STRUCTURAL AND RELIABILITY ANALYSIS OF SOLDER JOINT UNDER VIBRATION LOADING

By:

MUHAMMAD IKMAL BIN ILIAS

(Matric number: 123117)

Supervisor:

Dr. Abdullah Aziz Saad

May 2018

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 candidature for any degree.

Signed...
(MUHAMMAD IKMAL BIN ILIAS)

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 IKMAL BIN ILIAS)

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 IKMAL BIN ILIAS)

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.

TABLE OF CONTENTS

DECLARATION i

ACKNOWLEDGEMENTS ii

TABLE OF CONTENTS iii

LIST OF FIGURES v

LIST OF TABLES vi

LIST OF ABBREVIATION vii

ABSTRAK viii

ABSTRACT ix

CHAPTER 1: INTRODUCTION 1

- 1.1. Background 1
- 1.2. Problem statement 2
- 1.3. Objectives 2
- 1.4. Scope of works 3

CHAPTER 2: LITERATURE REVIEW 4

- 2.1 Introduction 4
- 2.2 Solder materials 4
- 2.3 Vibration testing 5
- 2.4 Accelerated test methods 6
- 2.5 Vibration response 7

CHAPTER 3: METHODOLOGY 9

- 3.1 Introduction 9
- 3.2 Computer-aided Design (CAD) Model 9
  - 3.2.1 LEDs build matrices 9
- 3.3 Finite Element Analysis (FEA) 12
  - 3.3.1 Materials 12
  - 3.3.2 Meshing 12
- 3.4 Pull strength test simulation 15
  - 3.4.1 Boundary condition 15
- 3.5 Random vibration simulation 16
  - 3.5.1 Modal analysis 16
  - 3.5.2 Random vibration analysis 17

CHAPTER 4: RESULT & DISCUSSION 19

- 4.1 Computer-aided Design (CAD) Model 19
- 4.1 Static structural analysis 19

4.1.1 Configurations..19
4.1.1 Pull strength test simulations..20
4.2 Random vibration..24
4.2.1 Modal analysis..24
4.2.2 Random vibration analysis..25
CHAPTER 5: CONCLUSION..31
5.1 Overview..31
5.2 Recommendation for future works..32
REFERENCES..33
APPENDICES..35
APPENDIX A..36
APPENDIX B..42
APPENDIX C..44

LIST OF FIGURES

Figure 2.1 Vibration tester and PCB assembly layout[9] 6

Figure 2.2 Mode shape of PCB with 9 mounting locations[10] 8

Figure 2.3 Mode shape of PCB with 8 mounting locations[10] 8

Figure 3.1 The CAD model with a dispensed adhesive dot 10

Figure 3.2 The CAD model with two dispensed adhesive dots 10

Figure 3.3 The CAD model with three dispensed adhesive dots 11

Figure 3.4 Complete assembled model 11

Figure 3.5 Two types of mesh (a) conforming mesh, (b) non-conforming mesh 13

Figure 3.6 Refinement features on the surface of solder joint 13

Figure 3.7 Edge sizing feature on the edges of solder joint 14

Figure 3.8 (a) Close-up look of the mesh (b) Mesh of the whole model 14

Figure 3.9 Fixed support feature onto bottom surface of base 15

Figure 3.10 Displacement feature onto LED 16

Figure 3.11 A fixed support applied on the model for modal analysis 17

Figure 3.12 Power Spectral Density input curves 18

Figure 4.1 Free body diagram (FBD) of the model 20

Figure 4.2 Overall equivalent stress occurred on (a) Model 1 (b) Model 2 (c) Model 3 21

Figure 4.3 Maximum equivalent stress occurred at solder joint for (a) Model 1 (b) Model 2 (c) Model 3 23

Figure 4.4 Maximum equivalent stress contour plot on circuit pad for (a) Model 3 (b) Model 2 (c) Model 1 27

Figure 4.5 Maximum equivalent stress contour plot at solder joint for (a)Model 3 (b)Model 2 (c)Model 1 28

Figure 4.6 Maximum equivalent stress contour plot on LED for (a)Model 3 (b)Model 2 (c)Model 1.................... 29

LIST OF TABLES

Table 2.1 The PCB properties 7

Table 3.1 Dimension for parts of geometry model 11

Table 3.2 Mechanical properties of materials 12

Table 3.3 Mesh independent study 15

Table 3.4 Power Spectral Density input values 18

Table 4.1 Comparison of configurations 19

Table 4.2 Maximum equivalent stress for each model for all bodies 22

Table 4.3 Maximum equivalent stress for solder joint of each model 24

Table 4.4 Maximum equivalent stress value at copper pad circuit 29

Table 4.5 Maximum equivalent stress value at solder joint 30

Table 4.6 Maximum equivalent stress value at LED joint 30

LIST OF ABBREVIATION

PCB	Printed Circuit Board
LED	Light Emitting Diode
FEM	Finite Element Method
FEA	Finite Element Analysis
CAD	Computer-aided Design
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 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 load. 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 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 at certain parameters. Overall, the research done allows the critical parameters of the models to be discovered of its behaviour in this analysis.

CHAPTER 1

INTRODUCTION

1.1. Background

In manufacturing process, electronic systems are often required to operate in severe vibration environments for extended periods without failing. Some instances are in automobiles, airplanes, fossil fuel power plants, communication systems, light and heavy manufacturing and others. The role of solder in electronic packages has expanded. In advanced designs, solder is an electrical interconnection as well as a mechanical bond, and must often serve as a thermal conduit to remove heat from joined devices. Moreover, interconnects become more critical as die size, chip-carrier size, and number of inputs/ outputs increases, while solder joint size and cost decreases. Electronic assemblies, such as television sets and radios may not operate in vibration environments, but they need to survive vibration when they are being transported from the manufacturer directly to the consumer in various types of shipping crates.

In a worst-case situation these stresses may cause one of the following failure modes, PCB delamination, solder joint fracture, lead fracture or component package fracture, if any single one of these modes occurred total failure would very probably follow[1]. As currently electronic devices are widely applied in automobiles application, the reliability of solder joint under vibration loading is also an issue in this study as its leads to solder joint failure.

Field failures in electronic equipment hardware compiled by the United States 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. It is also stated that this condition is due to handling, vibration loading, shock and thermal cycling. About 55% of the failures are due to high temperature and temperature cycling, while about 20% of the failures are connected to cases of vibration and shock. Then, there is also failures around 20% are due to humidity effects.[2]

Solders paste undergo creep to relax imposed stresses. Creep damage accumulates in the solder rather than in the more brittle components to which it is attached. Furthermore, the damage mechanism during thermomechanical fatigue is similar to that during creep deformation[3]. The most insidious environments for solder degradation are high temperature aging and thermomechanical fatigue because solders are microstructurally unstable and evolve with strain, temperature, and time.

1.2. Problem statement

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. It is probably safe to say that all electronic equipment will be subjected to some type of vibration at some time in its life.

Vibrations encountered during transportation and handling can produce many different types of failures in electronic equipment unless the proper consideration are given to the mechanical design of the electronic structure and the shipping containers[2]. Hence, the project aims **to study the structural and reliability of solder joint under vibration loading**. The study will be conducted mainly through ANSYS static structural and finite element simulation.

1.3. Objectives

The purposes of this research are:

1. To analyse solder joint under vibration condition using a finite element analysis.
2. To study the solder joint reliability under vibration condition
3. To compare the result obtained and to establish result of the analysis

1.4. Scope of works

Primarily, finite element analysis along with ANSYS static structural are used to determine the solder joint stresses/strain. Modified models will be analysed in structural analysis to analyse the solder joint structure and to identify the strength of the joint.

1. The LED is assembled on stretchable circuit. Then, it is analysed in random vibration to investigate the structure and failure occurred in the model.
2. An experimental is done to check the natural frequency of interconnection which is under vibration analysis.
3. Next, the assembly is put through under simulation to further investigate the stress level of the interconnector.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Solder joint fatigue and failure in semiconductor-based device that undergo vibration loading is one of significant concerns for automotive industry, and portable devices. Numerous studies and research has been keen to predicting the solder joint reliability in electronic devices under mechanical environment.

The finite element method (FEM) is widely used in industry as well as research for solving the problems arising in the engineering analysis[4]. In conducting experimental analysis on the prototype model of the LED fused to circuit pad, certain techniques and simulation are used to examine the dynamic response of the electronic device which subjected to vibration loading. Hence an in-depth study related this research was done from several literature published in this area to comprehend the manufacturing process of surface mount technology in the direction of addressing the reliability concerns.

2.2 Solder materials

Three types of solder materials commonly used in electronic packaging currently, lead solder and other two alternatives, lead-free solder and conductive adhesive. Lead solder are made from the composition of Lead and Tin which most commonly used leadfree solder in industry Sn63Pb37. Lead solder is easier to work with as it requires low temperature and less issues regarding to its quality. However due to restriction in law and new requirements for non-toxic materials, lead free solder and conductive adhesive are used as alternative [5].

Lead free solder is a Tin based alloy, common composition of lead free solder used are Sn-Ag, Sn-Cu, Sn-Zn and Sn-Ag-Cu. Instead, as alternative of lead free solder, it is being used to provide higher melting point or to satisfy the component and circuit requirement [6]. Conductive adhesives are a composite of adhesive epoxy and a conductive metal such as silver, copper, tin oxide or indium.

Concept of the composition is that soft material epoxy filler promotes good contact within and other particles as it deforms and shrink during curing process. Most common material is as the epoxy filler is silver as it is a good conductor, easy to available and has moderate cost. Conductive adhesive is applied as the solder joint by using same method as the solder paste, where it is dispensed to the circuit before placement of 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 [7].

2.3 Vibration testing

Vibration testing is the study of structure's response while exposed to a specific dynamic environment, the environment simulates in a reasonable manner to ensure that the structure will either survive or function properly when exposed to the dynamic environment under field conditions.

There are 2 types of vibration testing that have been used in the previous study for solder joint reliability analysis, sinusoidal vibration and random vibration. Random vibration is the actual phenomenon in everyday life it is not repetitive or predictable whereas sinusoidal vibration focuses upon a single frequency at any one time. (Jang et al. 2016) conducted forced vibration experiment to verify the position of the most stress concentration in FEA and to measure number of cycles to failure under various vibration amplitudes. The model was excited by magnet shaker with a sweeping sine signal around the first resonant frequency until model failed. The sweep range scanning test rang used was between 1400Hz to 1500Hz to determine the first order natural frequency.

In a research vibration test is performed with constant G-level and varying G-level input excitation. Model assembly subjected to sinusoidal vibration loading using electrodynamics shaker and vibration control test facility. The sweep frequency scanning test was from 20Hz to 1000Hz. The first order natural frequency was determined by the scanning test[8].

2.4 Accelerated test methods

In the paper published by (Liu et al, 2014), three types of accelerated test methods based on vibration loadings are conducted and compared for board level mechanical reliability evaluation. The first type is fixed frequency sine vibration. The second type is swept sine vibration within a narrow-band of frequency. And 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.

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.

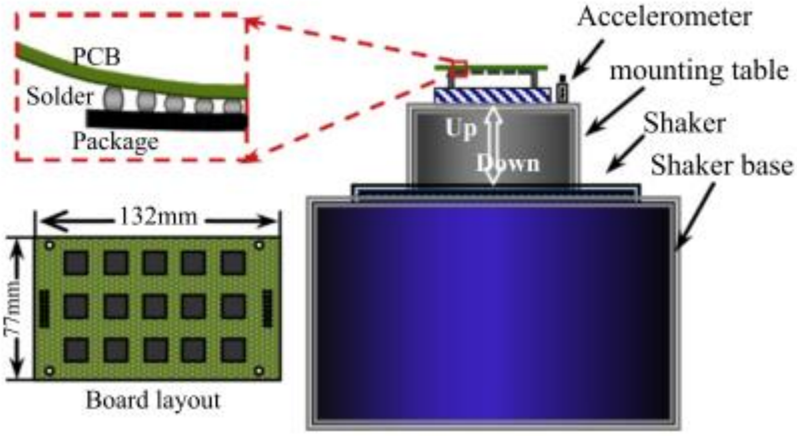


Figure 2.1 Vibration tester and PCB assembly layout[9]

During a vibration test as shown in Figure 2.1, 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 stress induced damage accumulates in each cycle. The failure cycle number could be predicted based on the maximum stress within the interconnect structure.

The vibrating amplitude is highly dependent upon 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.

2.5 Vibration response

In a research by (V.N. Somashekar et al.), in 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 random vibration input. It is shown here how the responses are accurately predicted for random vibration input for a design parameter variation of PCB. The results are also validated using vibration test on PCB.

In this study, a six-layer PCB used for space applications is 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.

Table 2.1 The PCB properties[10]

PARAMETER	**VALUES**
PCB size	250×201×2.1 mm
Density of Bare PCB	1985 Kg/m^3
Young’s modulus of Bare PCB	15 GPa
Poisson’s ratio	0.12
Boundary Condition	Fixed/clamped

The finite element model of bare PCB is analysed and validated for two different boundary conditions that are:

(i) PCB fixed/clamped at nine mounting locations shown in Figure 2.2 and

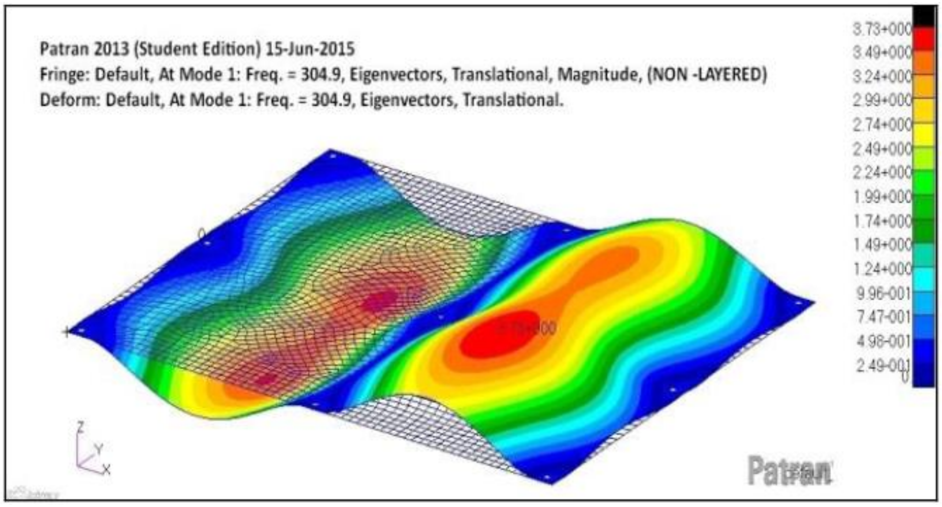


Figure 2.2 Mode shape of PCB with 9 mounting locations[10]

(ii) PCB with fixed at eight mounting locations shown in Figure 2.3

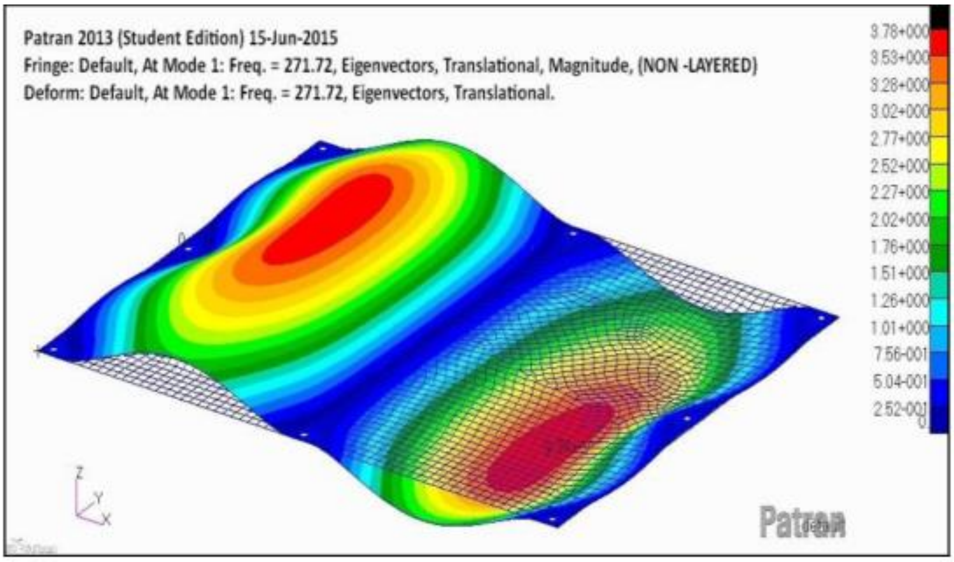


Figure 2.3 Mode shape of PCB with 8 mounting locations[10]

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 analysis and test results for both the cases are matching well. The random vibration response (overall Grms, which is a qualitative measure of intensity of vibration) has reduced for 1.6 mm PCB versus 2.1 mm PCB. This is very well evident from analysis and test results.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In the methodology, the methods used to approach the objective are discussed. It starts with the modelling of the LEDs after assembled on the circuit by using Solidwork v2016. Then, the models undergo several finite element analyses and were discussed. The procedures to assess the behaviour, reliability and strength of the solder joint are mentioned and shown when using finite element analysis. Lastly, the summarisation of this chapter is concluded.

3.2 Computer-aided Design (CAD) 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. The final assembled model of LED and solder joints are as shown below.

3.2.1 LEDs build matrices

The distinguished bonding of solder pastes and the LEDs joints for a three build are displayed and shown in Figure 3.1, 3.2, and 3.3. The first model was built by referring to the actual experimental build which was dispensed with three dots of the adhesive closely in straight line on the circuit plus another dot of adhesive on top of the LED joint. The procedure was repeated for the remaining two models of the build. However, the number of adhesive placed onto the circuit are in reduction of a dot for respective matrix.

For second model, the amount of adhesive put on the circuit are two dots only with the addition of a dot on top of the LED joint. Next, only a dot of adhesive is dispensed for the third model of the matrix with similar one dot on top of the joint. Table 3.1 shows the dimension for each parts of the geometry model.

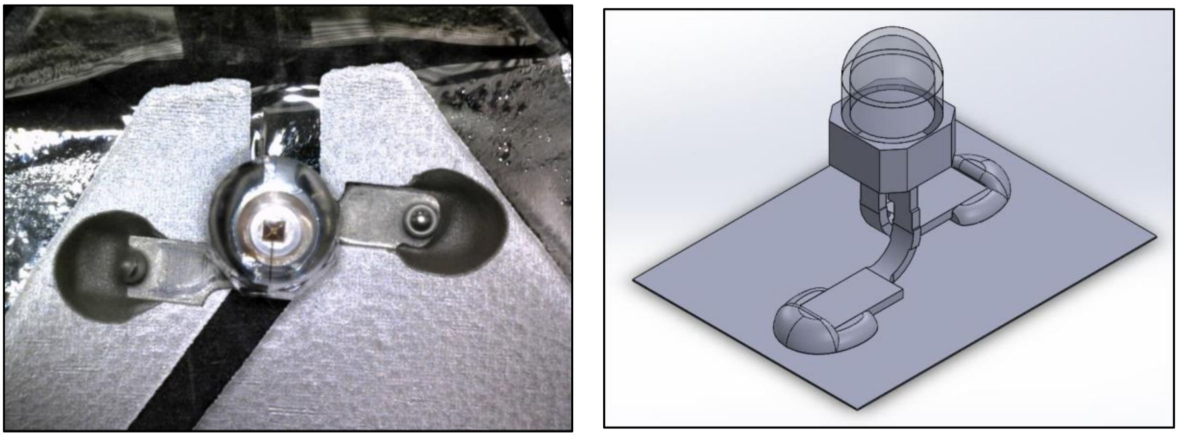


Figure 3.1 The CAD model with a dispensed adhesive dot

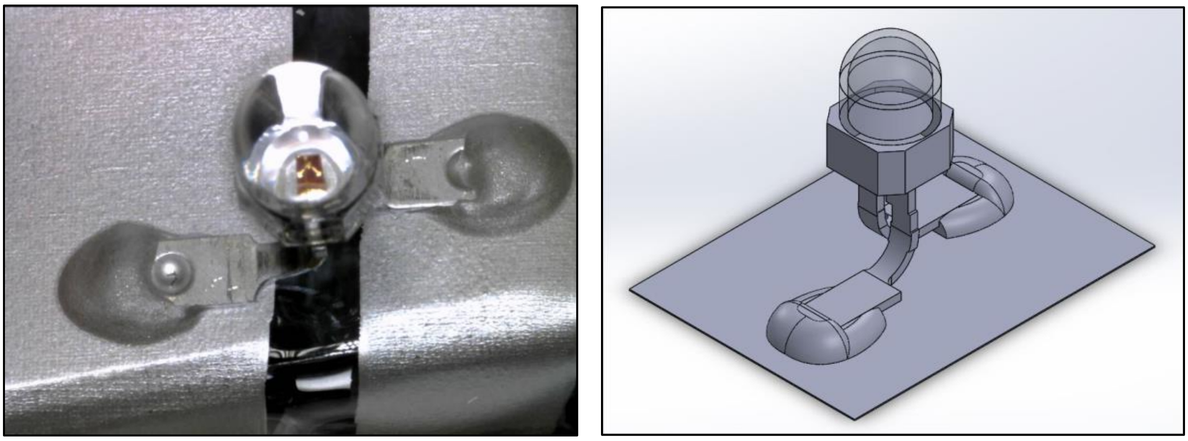


Figure 3.2 The CAD model with two dispensed adhesive dots

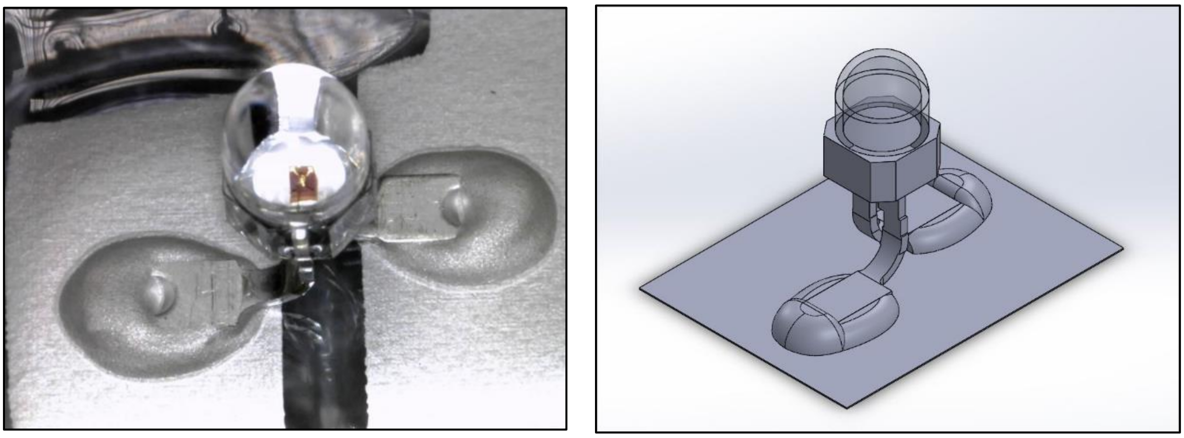


Figure 3.3 The CAD model with three dispensed adhesive dots

Table 3.1 Dimension for parts of geometry model

Parts	Dimensions (mm)
Base of the lead	5.5*2.5*0.5
Solder joint model 1	4.3*4.0*1.19
Solder joint model 2	5.0*4.0*1.45
Solder joint model 3	6.5*4.0*1.45
Copper pad circuit	20*18*0.08
Polycarbonate base	150*150*1.0

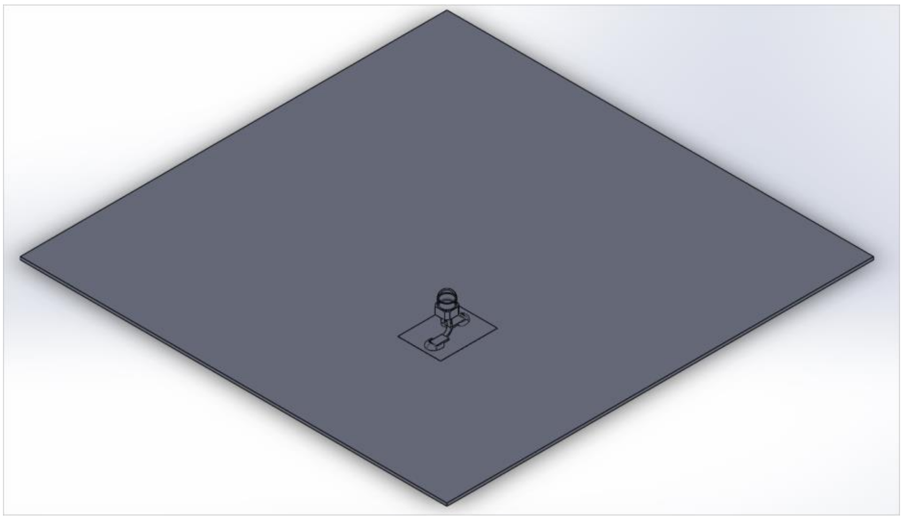


Figure 3.4 Complete assembled model

3.3 Finite Element Analysis (FEA)

After modelling process was done, all models of build matrices undergo finite element analyses in ANSYS v16.

3.3.1 Materials

The materials applied in the CAD models were based on the experimental model used by previous researcher[11]. Analyses are done to each type of adhesives configurations. Material properties were provided from the ANSYS materials library and manufacturer sources. Some adaptations were made to cater to the simulation settings. Note that the materials applied were assumed to be non-linear elastic-plastic materials. Materials' mechanical properties of the parts and adhesive properties are summarized in Table 3.2 below.

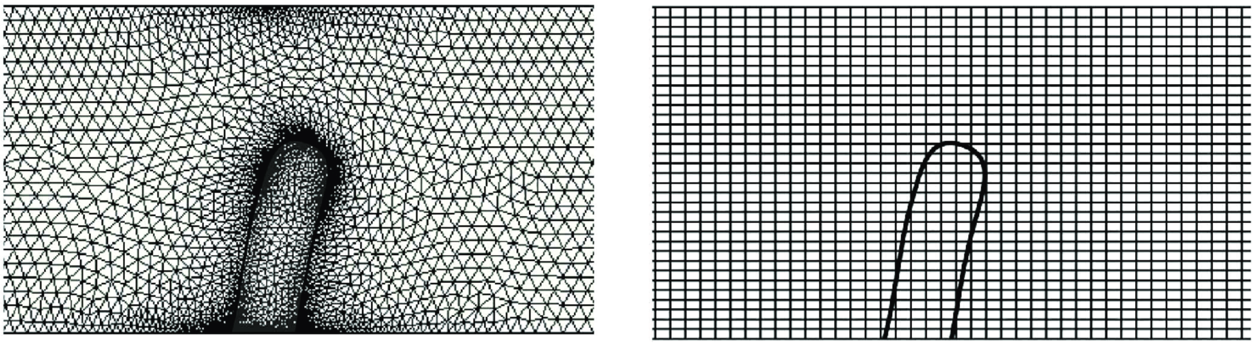
Table 3.2 Mechanical properties of materials[12]

Parts	Materials	Density (kg/m^3)	Young's modulus (MPa)	Poisson ratio	Yield strength (MPa)	Tangent modulus (MPa)	Ultimate tensile strength (MPa)
LED	Copper alloy	8300	110000	0.34	280	1150	491
Solder paste	Silver epoxy	4500	4140	0.32	24.1	38.62	34.5
Copper pad circuit	Copper	8900	130000	0.34	120	125	210
Polycarbonate base	Polycarbonate	1200	2506	0.38	63	0.05	65

3.3.2 Meshing

In finite element analysis, mesh feature is a process that influences the time consuming for solution, accuracy, convergence, and CPU memory consume for solution. This is because all the analysis that will be derived from the meshing process done to the model. Finding the best mesh on a specific model require trial and error processes.

There are two types of mesh available to be used onto the geometry model shown in Figure 3.5. The mesh applied to the assembled model is conforming mesh as there is a matching of nodes at the interface.



(a) (b)

Figure 3.5 Two types of mesh (a) conforming mesh, (b) non-conforming mesh

In simulation, conforming mesh will allow every node on one side of the interface can be matched with a node on the other side of the interface with a very low tolerance. This method is used so there is no additional interpolation required at a conformal interface. It will allow the computation in ANSYS faster and more accurate[13].

Refinement feature in Figure 3.6 and Edge Sizing feature in Figure 3.7 were added into the mesh instead of using automatic meshing. The refinement type is 1 and the number of division of the edge sizing is 10.

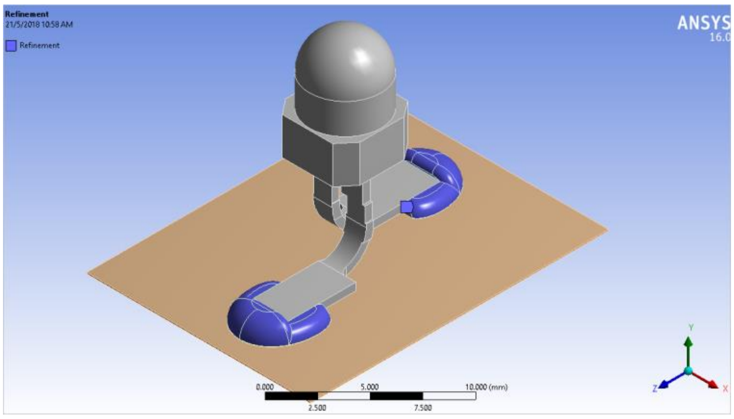


Figure 3.6 Refinement features on the surface of solder joint

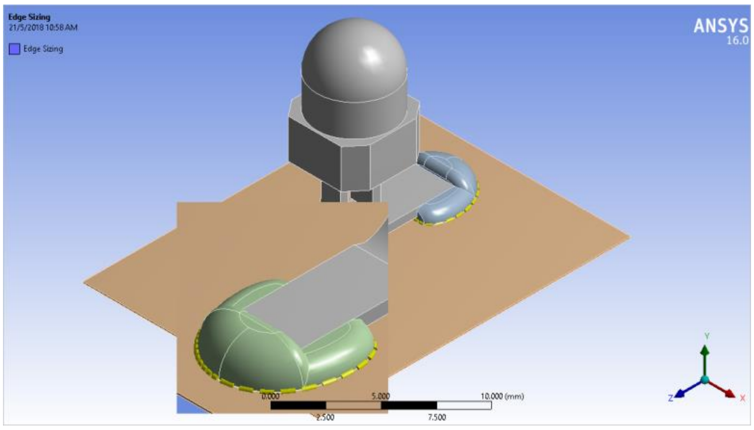
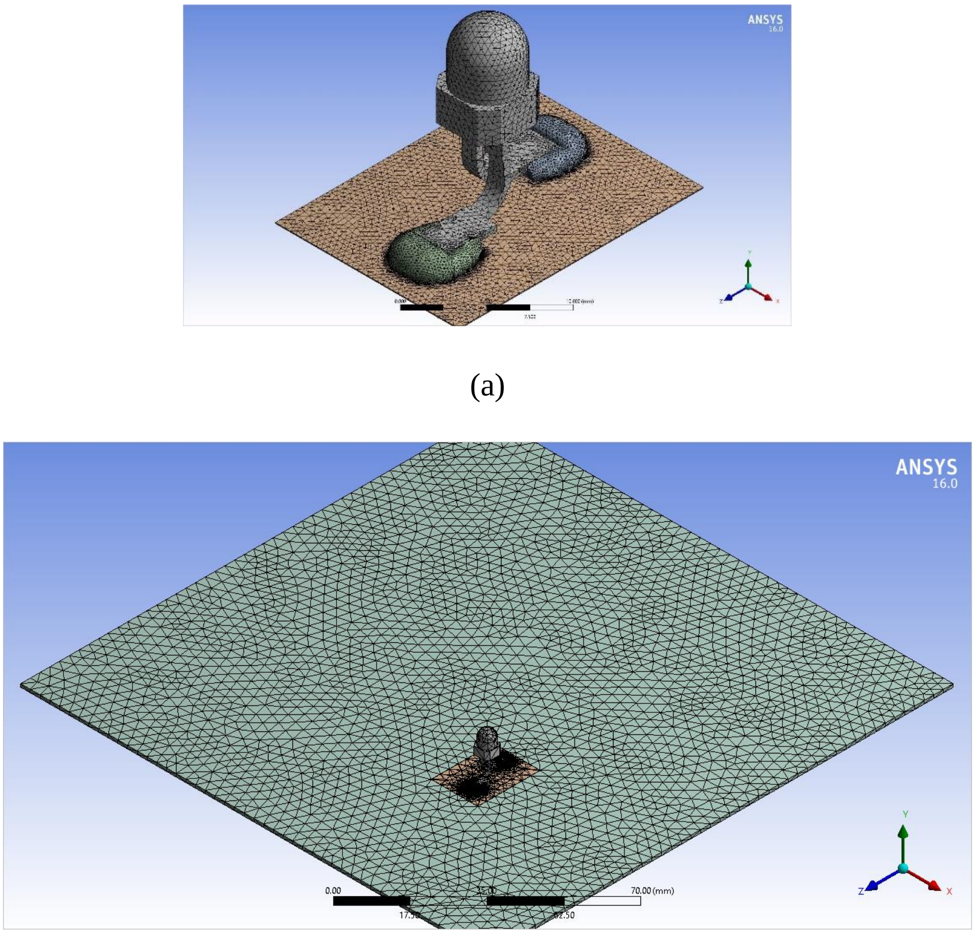


Figure 3.7 Edge sizing feature on the edges of solder joint

These features were added to decrease the size of the mesh generated. It will allow more accurate calculation in the simulation. Figure 3.8 below shows the final mesh applied onto the models.



(a)

(b)

Figure 3.8 (a) Close-up look of the mesh (b) Mesh of the whole model

A mesh independent study was done to generate the optimum mesh which will provide the best analysis. Table 3.3 show the mesh independent study done from various setup in ANSYS. The deviations produced from each setup are compared. Configuration or setup which made the smallest deviation is chose.

Table 3.3 Mesh independent study

	Setup 1	Setup 2	Setup 3	Setup 4 (Datum)
Relevance centre	Coarse	Medium	Fine	Fine
Smoothing	Medium	Medium	Medium	High
Elements	13498	24862	45 445	46 115

3.4 Pull strength test simulation

To simulate the actual pull strength test conducted by previous researcher, a structural analysis was conducted in ANSYS v16. It is done to analyse the soldier joint structural and its strength. The analysis was conducted for all models and with different solder joint material.

3.4.1 Boundary condition

Only two boundary conditions were used in this simulation that are fixed support and displacement. A fixed support feature was inserted into the simulation to simulate the base support of the assembly in the actual test. Fixed support was applied to the bottom of the base as shown in Figure 3.9.

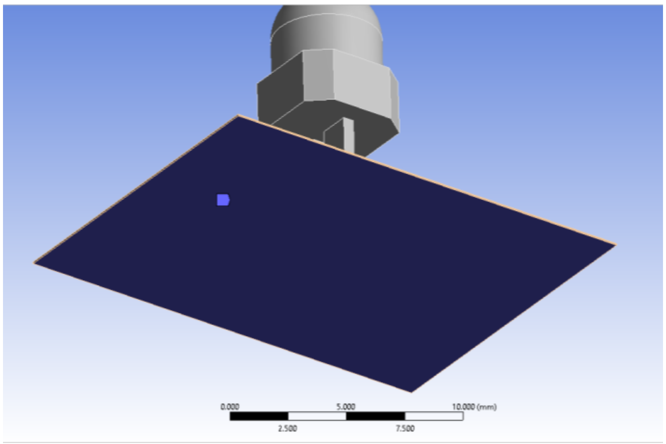


Figure 3.9 Fixed support feature onto bottom surface of base

A displacement feature at Y-axis direction for 1mm/minutes was applied at the bottom surface of the LED head part. This feature was added to indicate the pull force in the actual pull strength test as shown in Figure 3.10.

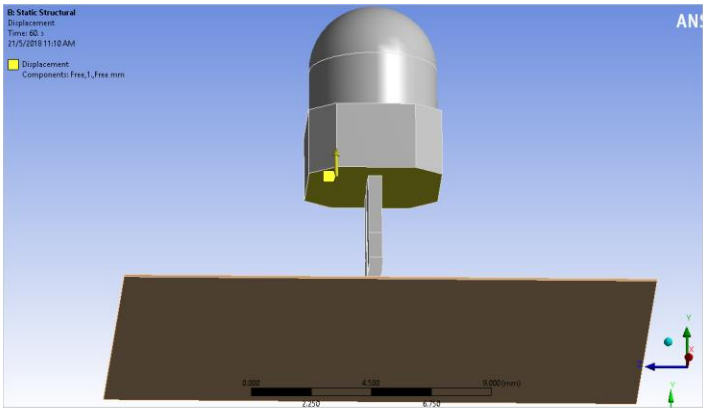


Figure 3.10 Displacement feature onto LED

3.5 Random vibration simulation

Random vibration analysis was done in ANSYS v16 to analyse the failure occurs on the models under random vibration loading.

3.5.1 Modal analysis

Before random vibration analysis was conducted, CAD models need to be analysed in modal analysis to determine the dynamic response. In this analysis, only Fixed Support was applied as boundary condition unlike pull strength test simulation. The surface of the bottom of the polycarbonate base is selected for fixed support as shown in Figure 3.11.

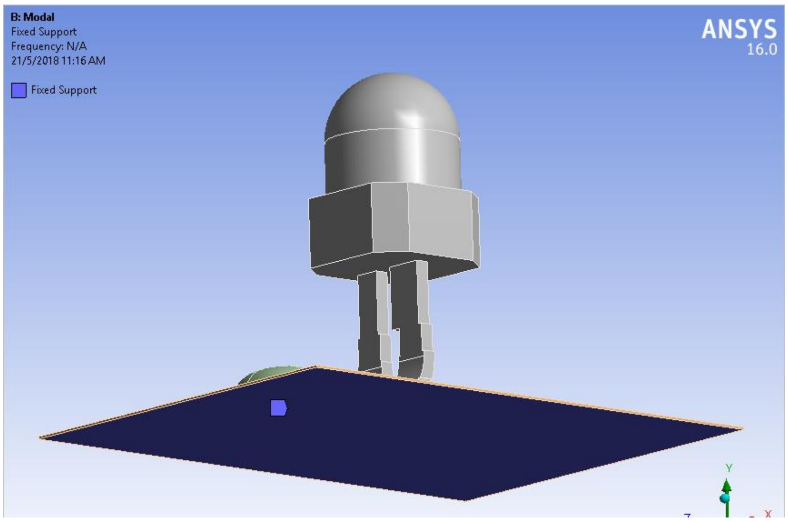


Figure 3.11 A fixed support applied on the model for modal analysis

3.5.2 Random vibration analysis

The solution from the analysis in modal analysis was used as input for this random vibration analysis. The number of maximum modes to find are 25 with limit search to range of 0 – 1000 Hz in the analysis settings.

Random vibration is a non-deterministic motion. The vibration pattern would be varied, to quantify the frequency excitation Power Spectral Density (PSD) input need to be assigned in the simulation. In this analysis, band limited white noise has been used which the spectral density has a constant value over a quantified frequency range as shown in Figure 3.12 and Table 3.4. Frequency range from 0 – 1000 Hz would make all the parts of the assembled model excited at the same time. The PSD input was referred from the JEDEC Standard 22b103B vibration, variable frequency. The standard is to test the reliability of package devices under various levels of application vibration to which component can be exposed[14].

The PSD input applied in this analysis was based on level where the component can be exposed to the most severe condition. PSD excitation was applied on the model's fixed support in Y-axis direction which was perpendicular to the model plane.

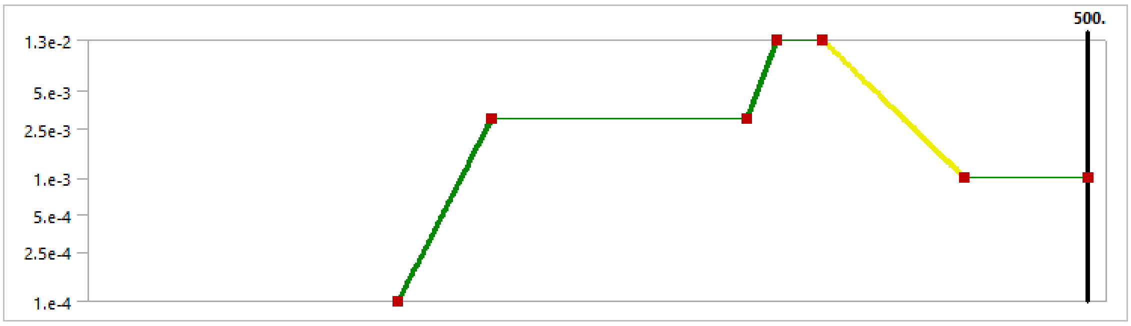


Figure 3.12 Power Spectral Density input curves

Table 3.4 Power Spectral Density input values

Frequency (Hz)	G Acceleration (G^2/Hz)
3	0.0001
6	0.003
40	0.003
50	0.013
70	0.013
200	0.001
500	0.001

CHAPTER 4

RESULT & DISCUSSION

4.1 Computer-aided Design (CAD) Model

The results of finite element analyses that are structural analysis and random vibration obtained from previous Chapter 3 would be discussed according in this chapter. From the analyses, the structural and reliability of all models of solder joint are obtained. In pull strength test simulation for structural analyses, two outputs are discussed that are total deformation and the maximum equivalent stress. The analyses were conducted for each material and for each model. Next, stress contour result for modal analysis are used to find the natural frequency of the model. The result from modal then used as input in random vibration analysis.

4.1 Static structural analysis

4.1.1 Configurations

A mesh independent study was conducted to compare the mesh generated for several configurations. Table 4.1 show the comparison for element available for each mesh configuration. The highest element generated was selected as datum to find the deviation. Then, the total deformation and maximum equivalent stress from the datum has been used to subtract the output from another configuration. The smallest deviation from the comparison done was selected as the best result.

Table 4.1 Comparison of configurations

	Setup 1	Setup 2	Setup 3	Setup 4 (Datum)
Relevance centre	Coarse	Medium	Fine	Fine
Smoothing	Medium	Medium	Medium	High
Elements	13498	24862	45 445	46 115
Deformation (m)	7.3512	6.0712	2.9712	3.9412
Deviation	3.41	2.13	0.97	–

4.1.1 Pull strength test simulations

A simulation of pull strength test was conducted for each model. Each model is based on the experimental model of the actual pull strength test discussed in Chapter 3. It is done to observe the structural integrity of the models and which part most likely to fail first. Besides, the effects of different configurations of adhesive on the solder joint also can be observed. Total deformation and maximum equivalent stress obtained from this analysis are discussed and compared.

The stress occurred for each model are shown in Figure 4.1 (a), (b) and (c). The maximum equivalent stress of all models mostly concentrated at the same spot that is at the intersection of solder joint and LED part. It can be seen that model 1 has the highest stress at 367.98 MPa produced after undergo simulation due to less surface area of solder joint that connected the LED and circuit pad. Hence, model 1 is most likely to break first when undergo actual pull strength test. From the Figure 4.1 (a), the red region showed that it has the highest amount stress concentrated there.

As model 3 has the lowest value of stress, it is estimated that the LED joint will take much longer time to break when pulled upward for 1mm/minutes. The maximum stress which indicated by red region is not shown for model 3 which mean it has the least concentrated stress in this analysis.

The simplified 2-dimension free body of the model is shown in Figure 4.1 with both boundary conditions that have been applied.

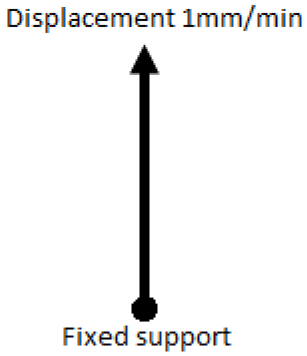
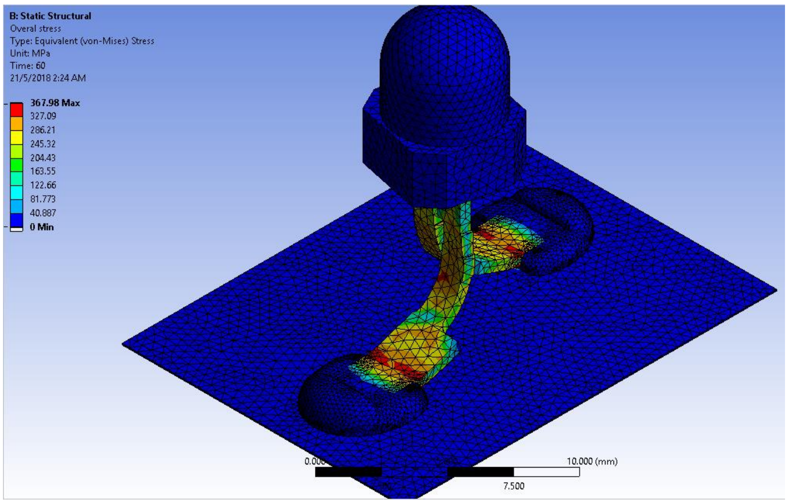
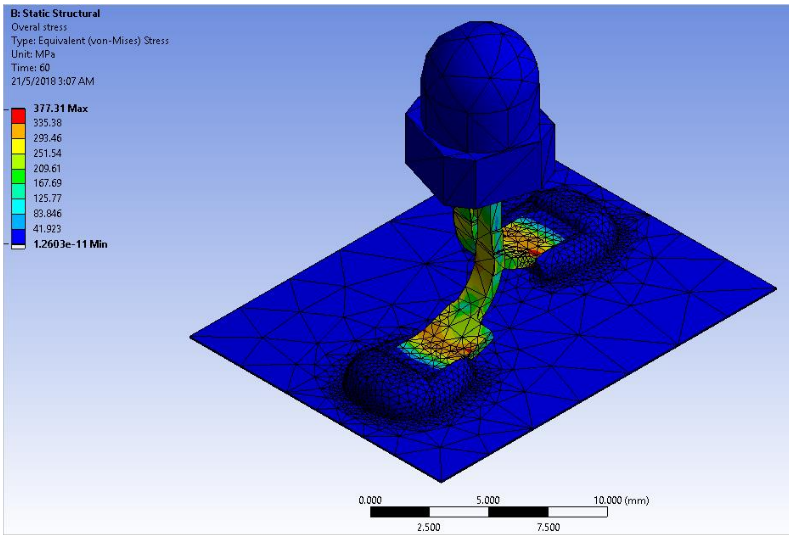


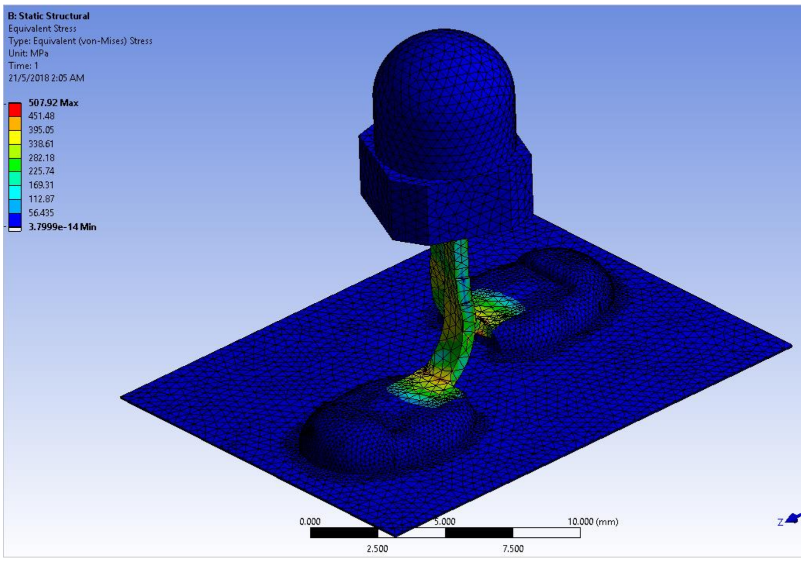
Figure 4.1 Free body diagram (FBD) of the model



(a)



(b)



(c)

Figure 4.2 Overall equivalent stress occurred on (a) Model 1 (b) Model 2 (c) Model 3

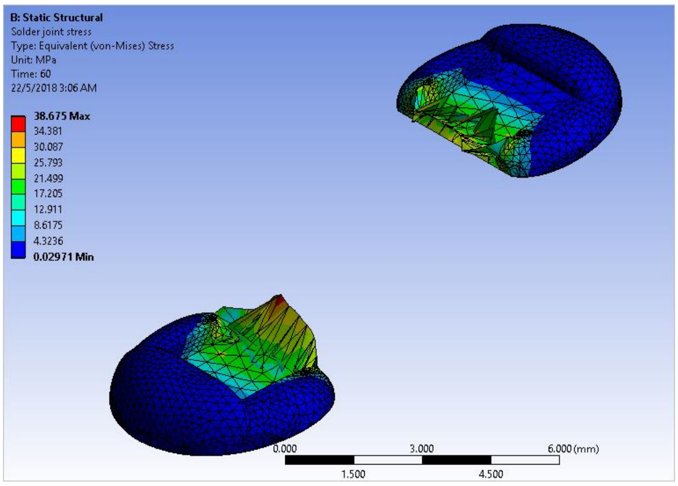
The maximum equivalent stress for each model is summarised in Table 4.2.

Table 4.2 Maximum equivalent stress for each model for all bodies

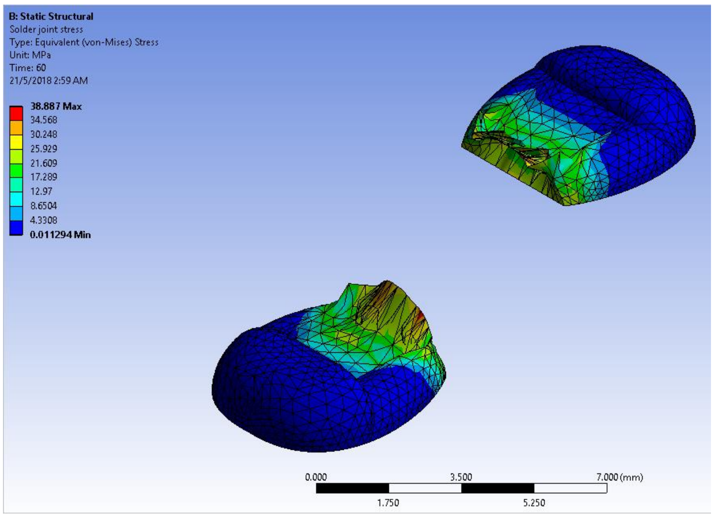
Model	Maximum Equivalent Stress (MPa)
1	377.31
2	367.98
3	341.96

In addition, this analysis of pull strength test simulation also emphasizes on the stress concentration at the copper pad circuit. This is because the solder joint act as the interconnector between the circuit pad and the LED joint. Thus, further stress analysis that focused on the solder joint is conducted too as it also coincides with the result for the all bodies of the model.

As shown in Figure 4.2, the maximum stress occurred at the edge of the opening of the solder joint. It is where the connection between the LED and circuit pad with solder joint as the medium. The highest stress was observed at the end of the solder joint that was pulled along with the LED joint.

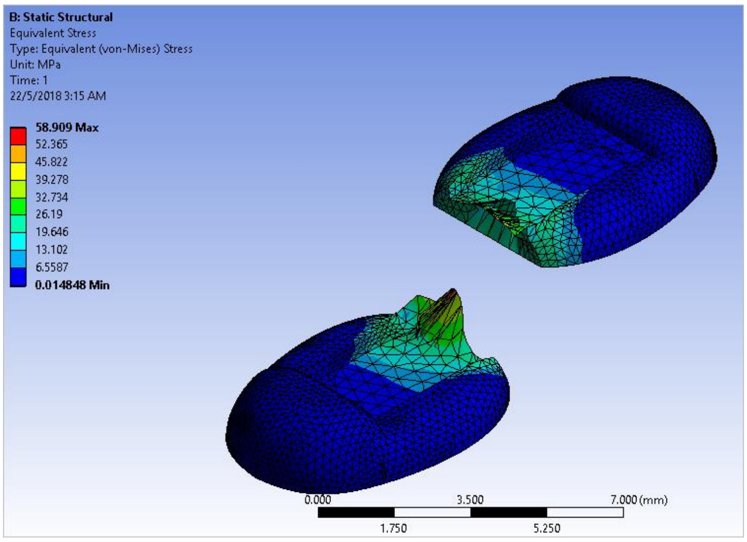


(a)



B: Static Structural
Solder joint stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 60
21/5/2018 2:59 AM

(b)



B: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
22/5/2018 3:15 AM

(c)

Figure 4.3 Maximum equivalent stress occurred at solder joint for (a) Model 1 (b) Model 2 (c) Model 3

Table 4.3 indicates that solder joint structure of model 3 has the highest stress exerted in pull strength test simulation. Whereas the stress exerted for solder joint of model has the smallest value, it means that the strength of solder joint structure become increasing from model 3 to model 1. The stress of each model exceeds the UTS value of the solder paste material, which can be deduced that all model totally fractured at the end of the experiment.

Table 4.3 Maximum equivalent stress for solder joint of each model

Model	Maximum Equivalent Stress (MPa)
1	38.675
2	38.887
3	58.909

Overall, it can be concluded that fracture will occur at the LED joint of model 3 whereas the fracture occurred at solder joint for model 1. Hence, the solder joint of model 3 has the higher reliability than remaining model. The differences for each configuration of how adhesive is dispensed for each model shows distinguish properties for each of them and has been discussed.

4.2 Random vibration

To determine the reliability of the solder joint and structure of the model under random occurrence of vibration environment where a range of frequencies were excited at the same time in a defined spectrum. The reason this analysis was conducted due to vibration may random in nature in a wide range of application for example vehicles travelling on rough roads or operational industrial equipment in which random loads may be encountered[4]. Also, it is essential to determine the resonant/natural frequencies and mode shapes as it can dictate the suitability of any application when in operation[15].

4.2.1 Modal analysis

Hence, to avoid the arbitrary loads on the assembled prototype by carrying out modal analysis of the structure in ANSYS. The computation of natural frequencies and the mode shape were derived from this process.

In this process, twenty-five modes were generated. For further analysis only twenty-two natural frequencies computed by ANSYS have frequency in the range of 3 – 500 Hz. These selected modes then were used to create the mode shape results. The deformation for each selected frequency then were computed. Table 4.4 shows the modes computed and its respective deformation that was generated for model 1.