that have to be use in the experiment. The list of tools and material have to be prepared before running an experiment. The tools that needed in this experiment is Printed Circuit Board (PCB) and stencil that had been designed that match with PCB's solder pad position. The other tools needed is squeegee blade and stirrer. The material that have be prepare for the experiment is 6 different type of solder paste which is SAC105, SAC305, SAC307, SAC405, Sn100C and SnPb. The other material that need to prepare is Acetone, tissue and marker pen. The tools and materials used in the experiment are listed in Table 3.2 below.

Table 3.2: The tools and materials used in the experiment

Materials	**Tools**
Solder Paste: • SAC 105 • SAC 305 • SAC 307 • SAC 405 • Sn100C • SnPb	Printed Circuit Board(PCB) (quantity=10)
Acetone	Stencil
Tissue	60° Squeegee blade
Marker Pen	Stirrer

The experiment will be run using DEK 265 Solder Printing Machine. The purpose of this machine is to run the solder printing process that will mount the solder paste on the top of PCB's solder pad. Figure 3.3 shows the DEK 265 Solder Printing Machine. The next machine involve in this experiment is Koh Young Aspire 2 inspection machine. This machine will analyse the solder filling in the PCB's solder pad in 3 different parameter in term of volume, height and area. Figure 3.4 shows the Koh Young Aspire 2 Inspection Machine.

Figure 3.3: DEK 265 Solder Printing Machine

Figure 3.4: Koh Young Aspire 2 Inspection Machine

3.4 Experiment on The Effect of Printing Speed

The experiment is carried out by setting the printing speed parameter while the printing load parameter will be constant. The squeegee and stencil that had been designed match with PCB's solder pad was been apply to the DEK 265 machine. Then, solder paste SAC105 had been apply on the stencil. Before the solder paste had been apply on the stencil, the solder paste must be stir using a stirrer to make the mixture of the solder paste blended well. Figure 3.5 shows the tools setup in the experiment.

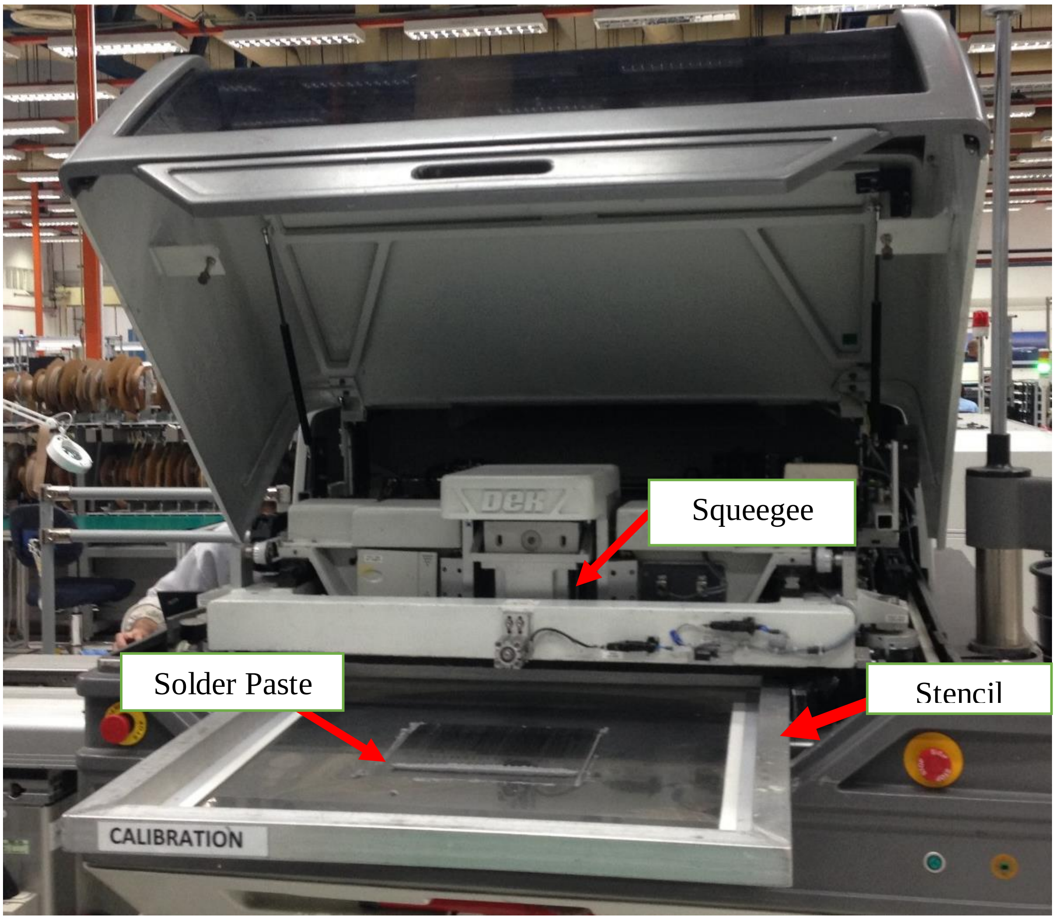


Figure 3.5: The tools setup in the experiment.

The printing speed of 15mm/s was been set to the control panel of DEK 265 solder printing machine while the printing load was been set at 9 kg. Then, the PCB must be label for data collecting purposes. After labelling the PCB, place it on the conveyer then click "run" on the control panel then the first printing process will be in the forward direction of printing. Figure 3.6 shows the PCB that had been mounted by solder paste. Next, place the other labelled

PCB on the conveyer and click “run” again for the reverse direction of printing process. Table 3.3 shows the printing mode of solder printing process. Repeat these step by adjusting the printing speed to 25mm/s, 35mm/s, 45mm/s and 55mm/s. After all these step finish, the stencil and squeegee will be taken out from the machine for cleaning process. The tools must be clean using alcohol like acetone to remove the excess flux and the material composition from the previous solder paste before applying another type of solder paste. After the cleaning process is finish, the experiment will be continue by apply the different type of solder paste which is SAC305, SAC307, SAC405, Sn100C and SnPb.

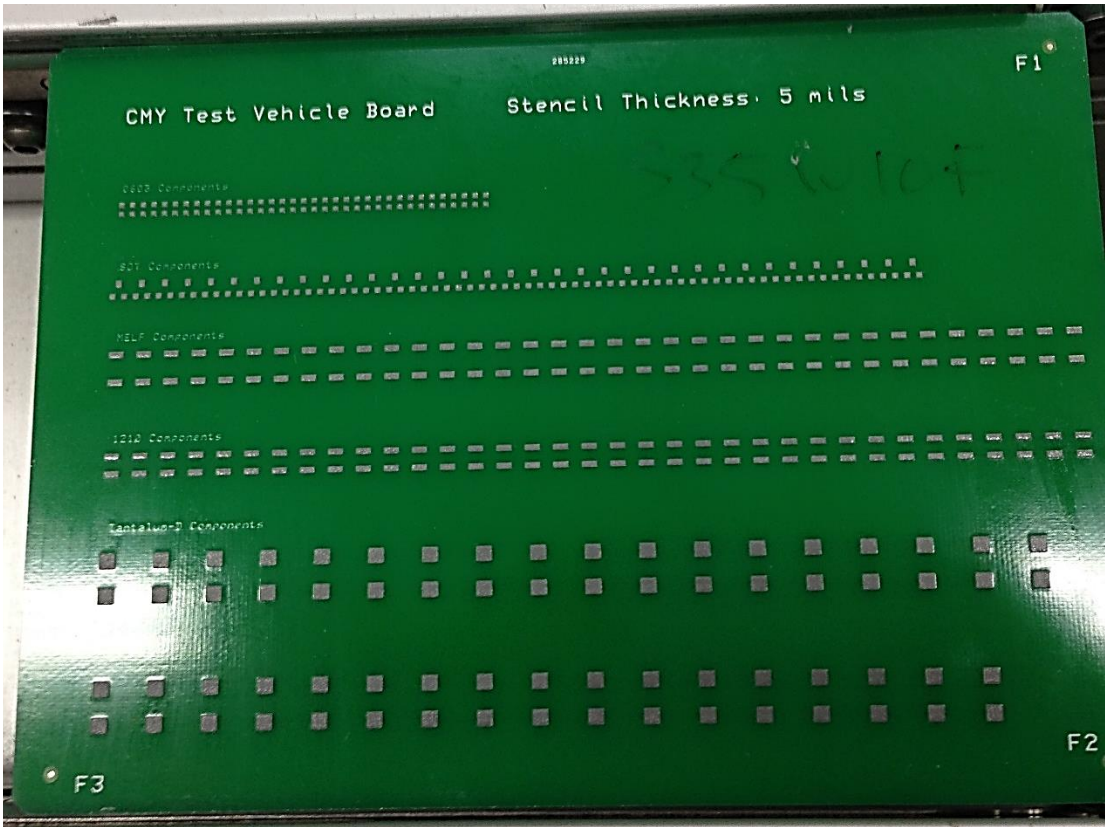
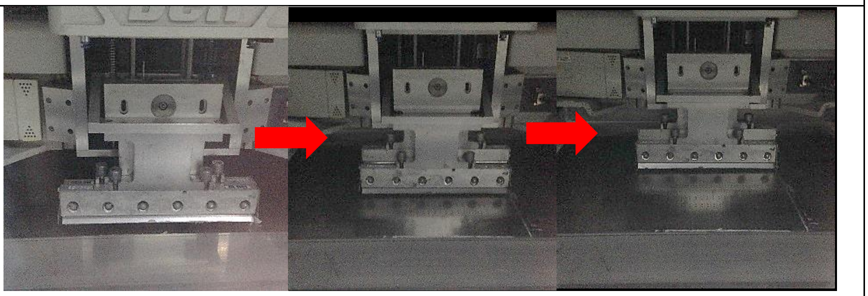
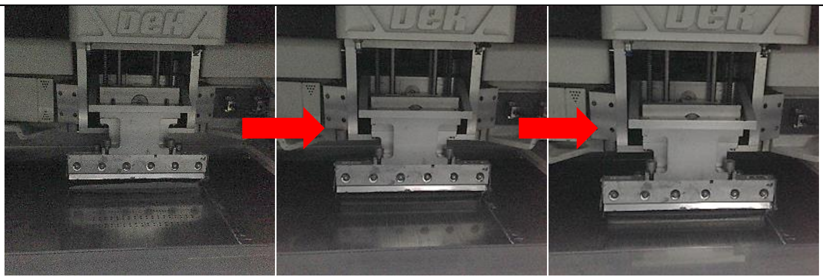


Figure 3.6: PCB that had been mounted by solder paste

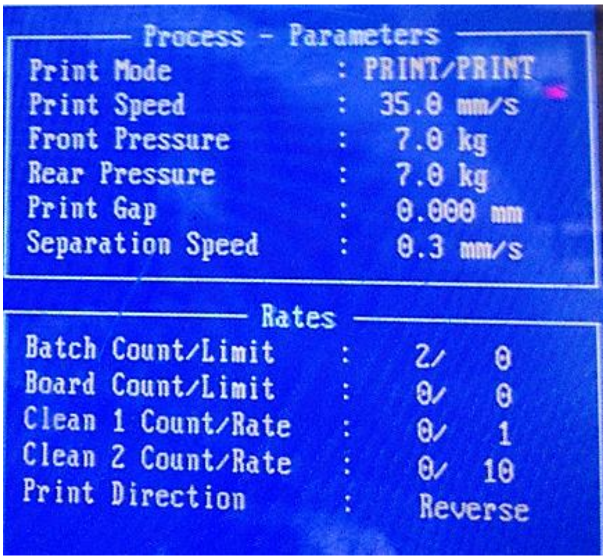
Table 3.3: The printing mode

Printing Mode	Figure
Forward direction	
Reverse direction	

3.5 Experiment on Effect of Printing Load

The experiment is carried out by setting the printing load parameter while the printing speed parameter will be constant. The squeegee and stencil that had been designed match with PCB's solder pad has been apply to the DEK 265 machine. Then, solder paste SAC105 had been applied on the stencil. Before the solder paste applied on the stencil, the solder paste must be stir using a stirrer to make the mixture of the solder paste blended well.

The printing load of 7kg was set to the control panel of DEK 265 solder printing machine while the printing speed was set at 35mm/s. Figure 3.7 shows the parameter that had been set at the control panel. Then, the PCB must be label for data collecting purposes. After labelling the PCB, it was placed on the conveyer then click "run" on the control panel then the first printing process will be in the forward direction of printing. Next, the other labelled PCB was placed on the conveyer and click "run" again for the reverse direction of printing process. Repeat these step by adjusting the printing load to 8kg, 10kg, and 11kg. After all these step finish, the stencil and squeegee will be taken out from the machine for cleaning process. The tools must be clean up using alcohol like acetone to remove the excess flux and the material composition from the previous solder paste before applying another type of solder paste. After the cleaning process is finish, the experiment will be continued by apply the different type of solder paste which is SAC305, SAC307, SAC405, Sn100C and SnPb.



```
Process - Parameters
Print Mode              : PRINT/PRINT
Print Speed             :  35.0 mm/s
Front Pressure          :   7.0 kg
Rear Pressure           :   7.0 kg
Print Gap               :   0.000 mm
Separation Speed        :   0.3 mm/s

Rates
Batch Count/Limit       :    2/   0
Board Count/Limit       :    0/   0
Clean 1 Count/Rate      :    0/   1
Clean 2 Count/Rate      :    0/  10
Print Direction         :    Reverse
```

Figure 3.7: The set parameter at the control panel

3.6 Experiment Analysis

The experiment continued by inspection process done by Koh Young Aspire 2 to determine the quality of solder paste on the solder pad. This machine measure the volume, height and area of solder filling in the solder paste. Firstly, the measurement of PCB and the component configuration of solder pad had to be teach to the machine using software KY-3030 system. For this experiment, the optimum height for the solder paste is 5mill. Figure 3.8 shows the measurement of component configuration that had been set to the machine.

spt_r2 / 385 Pads

PCB ID : 487

Shrinkage Rate: 99.987%

Figure 3.8: Measurement of component configuration setting

Next, the PCB that already had been through solder printing process is then placed on the conveyer of the inspection machine then click “STRART”. The conveyer had to be adjusted as the same length as the PCB before place it. The machine is then use the infrared light to measure the parameters which is volume, height and area of solder paste on the solder pad. Figure 3.9 shows the inspection process. The software that had been used is SPC Plus to measure the quality of the solder paste that had been mounted on the PCB. Figure 3.10 shows the software used in Koh Young Aspire 2. After done measure the parameters, the result will be shown in raw data. Figure 3.11 shows the sample high quality of solder filling result. Finally, the raw data have to be save in the desktop for data collection. The sample on non-good unit will be automatically shown in the system. Figure 3.12 shows the sample of non-good unit. The good quality sample means the solder amount on the solder pad is sufficient and had no defect in solder filling. The non-good unit means the solder filling is out of criteria which means there are must be insufficient solder of too much solder that can contribute a defect for the next process.

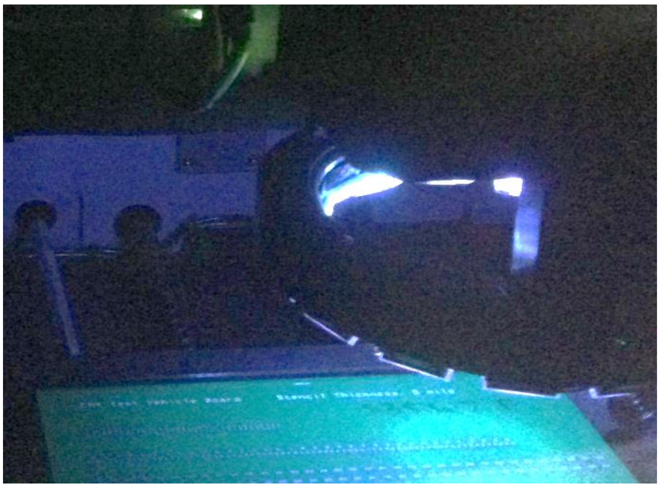


Figure 3.9: The inspection process.

Figure 3.10: The software used in Koh Young Aspire 2

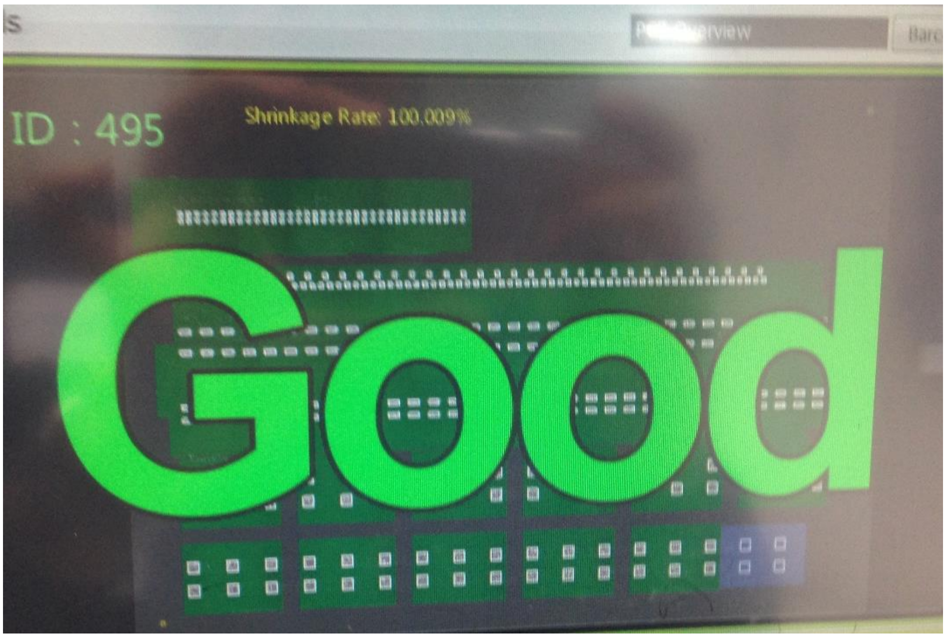


Figure 3.11: The sample high quality of solder filling result

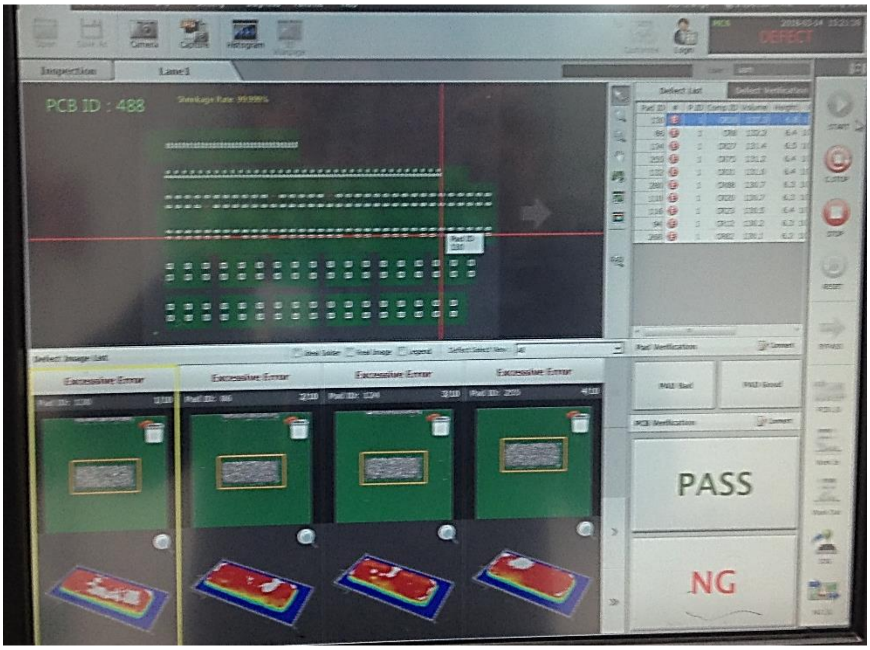


Figure 3.12: The sample of non-good unit

CHAPTER 4

RESULT AND DISCUSSION

4.0 Overview

The result of the experiment will be divided into 3 different study to analyse the performance of solder paste. Firstly, the study is to analyse the effect of printing direction mode on solder printing performance. During the solder printing process, there are 2 type of direction which is in forward direction and the reverse direction. The present study is to determine whether the printing direction mode will be effect the solder performance during solder printing. Secondly, the study is to analyse the effect of solder rheology during printing speed of the solder printing performance. The result will be determine the performance of solder printing using the different printing speed. The result also determine the effect of different solder rheology on the solder performance. The result also determine the effect of different solder rheology by the different type of solder paste influence the solder performance in various printing speed. Lastly, the study is to analyse the effect of solder rheology on printing load of solder printing performance. The result in this study will be determine the performance of solder printing using variety of printing loads

The raw data had been obtained during the experiment and the analysis of data gained in the discussion. The recorded data can be tabulated in the table based on component configuration. There are 3 component configuration that will be focus in this analysis data which is Tantalum-D component, 1210 component and 0603 component which contain 35 unit of solder pad each. This component is selected based on the area measurement. Tantalum-D has the biggest area of solder pad on PCB which is 10000mill^2. 1210 component has the medium area of solder pad which is 4000mill^2 while 0603 component has the smallest solder pad area which is 1085mill^2. The result that would be carried out is the area of solder filling and the height of solder filling on the solder pad. The average area and height is then calculated to get the average of 35 unit of solder pad. The optimum parameter of this study is shown in the Table 4.1.

The data will be influenced by the alloy composition in 6 different type of solder paste. The different composition in solder paste may affect the behaviour as the viscosity and the density of the solder paste is different. Table 4.2 shows the alloying composition on the 6 different type of solder paste.