STRUCTURAL AND RELIABILITY ANALYSIS OF SOLDER JOINT UNDER VIBRATION LOADING

By:

MUHAMMAD ASYRAF BIN WAN CHEK

(Matric number: 128953)

Supervisor:

Dr. Abdullah Aziz Saad

May 2019

This dissertation is submitted to

Universiti Sains Malaysia

As a partial fulfilment of the requirement to graduate with honours degree in

BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in a candidature for any degree.

Signed...
(MUHAMMAD ASYRAF BIN WAN CHEK)

Date ....

Statement 1

This thesis is the result of my own investigation, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed...
(MUHAMMAD ASYRAF BIN WAN CHEK)

Date ....

Statement 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed...
(MUHAMMAD ASYRAF BIN WAN CHEK)

Date ....

ACKNOWLEDGEMENTS

First and foremost, I would like to give my deepest gratitude to my supervisor, Dr. Abdullah Aziz Saad for continuous support and guidance throughout the whole project. Despite being extraordinary busy with his duties, I am very thankful of the time allocated for meeting me. I am grateful of the help and ideas given as I am finishing this project.

Moreover, I would like to thank the coordinator, Dr. Ikhwan Zaini and group of lecturers that have helped me so much in writing the report. They conducted several seminars about thesis writing.

I also would like to express my gratitude to postgraduates, Dr. Mohammad Hafifi Hafiz Ishak and Mohamad Fikri bin Mohd Sharif for helping me in technical issues that I am lacking as I am working on this project.

I would like to thank the technical staffs for coping with me when I were using the facilities especially computer in CAD Laboratory and School of Mechanical Engineering, Universiti Sains Malaysia for giving me chance in developing this project.

Finally, my highest appreciations to my beloved family and friends for the moral support and encouragement which helped me in the completion of this research.

Title of thesis: **Structural and Reliability Analysis of Solder Joint under Vibration Loading**

Date of submission (Academic year): **29th May 2019**

Candidate (Matrix no.): **Muhammad Asyraf Bin Wan Chek (128953)**

Name of supervisor: **Dr. Abdullah Aziz Saad**

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATION

PCB	Printed Circuit Board
PB-Free	Lead-Free
PBGA	Plastic Ball Grid Array
PWB	Printed Wiring Board
ICA	Isotropically Conductive Adhesive
LCP	Liquid Crystal Polymer
FEM	Finite Element Method
SN	Stress Life
FEA	Finite Element Analysis
CAD	Computer-Aided Design
RAM	Random Access
UTM	Universal Testing Machine
PSD	Power Spectral Density
UTS	Ultimate Tensile Strength
RMS	Root Mean Square

ANALISIS STRUKTUR DAN KEBOLEHPERCAYAAN SAMBUNGAN PATERI DI BAWAH BEBAN GETARAN

ABSTRAK

Kebanyakan peranti kini mengandungi bahagian elektronik dan sering dikendalikan dalam persekitaran getaran untuk tempoh yang panjang tanpa gagal. Peranan pateri dalam peralatan elektronik telah berkembang dan bertindak sebagai penyambungan elektrik serta ikatan mekanikal ke bahagian-bahagian tertentu tatkala berlakunya penurunan dari segi kos dan saiz. Oleh itu, adalah penting untuk menyiasat struktur sendi pateri dan kebolehpercayaannya di bawah beban getaran. Model telah disimulasikan dalam analisis struktur untuk mensimulasikan ujian daya tarikan untuk mempelajari struktur dan kekuatannya semasa berada di bawah getaran rawak untuk mempelajari tingkah laku dan tindak balas dinamik di bawah kekerapan rawak tertentu. Dalam analisis struktur, model tertakluk kepada pemuatan anjakan paksi-Y di mana model terdedah kepada julat frekuensi 3 Hz hingga 500 Hz dalam analisis getaran rawak, Hasil daripada analisis struktur menunjukkan bahawa struktur pateri mempengaruhi kekuatan sambungan antara pad litar dan sambungan LED. Kandungan perekat yang lebih tinggi dapat memberikan kekuatan lebih kepada struktur. Semua bahagian model CAD teruja apabila didedahkan dengan frekuensi dan kegagalan berlaku di beberapa bahagian model pada keadaan tertentu. Secara keseluruhannya, penyelidikan yang dilakukan membolehkan parameter kritikal model ditemui berdasarkan kelakuannya dalam analisis ini.

STRUCTURAL AND RELIABILITY ANALYSIS OF SOLDER JOINT UNDER VIBRATION LOADING

ABSTRACT

Most of the devices nowadays contained electronic parts and often operated in a vibration environment for extended periods without failing. The role of solder in electronic packages has expanded and act as electrical interconnection as well as mechanical bond to certain parts while the joint decrease in size and cost. Hence, it is essential to investigate the structure of the solder joint and its reliability under vibration loading. Models were simulated in structural analysis to simulate pull strength test to learn the structure and its strength while under random vibration to learn its behaviour and dynamic response under certain random excitation frequency. Models were subjected to Y-axis displacement loading in structural analysis where as in random vibration analysis, the models are exposed to frequency range of 3 Hz to 500 Hz. Result from structural analysis shows that the structure of solder joint affects the strength of interconnection between circuit pad and LED joint. Higher number of adhesives able provide more strength to the structure. All parts of the CAD model were excited from the exposed frequency and failure occurred at some parts of the model.

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

The most company nowadays use the electronic system as a control module, especially Automation Company. Electronic systems are often required to operate in severe vibration environments for extended periods without failing. Not automation only use this system but some of application and company also using the electronic system such as airplanes, fossil fuel power plants, oil drilling equipment, elevators communication systems, petroleum refineries, light, and heavy manufacturing and others. Besides that, solder joint of the electronic system always in problem when under vibration condition.

The solder is an interconnecting electrical or mechanical bond and must often serve as a thermal conductive to remove heat from joined devices. For example, in electronic assemblies, such as television sets and radios may not operate in vibration environments of the sound wave, but they need to survive that vibration when they are being transported from the manufacturer directly. But they have the problem when the interconnecting of the electrical become failures.

One of the causes of those system failures is about the exposure to fatigue when the larger number of the low amplitude cycle or fewer number of high amplitude occurs. For both vibration and temperature heating cycle contribute to this failure [1]. However, most of the reliability of those systems is focused on thermal loading, while relatively few works consider mechanical loading and even fewer works consider a combination of effect for both thermal and vibration.

Besides that, failures in electronic equipment hardware by the U.S. Air Force over a period of about 20 years show that about 40% of these failures are related to connectors, 30% to interconnects, and 20% to component parts [2]. It is also stated that this condition is due to handling, vibration loading, shock, and thermal cycling. The most insidious environments for solder degradation are high-temperature aging and thermomechanical fatigue because soldiers are microstructural unstable and evolve with strain, temperature, and time.

Furthermore, the mechanical performance of the component has been used from that electronic system can be analysed through finite element modelling. The model is first verified by using experimental of vibration loading to obtain the result for the natural frequency, mode shapes and transfer function. From this result, it can verify the structural and reliability for the solder joint in the electronic system.

1.2 PROBLEM STATEMENT

The traditional solders from the older company that contains a mixture of tin and lead (SnPb) are no longer used in the electronic industries. They are converting from the tin-lead solders to lead-free (Pb-free) soldiers in order to meet new environmental and green requirements [1]. There are some issues about the problem of the strength of the new solder joints due to some problem. A lot of studies have been done to study on the microstructure of the solder joint. Besides that, the most important aspects of the solder joint are strength and stress in their structure which can only be improved by adding some Nanoparticles to the joints.

Electronic equipment can be subjected to many different forms of vibration whether due to transporting the equipment from the manufacturer to the customer or due to an active association with some sort of a machine or a moving vehicle. There are few variables that affect the solder joint failure under vibration condition such as solder material, geometry structure of solder joint and solder joint finishing treatment. Besides that, the effect of the solder joint failure can be carried out using an experiment of vibration loading to obtain the result of the natural frequency, mode shapes and transfer function.

Hence, the project aims to study the structural and reliability of solder joint under vibration loading. Thus, a study of finite element analysis along with ANSYS static structural are used to determine the solder joint stresses.

1.3 OBJECTIVE

The objectives of this project are listed as below:

1. To construct a finite element model to perform an analysis on solder joint under vibration condition.
2. To study the relationship between solder joint reliability and material of solder under vibration condition.

1.4 RESEARCH SCOPE

The main focus of this work is to perform a finite element analysis using a model of electronics components assembly and conduct tensile and vibration test on similar geometry. Modified models will be analysed in structural analysis to analyse the solder joint structure and to identify the strength of the joint. In this project, an assembly model is developed to cater to the finite element analysis under the vibration condition.

The study is performed by:

1. An experimental is done and need to check the natural frequency of interconnection when under vibration analysis.
2. Models were also being analysed in modal and harmonic response analysis to obtain the vibration modes on the models and then being analysed in random vibration to analyse the structure and failure occurred in the model.
3. Assembly is put through under simulation software (ANSYS) to further investigate the stress level of the interconnector.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Solder joint fatigue or failure in electronics device assemblies under vibration are one of the key concerns for the automotive industry and portable devices. Many studies and research has been dedicated to predicting the structure and reliability of solder joint in electronic assembly devices under the mechanical environment, especially in vibration load. Thus, to analyse this problem, experiment and simulation are mostly used to make the dynamic response of the system.

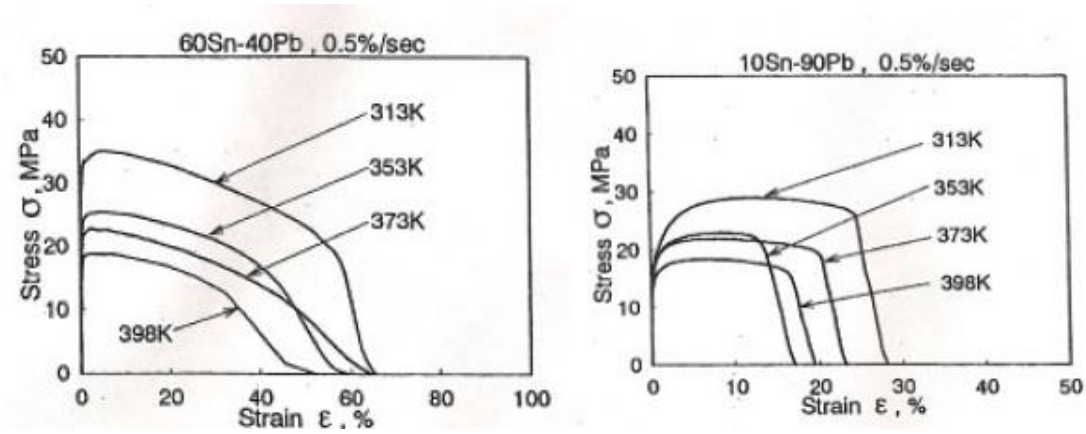
2.2 SOLDER MATERIAL

A material of solder joint is more important to measure the strength and hardness based on environment load. Most of the traditional company does not longer used tin-lead solder (Sn-Pb) in the electronics industry. They are converting from the tin-lead solders to lead-free (Pb-free) soldiers in order to meet new environmental and Green requirements [1].

Lead-free solder is a Tin (Pb) based alloy, common composition of lead-free solder used are Sn-Ag, Sn-Cu, Sn-Zn, and Sn-Ag-Cu. Instead, based on the lead-free solder, it is being used to provide a higher melting point or to satisfy the component and circuit requirement[2]. Based on this, when higher melting point satisfies to the component of the circuit, it will damage the overall circuit especially the solder interconnecting system. Conductive adhesives are a composite of adhesive epoxy and a conductive metal such as silver, copper, tin oxide or indium.

A most common material is as the epoxy filler is silver as it is a good conductor, easy to available and has moderate cost. The conductive adhesive is applied as the solder joint by using the same method as the solder paste, where it is dispensed to the circuit before placement of the electronic component. Conductive adhesive does not need pressure for curing and curing temperature is between 130ºC-180ºC which suitable for the thermal sensitive electronic components[3].

Solder has proven to be the most important and valuable as a mechanical and electrical interconnect material in the electronics industry primarily due to its low melting point, the behaviour of wetting, electrical properties, and availability. Solder has also proven to be a very difficult material to understand for the most of company electronic and electrical as it exhibits such phenomena as age-and cycle softening, grain-growth hardening, strain-rate hardening, and “super-plasticity” [4]. Besides that, plastic behaviour of solder depended on the temperature and specifically of the yield stress. Figure 1 below show as the behaviour type of solder which is 90Pb-10Sn and 60Sn-40Pb as a function of temperature.



(a) (b)

Figure 2.1: Stress-strain curve temperature dependence on the two type of solder (a) 60Sn-40Pb (b) 90Pb-10Sn

2.3 SURFACE MOUNT TECHNOLOGY (SMT)

There are two basics fatigue analysis methods applied in previous studies, stress-life (SN) which uses calculated elastic stresses and empirical stress vs. cycle fatigue curves which applicable to high cycle fatigue. Failure from the fatigue surface SMT can be generated based on the size, type of material, ambient temperature used and placement location or environment that could be established. Reliability predictions were based on part counts and maximum temperatures, not the physical construction detail [5]. Furthermore, fatigue life can be quantified as a function of the imposed environment and the structural and material properties of the component.

During the fatigue testing, specimen or component was subjected at certain bandwidth random vibration excitation spectrum centered about the specimen's first resonant frequency. This narrow band excitation primarily limits the response of the printed circuit board (PCB) to its fundamental bending mode of vibration. Thus, narrowband excitation, higher order modes contribute less than 5% to the response of the PCB [5]. Figure 2 below shows us about finite element fatigue of the PCB specimen in the first bending mode shape.

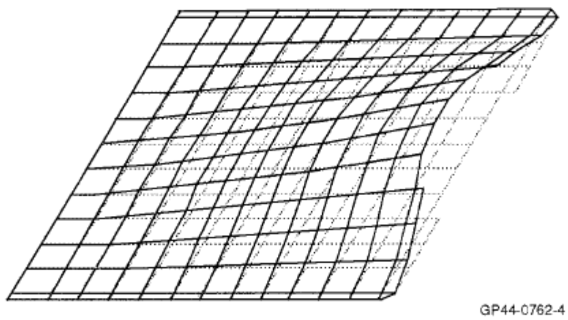


Figure 2.2: Finite element fatigue of the PCB specimen in the first bending mode shape.

2.4 PRINTED CIRCUIT BOARD (PCB)

A printed circuit board (PCB) and electrical electronic component are normally used conductive tracks, pads and other features for their safety support. Components like capacitors, resistors, or active devices are generally used as a soldering on the PCB. Advanced PCBs in our new era may contain components embedded in the substrate. PCB can be single sided, double sided or multi-layer based on what user want to use. Multi-layer PCBs allow for much higher component density.

Electronic components are fixed to the board by soldering process. The copper tracks at the board link the components together and form a complete circuit. Most of the boards are made from fiberglass or glass reinforced plastic with copper traces. The boards can be in single-layer for simple electronic devices and can be up to twelve layers for complex hardware such as computer motherboards and graphic cards. The boards normally come in green colour and also can come in any colour [6]. Figure 2.3 shows the green solder mask is applied to the majority of the PCB.

Printed circuit boards is one of the factors that has enabled electronic circuits to be smaller, more compact, and contained on a convenient. Holes into circuit boards will allow components such as resistors and capacitors to be inserted with the soldered through automation. Today, just about every electronic appliance in our home contains a printed circuit board of some type such as computers, printers, televisions, stereos, musical instrument amplifiers and synthesizers, digital clocks, microwave ovens, telephone answering machines and even cell phones.

One of the important examples is the "motherboard" in a computer which is the main printed circuit board that is that also known as a heart of a computer. Other circuit boards inside a computer performs functions such as RAM (random access memory), power supplies, modems and video "cards" also use the PCB as connection between others.

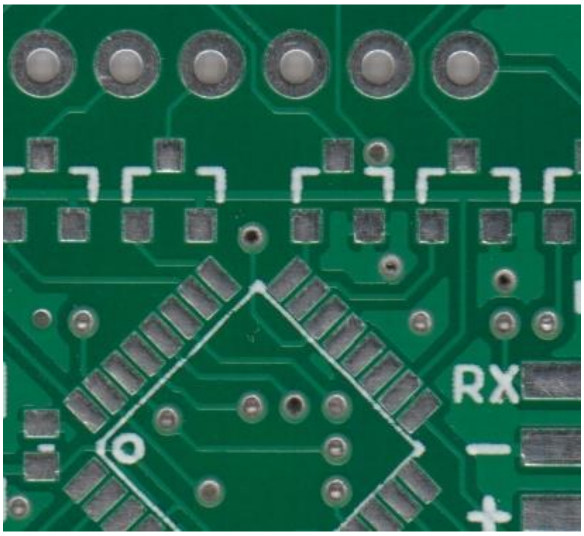


Figure 2.3: The green solder mask to the majority of the PCB

2.5 VIBRATION TEST FOR FATIGUE LIFE OF PBGA COMPONENT

Vibration test was used to check to time the failure for plastic ball grid array (PBGA) under a specific excitation. Besides that, this test also necessary to identify or check the stresses on the solder balls when conducting a fatigue life assessment of the components. From this analysis, both experimental and simulation was running and conduct to get the stress or fatigue affect to the PBGA component. Figure 2.3 shows the setup experimental of the vibration test into PBGA component. For this case, the PBGA component was assembly together with a printed wiring board (PWB) on the shaker with one of the two opposite edges clamped together while the other is kept free. Then, it was excited with a harmonic displacement of 131 Hz, that is, the first natural frequency of the test vehicle [7].

Furthermore, finite element analysis (FEA) is used for the stress analysis of the solder balls on the PBGA components to compare with the experimental value get from the vibration test. Figure 2.4 shows the constructed FEA model with commercial computer software ANSYS. The boundary condition was set for one of the two opposite edges are set as clamped and the other is left free to reflect the real edge conditions of the test vehicle.

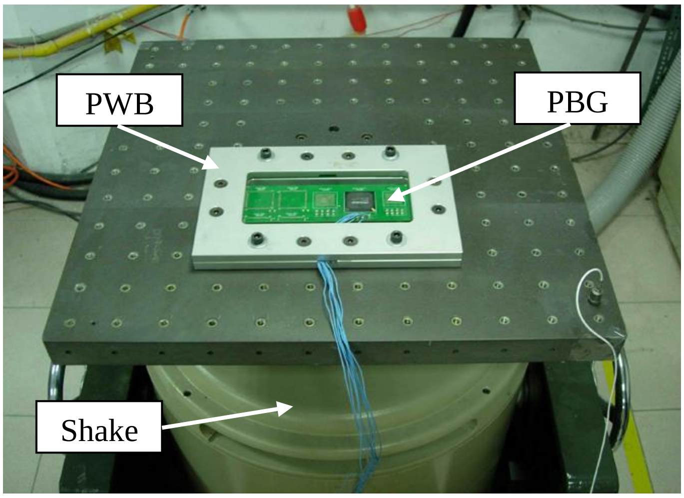


Figure 2.4: Test component mounted on the vibration shaker with a fixture.

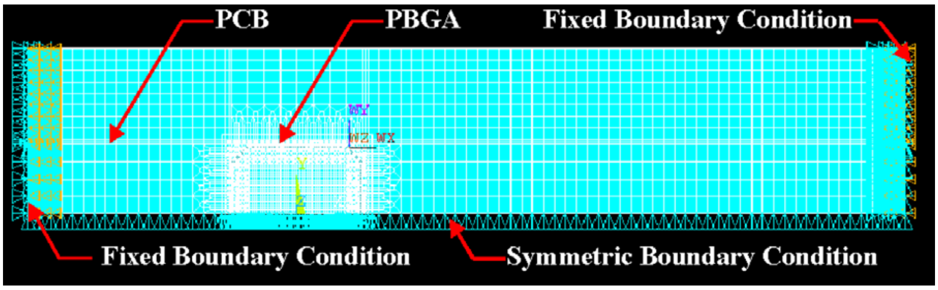


Figure 2.5: The FEA model with boundary condition.

2.6 ACCELERATED TEST METHOD

In the paper, three types of accelerated test methods based on vibration loadings are analysed. For the first type is fixed the frequency sine vibration and the second type is swept sine vibration within a narrow band of frequency. Lastly, the third type is swept random vibration within a narrow band of frequency. The PCB responses were recorded using a high-speed strain data acquisition system [8].

They conduct three types of vibration tests, fixed frequency sine vibration, narrow-band random vibration, and narrow-band swept sine vibration. Then they compared the loading intensities and features with the results of strain measurements. The failure processes are monitored by using a system of high-speed data acquisition. The failure feature and life data are analysed.

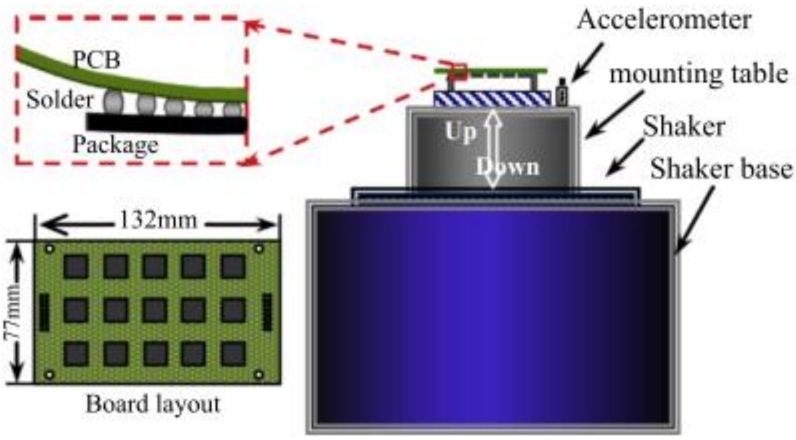


Figure 2.6: The vibration tester and PCB assembly layout

During a vibration test as shown in Figure 2.6, the shaker provides fast loads to the rigid base periodically. The loading acceleration and the frequency can be set as the testing parameters. It is observed that continuous and periodic acceleration will cause the rapid bending which similar to the situation in drop test. The failure of the board level interconnect is mainly dependent on maximum peeling stress which is determined by the amplitude and rate of PCB deformation. The failure cycle number could be predicted based on the maximum stress within the interconnect structure.

The vibrating stress amplitude is highly dependent on the frequency ratio. As it is varying, the fixed frequency vibration may miss the largest amplitude loading. The other two vibrations that are random and swept sine, within a narrow-band frequency could eliminate the influence from the boards. The differences of these methods are the loading density and repetitions. From the plot of solder fatigue cycle numbers show the characteristic life of the solder interconnects are verified with loading repetition evaluation. The failure processes of those three types of methods are similar, four failure stages can be found from failure process data. The first stage is a period without macroscopic crack, during which damage accumulated. The second stage is with crack fast propagations driven by high speed vibration loading. Another gradual crack increase stages were observed before the ultimate failure.

2.7 VIBRATION RESPOND

By using basic FEA tool to accurately investigate the dynamic characteristics of the PCB and avoid costly testing methods which require hardware. Here the normal modes and frequency response functions of PCB are determined and validated using vibration test on PCB. The validated model is used to predict vibration response for the PCB.

In this study, a six-layer of the PCB used for space applications has been be considered. The PCB is modelled as isotropic plate with equivalent material properties such as Young's modulus, Poisson's ratio and mass density. Finite Element model consists of 3364 quadrilateral shell elements. Young's modulus of elasticity for PCB is obtained as 15 GPa by conducting a three-point bending test for three different samples of PCB [9]. The finite element model of bare PCB is analysed and validated for two different boundary conditions that are shown in Figure 2.7 and 2.8. Figure 2.7 shows the PCB fixed or clamped at nine mounting locations and Figure 2.8 shows the PCB with fixed at eight mounting location.

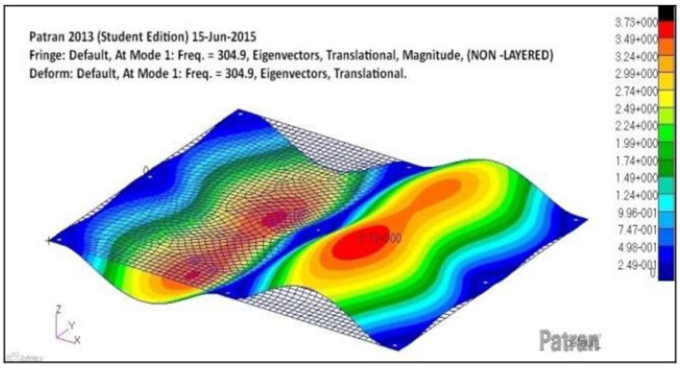


Figure 2.7: The PCB fixed or clamped at nine mounting locations

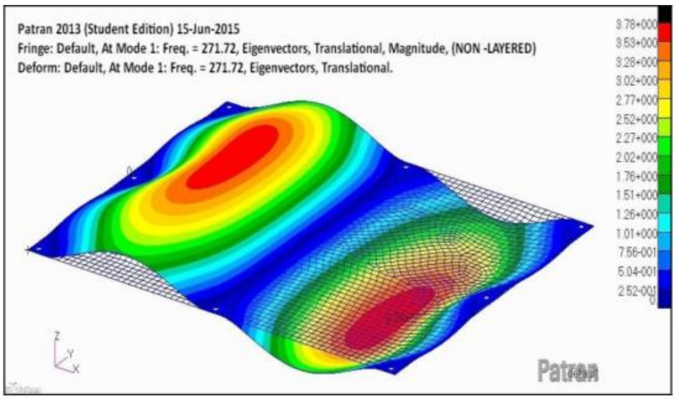


Figure 2.8: The PCB with fixed at eight mounting location

The FEM and experimental test results are compared. Analysis and test results for first few fundamental frequencies of the PCB at 9 mounting locations and for PCB at 8 mounting locations are compared. The random vibration response has reduced for 1.6 mm PCB versus 2.1 mm PCB [9]. This is very well evident from analysis and test results.

2.8 FATIGUE CRACK OF LEAD-FREE SOLDER JOINTS

Nowadays, all electronic and electrical component use solder joints as their connection to the board. The reliability issue for portable electronic devices such as mobile phones, MP3 players is solder joint failure caused by drop impact, vibration impact or others. Based on this impact will cause large stress amplitudes and rapid fatigue damage in the solder joints connecting chip packages to the circuit board. Figure 2.9 shows the scanning electron micrograph of a cross-section through the center of the failed solder joint. For the lead-free solder joint, the cracks or failure occur entirely within the bulk solder material [10]. From this picture, the crack or failure occurs at the corner of connection between copper pads and solder joint. So it can be observed that solder joint will affect if has any connection or intersect with them. Only 5 to 30 percent of the error which is attributed to material property variations in the solder, manufacturing variations in the resist opening diameter, and variations in the crack front from the assumed crack progression [10]. Furthermore, the crack at the circuit board interface is longer because the surface generates higher stresses in the solder joint at the board interface compare to package interface.

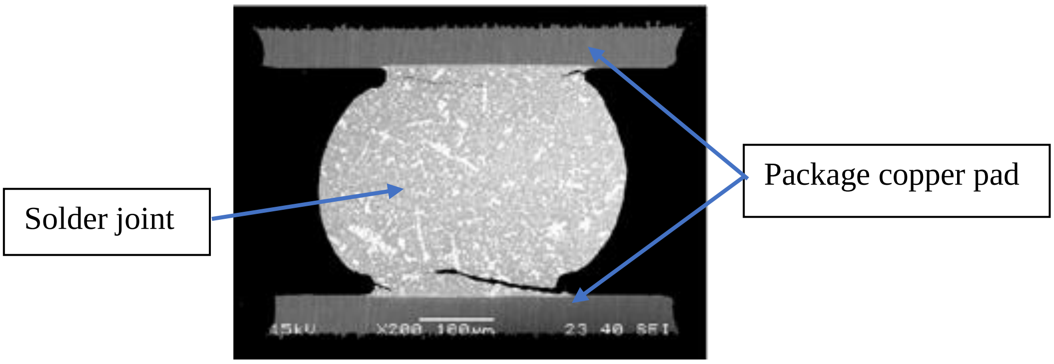


Figure 2.9: The scanning electron micrograph of cross-section through center of failed solder joint.

2.9 RELIABILITY OF ISOTROPICALLY TO ADHESIVE JOINT

The reliability of the isotropically conductive adhesive (ICA) solder joint was attached on liquid crystal polymer (LCP) substrate was compared to the reliability of lead-free solder attachments on the same substrate material [11]. Figure 2.10 shows the voltage or power has been used to achieve the failure of this component. Thus, it can be considered that at 989 cycles of real-time will occur the crack to the adhesive solder joint [11]. From Figure 2.11, the cracking of the chip resistors' solder joints was observed to occur through the solder bulk rather than at the intermetallic layer of the solder joint. Besides that, when using isotropically conductive paste, the most important is to test different adhesives, and pick the one best suitable to the equipment and application at hand. This is because some of the ICA occurs stress and failure state at the interface between component lead and the attachment material, different plating materials should be considered as well [11].

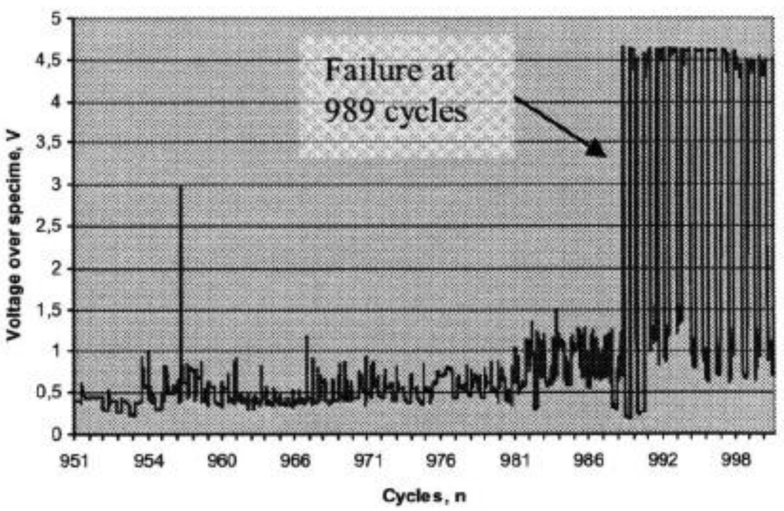


Figure 2.10: The real-time resistance measurement of failure solder joint.

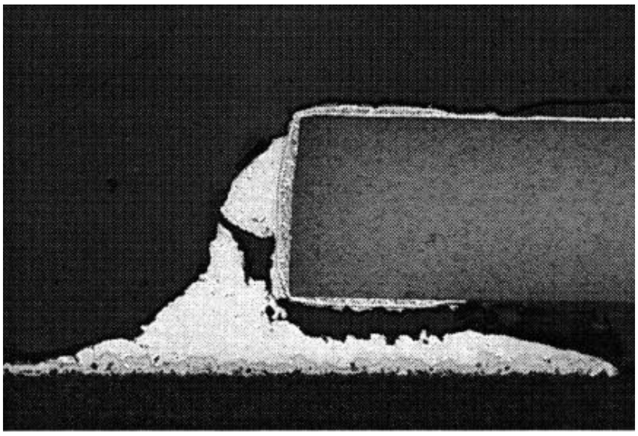


Figure 2.11: The crack and failure of the adhesive solder joint.

2.10 HARMONIC RESPONSE OF ADHESIVE JOINT ANALYSIS

Harmonic response usually was used to determine the steady state response of linear structure to load with respect of time, thus it enable to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations. Stress of the overall model has been used can be determined from harmonic response analysis. Figure 2.12 shows an example of the adhesive under harmonic response analysis with relative sensitivity depending on the frequency [12]. Usually, the harmonic response analysis is to determine the stress equivalent, displacement or relative sensitivity of the frequency response. From figure below the adhesive joint and adhesive mounting pad are second place of highest relative sensitivity of the harmonic response after ten-thousand (10,000 Hz) frequency [12].

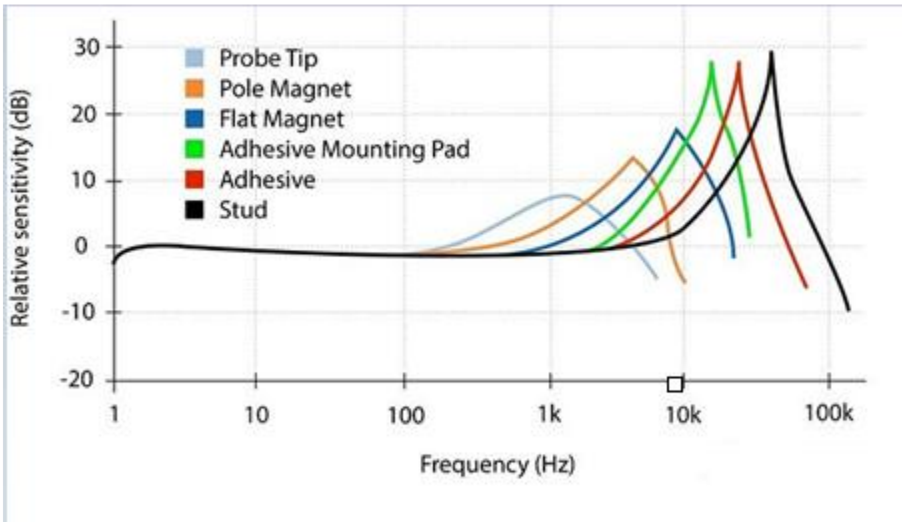


Figure 2.12: The Frequency response of accelerometers for different mounting

2.11 FINITE ELEMENT ANALYSIS

A limitation to evaluate directly the solder joint stresses since it is too small the finite element analysis was used to obtain for the fatigue estimation. There are four models that have been used in other research detailed FE models and smeared FE models. From the FES, ANSYS Simulation was used to get the detailed result of design. Detailed FE models, however, are not recommended since for a commercial case as the time required to build and solve such a model is excessive when simplified models produce data does not get more accurate data and result.

When doing simulation, the variation in mesh densities is applied in the model in order to examine the convergence of the analysed frequency results. Therefore, the large size of mesh used will be detailed for the point to relocate the frequency result. For verification of the FEA model, natural frequencies test vehicle are examined experimentally with a modal testing method [7]. From this FEA also, we can obtain the result of stress, strain, displacement, maximum or minimum load apply and so on based on statically basic simulation part.

CHAPTER 3

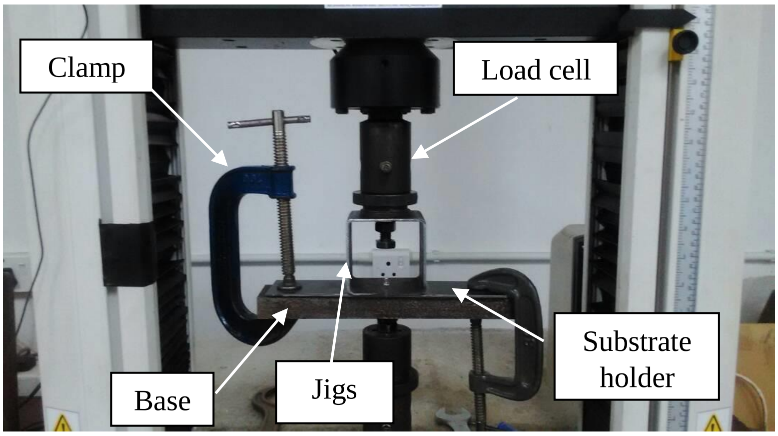
METHODOLOGY

3.1 OVERVIEW

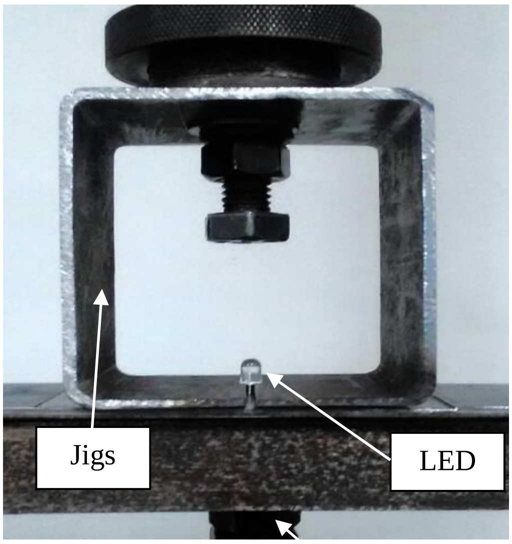
In this chapter, the method approach is discussed. The method to assess the structural, reliability and strength of the joint is presented in detailed for both experimental and simulation. Firstly, this chapter is discussed the development of experimental models and method conducting the experiment. Development of 3D CAD models in Solidworks 2018 and Analyses of structures, strength and reliability in ANSYS 19 is also have been characterized in detailed.

3.2 LED PULL STRENGTH TEST

In order to determine the optimum amount of adhesive, the LEDs were bonded to the circuit by using the adhesive with different build matrices and the bonding strength was evaluated using pull test. Each of the build matrices had 4 samples be tested and average readings were taken as final results. Universal Testing Machine (UTM) (INSTRON 3367) was used as the measurement instrument with additional support mechanisms in order to fulfil the requirement of the test. Special jig was custom made to fix the model on the UTM. Model was placed on the substrate holder, holding jig then placed on the model then clamped with the substrate holder, the holding jig have hole at the centre to allocate the led. Pull out jig was designed to connect with the UTM crosshead and it was attached below of the led lens flat spot. Led was pull at rate 1mm/minute until part of the model is pulled off. Placement of model with the jigs is shown in Figure 3.1 (a) and (b)



(a)



(b)

Figure 3.1: (a) The experiment setup for pull test measurement (INSTRON 3367) (b) Jigs attachment on the models (LED)

3.3 VIBRATION LOADING TEST

The same type of samples was used for the vibration test by using shaker machine. Each build matrix had two samples, one of them was positioned at horizontal plane and the other one was positioned at vertical plane. All of the samples were attached on custom design L-shaped base made of aluminium sheets with thickness of 5 mm as illustrated in Figure 3.2 and run at the same time under swept random vibration with frequency range of 3 Hz - 500 Hz for 4 hours by using shaker. The natural frequency of the structure was 477 Hz which determined prior to the vibration test by using modal testing method. The bonding between the LEDs' joints and the circuit were observed at every time interval of 30 minutes. The samples were evaluated again by connecting the power supply to test the electrical conductivity in terms of current variation before and after the vibration loading. The bonding was also further inspected by capturing the bonding image using Hirox Digital Microscope (KH-3000) and scanning the bonding using X-ray machine (XT V 160) to detect any failure such as cracks in the adhesive.

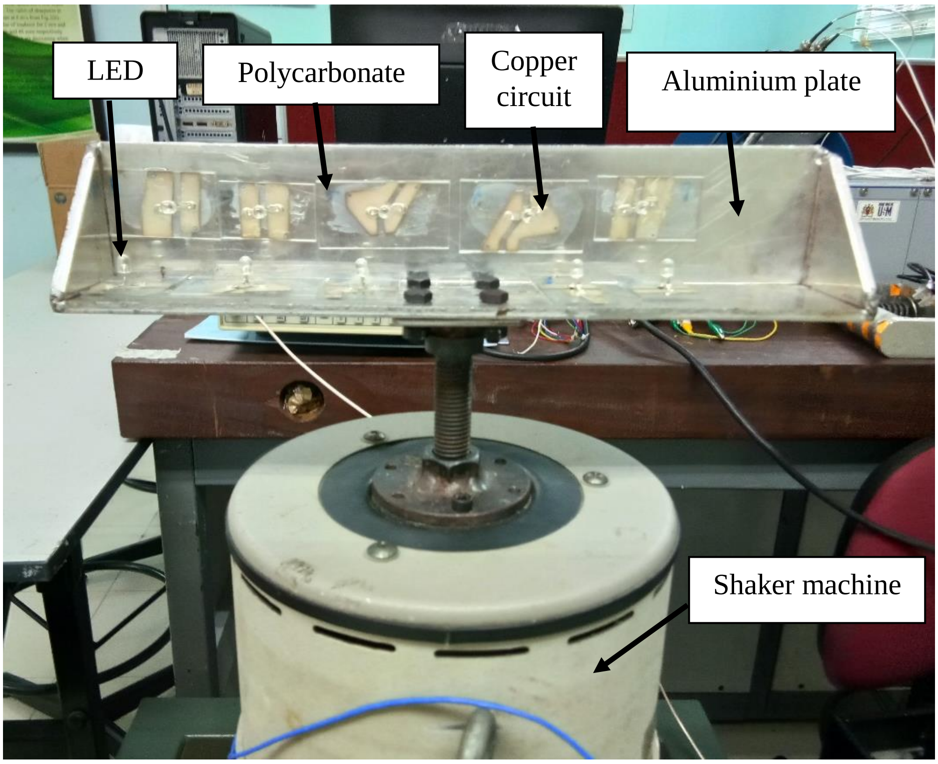


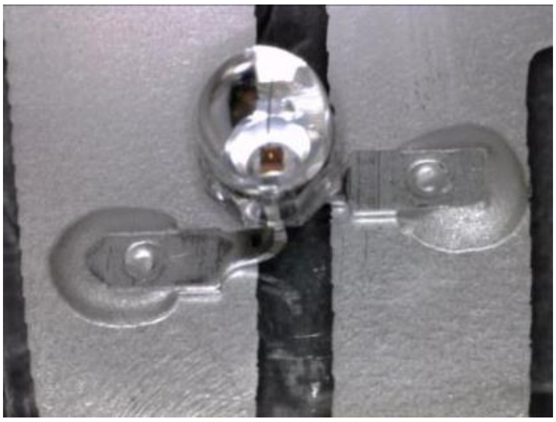
Figure 3.2: The vibration test setup for LEDs joints.

3.4 EXPERIMENTAL MODEL

In the LED pull strength test and vibration test experiment, four different models were built. An electronic component, led was assembled to a simple circuit. The circuit base was built with Lexan 8010 polycarbonate film, the polycarbonate base was padded with the silver layer as an electrical connectivity circuit. Each matrix provided different strength to hold the LED's joints against any force or vibration exerted on the LED. Non-lead solder paste, silver epoxy was used as solder joint with each different dispensing methods. The distinguished bonding of solder joint and the LEDs for a four build are shown in table 3.1

Table 3.1 Dispensing method for four model.

Model	Dispensing method
 Model 1	Led was placed on the 3 dot of solder paste that had been dispensed on the circuit, then one dot of solder paste was dispensed on the edge of led lead base
 Model 2	Led was placed on the dispensing 3 dots of the adhesive close in straight line on the circuit but with reduction number of dots in straight line one by one.
 Model 3	Led was placed on the circuit and 3 dots of solder paste were dispensed at each sides of the led lead base. Tip of dispensing needle was moved and followed LED joint frame while dispensing the adhesive continuously

 Model 4	Led was placed on the circuit and 2 dots of solder paste were dispensed at each sides of the led lead base.

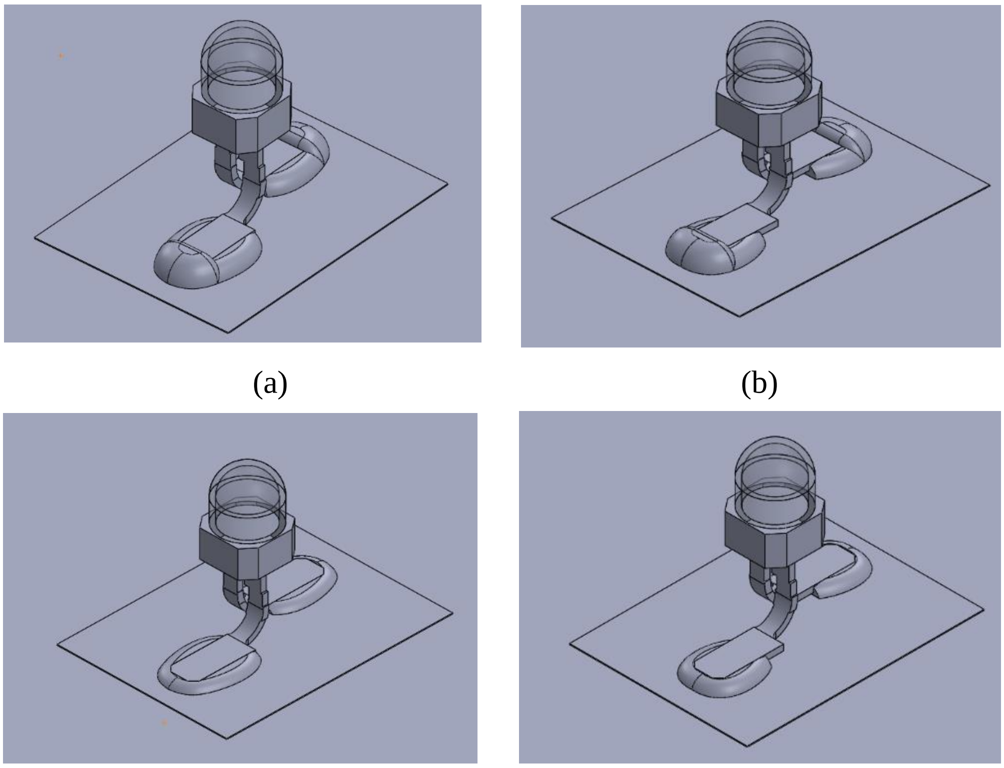
3.5 GEOMETRY MODEL

In a conventional mechanical designing process, a geometry model of the design concept is created in a CAD software such SolidWork and then exported to mesh generator or pre-processing software to create a finite element mesh such ANSYS. Thus, several CAD models as developed based on the assembly of prototype model that had been used by previous researcher. The base model was developed by referring to the actual component specifications. Note that the additional parts dimensions such as solder joints features are estimated as it was not a geometry structure. So it has been approximately developed with respect to the actual shape. Whereas, models of the copper pad circuit and polycarbonate base was modified to cater the needs of simulation as shown in Table 3.2. The component is assembled as in the experimental prototype models, Figure 3.3 shows the both assembled 3D CAD model of led with solder joint and Figure 3.4 shows the complete assembly model.

Table 3.2 Dimension for each parts of geometry model

Parts	Dimensions (mm)
Base of the lead	5.5×2.5×0.5
Solder joint model 1	6.5×3.5×1.2
Solder joint model 2	5.0×3.5×1.2
Solder joint model 3	6.5×3.5×0.5
Solder joint model 4	5.0×3.5×0.5

Parts	Dimensions (mm)
Copper pad circuit	20×15×0.08
Polycarbonate base	150×150×1



(a) (b) (c) (d)

Figure 3.3: The assembled 3D CAD model of led with solder joint (a) model 1 (b) model 2 (c) model 3 (d) model 4

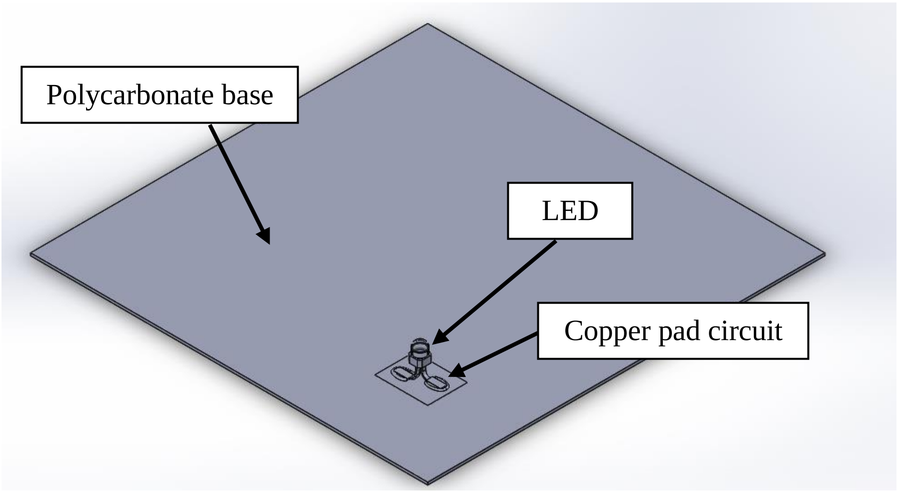


Figure 3.4: Complete assembly model

3.6 MATERIAL PROPERTIES

Materials applied were based on the material used in the experimental prototype model. Since complete material properties for the experimental prototype cannot be determined, the materials in the simulation were modified to cater to the simulation needs. Materials' mechanical properties were provided from the ANSYS materials library and manufacturer sources that not exactly the same as in the experiment. All material was assumed to be non-linear elastic-plastic materials. Materials' mechanical properties of the parts are summarized in Table 3.3.

Table 3.3: Materials' mechanical properties of the parts

Part	Materials	Density (kg/m^3)	Young's modulus (MPa)	Poisson ratio	Yield strength (MPa)	Tangent modulus (MPa)	Ultimate tensile strength (MPa)
Copper pad circuit	copper [13]	8900	130000	0.34	120	125	210
Polycarbonate base	Polycarbonate [14]	1200	2506	0.38	63	0.05	65
Solder joint	silver epoxy [15]	4500	4140	0.32	24.1	38.62	34.5
Lead	copper alloy [13]	8300	110000	0.34	280	1150	491

3.7 MESHING

Meshing is a process that influences the time consuming for a solution, accuracy, convergence, and time running to consume for a solution. Generating a conformal and structured grid mesh model is needed to ensure the multi-parts model mesh is connected to allow analysis running. Conformal meshing is a match meshing nodes at the different edges and faces. Otherwise, the non-conformal mesh is developed. Due to the models developed are multi parts which need to locally modified and contains a complex geometry, unstructured grid, and non-conformal mesh was developed. In the simulation, the conforming mesh will