STRUCTURAL ANALYSIS OF STRETCHABLE ELECTRONICS JOINT UNDER TENSILE AND VIBRATION LOADING

By

NADIRAH BINTI MUSTAFFA

(Matrix no: 120804)

Supervisor:

Dr. Abdullah Aziz Saad

JUNE 2017

This dissertation is submitted to

Universiti Sains Malaysia

As partial fulfilment of the requirement to graduate with honors degrees in

BACHELOR OF ENGINEERING

(MANUFACTURING ENGINEERING WITH MANAGEMENT))

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

ACKNOWLEDGEMENTS

My completion of Final Year Project will not be a success without many people. Hereby, I would like to acknowledge my heartfelt gratitude to those I honor.

First and foremost, I am expressing my praise to the Almighty for His guidance and blessing. I would like to deliver my utmost gratitude to my supervisor, Dr. Abdullah Aziz Saad, lecturer of the School of Mechanical Engineering, Universiti Sains Malaysia for his constant support, guidance and supervision throughout the completion of this project and thesis. I would also like to thank Mr. Fikri Sharif and Aqil Azman for all the help and assistance throughout of this project.

Subsequently, I would like to thank my friends who gave suggestions and comments on my work for further improvements. Last but not least, I would like to thank my family for their support and continuous motivation during my final year study. With their support, I managed to perform well for my final year of undergraduate.

Title of thesis:

Structural Analysis of Stretchable Electronics Joint Under Tensile and Vibration Loading

Date of submission (Academic year): **7th June 2017**

Candidate (Matrix no.): **Nadirah binti Mustaffa (120804)**

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

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 .... (Candidate)

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 .... (Candidate)

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 .... (Candidate)

Date ....

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

ANALISIS STRUKTUR SAMBUNGAN ELEKTRONIK ELASTIK BAWAH BEBAN TEGANGAN DAN GETARAN

ABSTRAK

Sambungan pateri adalah bahagian yang penting dalam alatan elektronik sebagai sambungan elektrik dan ikatan mekanikal. Oleh itu, kajian terhadap kebolehpercayaan sambungan pateri amat penting untuk mengetahui kebolehan sambungan di dalam alatan untuk terus berfungsi dalam pelbagai keadaan. Kebolehpercayaan sambungan juga boleh dipengaruhi oleh struktur dan bahan sambungan pateri. Oleh sebab, alatan elektronik banyak digunakan dalam industri automotif, kebolehpercayaan sambungan terhadap getaran juga menjadi tumpuan dalam kajian ini. Dua Led 3D model yang disambungkan ke litar dengan stuktur sambungan pelekat konduktif yang berbeza dihasilkan dengan Solidworks 2016, ANSYS workbench 16.0 pula digunakan untuk mensimulasikan model. Tujuan kajian ini adalah untuk mengkaji struktur sambungan, kekuatan, dan kebolehpercayaan sambungan terhadap getaran. Dua analisis telah dilakukan, struktur dan getaran rambang. Model yang disimulasikan dalam analisis struktur ditarik dengan kadar 1mm/minit pada paksi-Y untuk menganalisis struktur dan kekuatan sambungan. Analisis dibandingkan dengan keputusan eksperimen sebagai pengesahan. Kekuatan bahan ditukar dalam analisis untuk mengkaji hubungan kekuatan bahan dengan kebolehpercayaan sambungan. Model disimulasikan dalam analisis getaran rambang dengan julat frekuensi getaran antara 3Hz-500Hz. Struktur sambungan dianalisis berdasarkan tekanan maksima dan kontur pada model. Daripada analisis terhadap struktur model, boleh disimpulkan bahawa struktur sambungan mempengaruhi kekuatan sambungan dan kekuatan bahan yang lebih tinggi mempunyai kebolehpercayaan yang lebih tinggi. Keputusan analisis getaran rambang menunjukkan julat frekuensi yang dikenakan pada model memberi kesan dan kegagalan berlaku pada struktur sambungan permukaan pelekat konduktif dengan litar.

STRUCTURAL ANALYSIS OF STRETCHABLE ELECTRONICS JOINT UNDER TENSILE AND VIBRATION LOADING

ABSTRACT

Solder joint is the major part of electronic devices to form an electrical connection and mechanical bond. Thus, study of electronics joint reliability is significant to determine the ability of the interconnection to maintain its functionality under subjected environments. Reliability of electronics joint is actually can be affected by its structure and materials used. As currently electronic devices is widely applied in automobiles application, the reliability of electronics interconnections under vibration loading is also an issue in this study as its leads to joint failure. Two 3D led with different conductive adhesive electronic joint structure that connected to a circuit model is developed by using Solidworks 2016, whereas ANSYS workbench 16.0 was used to simulate the models. Purpose of this research is to study the structural of conductive adhesive joint, strength and its reliability under vibration load. Two analyses were done, structural analysis and random vibration. Models was simulated in structural analysis and subjected to 1mm/min Y-axis displacement loading to analyze the adhesive joint structure and its strength. The analysis was compared with pull strength test experimental result as verification. Then different adhesive joint material strength was applied in both models to study the relationship between adhesive joint materials with its strength. Models were simulated in random vibration analysis to study the adhesive joint reliability in the frequency ranges 3Hz-500Hz. Structural of adhesive joint was analyzed based on the maximum equivalent stress contour and concentration on the model. Maximum equivalent stress value was analyzed to determine adhesive joint strength and reliability. From the structural analysis it can be concluded that joint structures affect the interconnection strength and higher material strength is more reliable than lower material strength. Random vibration analysis results showed both solder joint model is affected by the excited frequency and failure occurred at the connection between joint and copper pad circuit.

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Solder joint failure is a dominant failure in electronic devices applied in automotive industry due to its constant exposure to the various environments, especially vibrations. Increase of product functionality corresponding with miniaturization, surface mount technology is the most appropriate application to connect the components with circuit board [1]. In this technology application the strength of interconnection may be affected by the several factors such as joint structure and materials properties.

Vibration loads that triggered from the forced vibration by rapid, comparatively small movements in reversing directions that can cause failures in electronic component interconnections is important to be considered as issue for its' applications in transportation, portable devices and machines. Vibration caused the stress on the PCB interconnection with electronic components. This stress is due to a combination of the bending moments in the PCB and the inertia of the component mass, one of the effects of this stress is the solder joint fracture which will leads to total failure [2]. Thus it is needed to study the reliability of joint under vibration loads.

Electronics joint structure analysis is evaluated by the strength of different joint structure produce by different method. Electronics conductive adhesive joint reliability analysis under vibration loads is the evaluation of joints ability to withstand failure under vibration loading in a range of frequency excitation. As well as structural analysis is needed in this study to analyze the effects of vibration loads on the electronics conductive adhesive joints and its' structures. From these analysis the critical area on joint and its' reliability will be determine. Structural and reliability analysis will be analyzed under simulation and experimentation. An electronic component on PCB joint with electronic conductive adhesive joint will be modeled for FEA. Modeled adhesive joint will be simulated with structural analysis and random vibration analysis. For the validation an experimental with same model will be conducted.

Although, the study of electronics joint structure and reliability under vibration loading has receive a lot of attention for years until recent, but the detailed studies in this area is still warranted. This study is vital in designing, manufacturing process and controlling quality of electronic and microelectronic devices and packages. Thus, it is very important to study the structural and reliability of solder joint under vibration and tensile loading.

1.2 PROBLEM STATEMENT

Electronic products are commonly subjected to vibration conditions according to nature of stress. The vibration loading is the one of major source of damage in electronic components and system. It is announced by Steinberg that 20% of electronic products are destroyed due to vibration [2]. The problem of vibration fatigue could be due to solder joints fatigue as it is the major component in the electronic products and it is vital element to analyze the cause of failure. Conducting reliability and structural analysis of electronics joints under vibration loading is important to develop a stable electronics and microelectronics packages.

There are few variables that affect the electronics joint failure under vibration condition such as joint material, geometry structure and finishing treatment. Firstly, this study will focus on reliability of structural and materials strength of conductive adhesive joint. Secondly, the reliability of conductive adhesive joint under random vibration loading is investigated. Both studies will be analyzed with simulation and experimentation.

1.3 OBJECTIVES

1. To develop Finite Element model and analyze conductive adhesive electronics joint structure and reliability subjected to the random vibration loading
2. To assess the strength of conductive electronics joint structure under tensile loading

1.4 SCOPE OF WORKS

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. In this project, the model is modified to cater the needs of performing structural and random vibration analysis. Models will be analyzed with ANSYS in structural analysis to analyze the conductive adhesive electronics joint structure and to identify the strength of the joint. Models were also being analyzed in modal analysis to obtain the vibration modes on the models and then being analyzed in random vibration to analyze the structure and failure occurred in the model. The results of experiments were used to verify the simulation analysis results.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Electronic interconnection fatigue and failure in electronics device assemblies under vibration is one of key concerns for automotive industry, and portable devices. Many studies and research has been dedicated to predict the joint reliability in electronic assembly devices under mechanical environment. Experimental techniques and simulation are mostly used to analyze the dynamic response of the system subjected to vibration and tensile loading. An in-depth study of the literature published in this area is necessary to understand the manufacturing process of surface mount technology and approaches used in analysis to address the reliability concerns.

2.2 SURFACE MOUNT TECHNOLOGY (SMT)

Two types of electronic component used in electronic devices, through hole and surface mount devices. These two components use different mounting technologies which are through holes technology and surface mount technology (SMT). Traditionally, through holes technology are commonly used in electronic devices, since the electronic technology is towards miniaturization, compact and high packing densities in a circuit board, the application of surface mount technology are more reliable for the features. SMT is a method where component is mounted directly onto surface of printed circuit board, the solder joint will form mechanical and electrical connection. The process step involve in the surface mount assembly process, starts with solder paste dispensing, components placement and reflow. Soldering paste is printed on the circuit board at the area where only component will be soldered. Solder can be paste using two methods stencil printing and syringe dispensing. Solder paste usually go through an inspection with either 2D technology or 3D technology, to ensure solder paste is correctly applied and the paste structure is even. Components are then placed on the printed solder paste.

Circuit board goes through to solder oven to melt the solder paste to form solder joint between the components and circuit board. During reflow process, in industry nitrogen gas is exposed to the board and warm up gradually with heated air to melt soldering paste and let the flux vaporized [3].

2.3 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 lead-free solder in industry Sn63Pb37. Lead solder is easier to work with as it is 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 [4]. 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 been used to provide higher melting point or to satisfy the component and circuit requirement [5]. 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 promote good contact within and other particles as it deform 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 dispense 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 [6].

2.4 RANDOM VIBRATION

Random vibration is the most related vibration condition subjected in the environment conditions, it is due to destabilize created by rotational and tumbling masses. Automobiles and portable electronic components are mostly exposed to random vibration. A system can vibrate in a standard manner that is known as vibration modes, the vibration modes are representing the degree of freedom of the vibration in the system with a certain natural frequency. Range of frequency varied according to its application, it had been tested that 2.5 Ton truck provide acceleration level 15 G and 19G with frequency 15Hz-40Hz at speed of 10-15 mph [2]. In the automobiles vibration frequency range is varied with the system, vibration while driving, suspension system, and steering system, which the vibration range from 3Hz-1KHz [7].A range of frequency is excited at the same time, which means that frequency loading correspond to all masses are represented at the same time which produce large displacement amplitude to produce impact of failure to analysis system. To quantify random frequency, Power Spectral Density (PSD) input is needed. PSD is the input curve of acceleration units per Hz^2 versus range of frequencies. PSD curve input is translated to the Root Mean Square value which the value of each curve is formulated to a single quantity, RMS level is related by area under the PSD curve. As the random vibration is the nature response statistic it is represented in zero mean normal Gaussian distribution which presented the quantify value in the range of probability of occurrences. In random vibration, RMS value of acceleration, velocity, stress and strain can be quantified. The probability is presented in three levels as shown in Figure 2.1 as example, 1σ, 2σ, 3σ, where the quantify values in between +1σ and -1σ are 60.3% of the time, +2σ and -2σ are 95.4% of the time and +3σ and -3σ are 99.73% of the time. The probability of occurrences shows that most failure generated by the 3σ RMS level [8],[2].

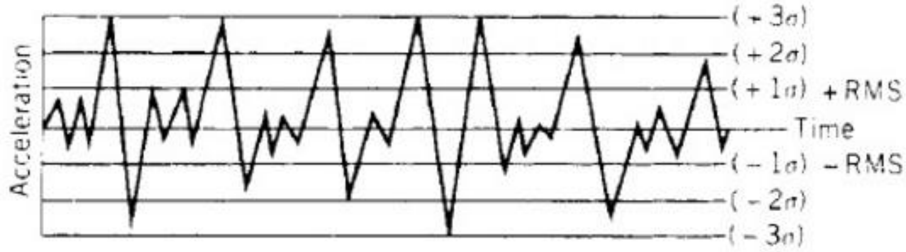


Figure 2.1: Probability of occurrences over time history of random vibration acceleration traces [2]

2.4.1 FINITE ELEMENT ANALYSIS

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 2 models that have been used in other research detailed FE models and smeared FE models. 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 of appropriate accuracy much more quickly and with less effort [9]. The simulation of FEA used to check the stresses on solder balls when conducting a fatigue life assessment of the component, the boundary condition setting that identical to the vibration test is used in the analysis. The variation in mesh densities is applied in the model in order to examine the convergence of the analyzed frequency results. For verification of FEA model, natural frequencies test vehicle are examined experimentally with modal testing method [10]. Besides, FEA is used to obtain stresses for the fatigue estimation. The analysis then was validated with experimental modal analysis to determine natural frequencies and mode shapes [11]. Researcher also, employed FEA to get numerical deflection of solder joints similar to the out-of-plane displacement measured with the scanning vibrometer. The package is accurately modeled. The modal analysis is first performed on the global model and compared with experimental results. Equivalent strain in the critical area of solder joints is then determined with global-local approach [12].

2.4.2 VIBRATION TESTING

Vibration testing is the study of structure's response while exposed to a specific dynamic environment, the environment simulate 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 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 [13]. 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 [14].

CHAPTER 3

METHODOLOGY

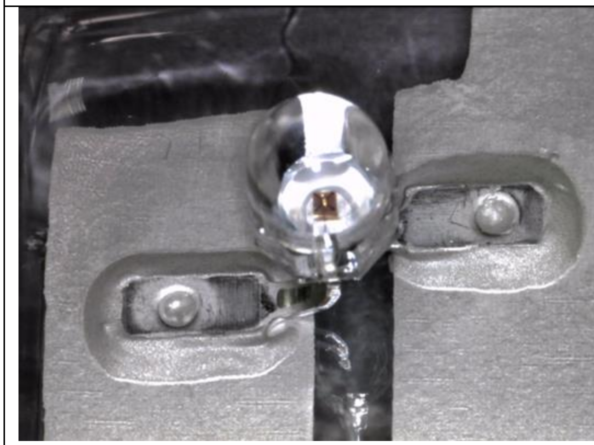
3.1 OVERVIEW

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 2016 and Analyses of structures, strength and reliability in ANSYS 16.0 is also have been characterized in detailed.

3.2 EXPERIMENTAL MODELS

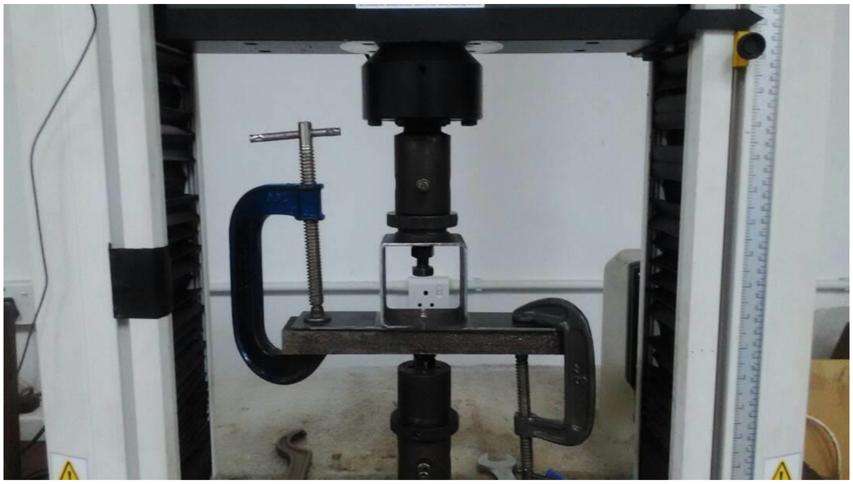
In the pull strength test experiment, two different models were built. An electronic component, led was assembled to a simple circuit. The circuit base was built with Lexan 8010 polycarbonate film, the polycarbonate base was padded with the silver layer as an electrical connectivity circuit. Non-lead conductive adhesive, silver epoxy LOCTITE ABLESTIK ABP 2032S was used as solder joint with two different dispensing methods. The dispensing process was done manually by using syringe is summarized in Table 3.1.

Table 3.1 Dispensing method for both model 1 and 2

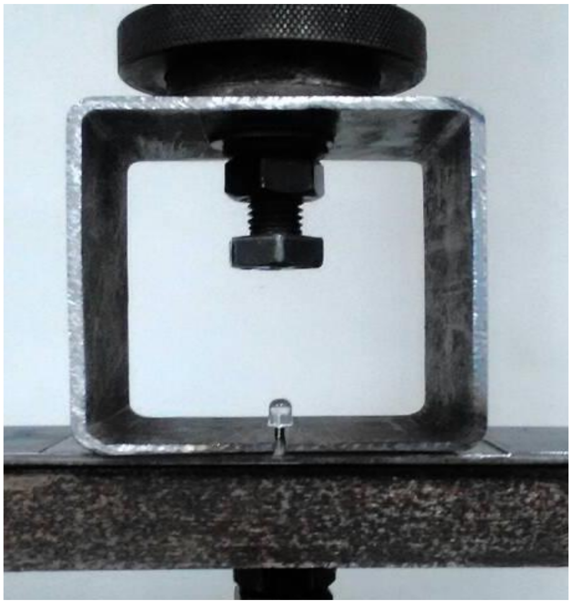
Models	Joint material	Dispensing method
 Model 1	Silver epoxy (LOCTITE ABLESTIK ABP 2032S)	Led was placed on the circuit and 3 dots of solder paste were dispensed at each sides of the led lead base.
 Model 2	Silver epoxy (LOCTITE ABLESTIK ABP 2032S)	Led was placed on the 3 dot of solder paste that had been dispensed on the circuit, then one dot of solder paste was dispensed on the edge of led lead base

3.3 PULL STRENGTH TEST

Pull strength test was conducted by using Universal Testing Machine (UTM). Special jig was custom made to fix the model on the UTM. Model was placed on the UTM fixture, holding jig then placed on the model then clamped with the UTM fixture, the holding jig have hole at the center 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) Placement of models and jig on the UTM fixture, (b) Jigs attachment on the models

3.4 GEOMETRY MODEL

Two 3-dimensional (3D) CAD model was developed based on the assembly prototype model that had been used in pull strength test experiment as shown in Table 3.1. CAD model of led was developed by referred to the actual component specifications from manufacturer. The actual joint dimension cannot be determined as it is not a geometry structure. Therefore, the joint 3D CAD model was developed approximately as the actual prototype. 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 3.2 shows the both assembled 3D CAD model of led with electronics conductive adhesive joint and Figure 3.3 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
Joint structure model 1	6.5×3.5×0.5
Joint structure model 2	6.5×3.5×1.2
Copper pad circuit	20×15×0.08
Polycarbonate base	150×150×1

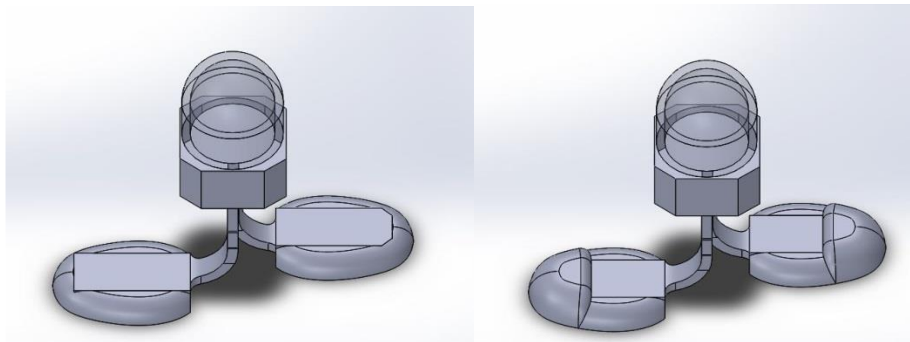


Figure 3.2: Assemble model of Led with conductive adhesive joint

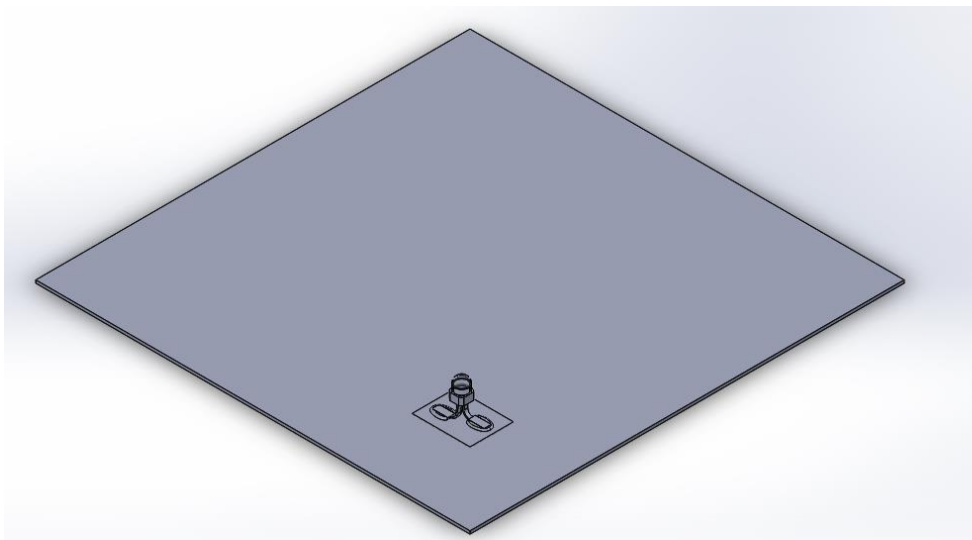


Figure 3.3: Complete assembly model

3.5 MATERIALS

Materials applied were based on the material used in the experimental prototype model. Since, complete material properties for experimental prototype cannot be determined, the materials in the simulation were modified to cater the simulation needs. Materials' mechanical properties were provided from the ANSYS materials library and manufacturer sources that not exactly same as in 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 Mechanical material properties

Parts	Materials	Density (kg/m³)	Young's modulus (MPa)	Poisson ratio	Yield strength (MPa)	Tangent modulus (MPa)	Ultimate tensile Strength (MPa)
Copper pad circuit	Copper [15]	8900	130000	0.34	120	125	210
Poly-carbonate base	Polycarbonate [16]	1200	2506	0.38	63	0.05	65
Adhesive joint	silver epoxy [17]	4500	4140	0.32	24.1	38.62	34.5
Lead	copper alloy [15]	8300	110000	0.34	280	1150	491

3.6 MESHING MODEL

Meshing is a process that influences the time consume for solution, accuracy, convergence, and CPU memory consume for 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, non-conformal mesh is developed. Due to the models developed is multi parts which need to locally modified and contains a complex geometry, unstructured grid and non-conformal mesh was developed. Therefore, it was time consuming to define the tolerance values to match all nodes. But, the analysis still can be computed, since the contact region of each parts was well defined and recent numerical analysis in ANSYS allow the computation of non-conformal mesh interpolation [18].

3D model was meshed in ANSYS 16.0. Meshed can be either setup manually or automatically. In this research the electronic component assembly model was meshed manually by using hex dominant method for four important parts in assembly bodies which includes conductive adhesive joints, led component and copper pad circuit. Polycarbonate base was meshed automatically. Mesh sensitivity had been analyzed as shown in Table 3.3, from the result of mesh sensitivity analysis there was only 2.73% difference between the result obtain from both methods, but automatic method consumed more running time and large amount of computer memory. Therefore, hex dominant method was used because it could reduce the run time. Besides, the elements would align in direction of flow which could reduce the numerical error. Body sizing was adjusted according to which parts that need to be focus to increase the mesh density to obtain more accurate finite element analysis result. 0.2mm element size was applied for conductive adhesive joint and led, 0.5mm for the copper pad circuit and 1.5mm applied on the polycarbonate base. Fully mesh model is shown in Figure 3.4.

Table 3.4 Summary of mesh model sensitivity

Body part	Mesh method	Element size (mm)	No. of element	Maximum equivalent stress (MPa)
Solder joints	Automatic	0.2	133345	16.939
	Hex dominant	0.2	29537	16.483

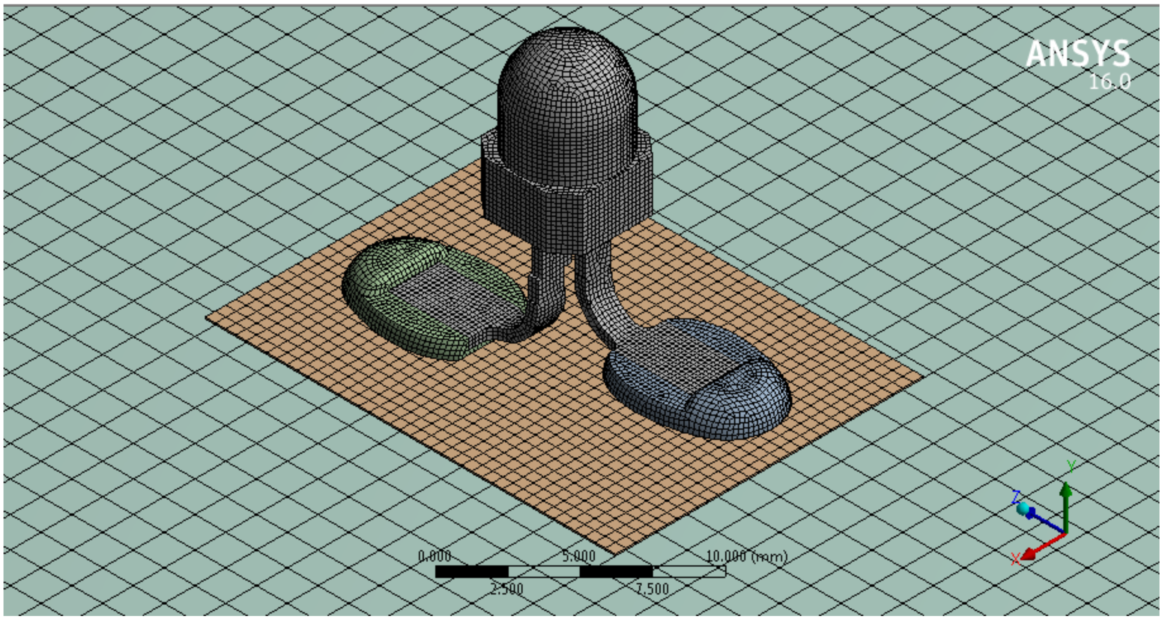


Figure 3.4: Mesh model

3.7 STRUCTURAL ANALYSIS

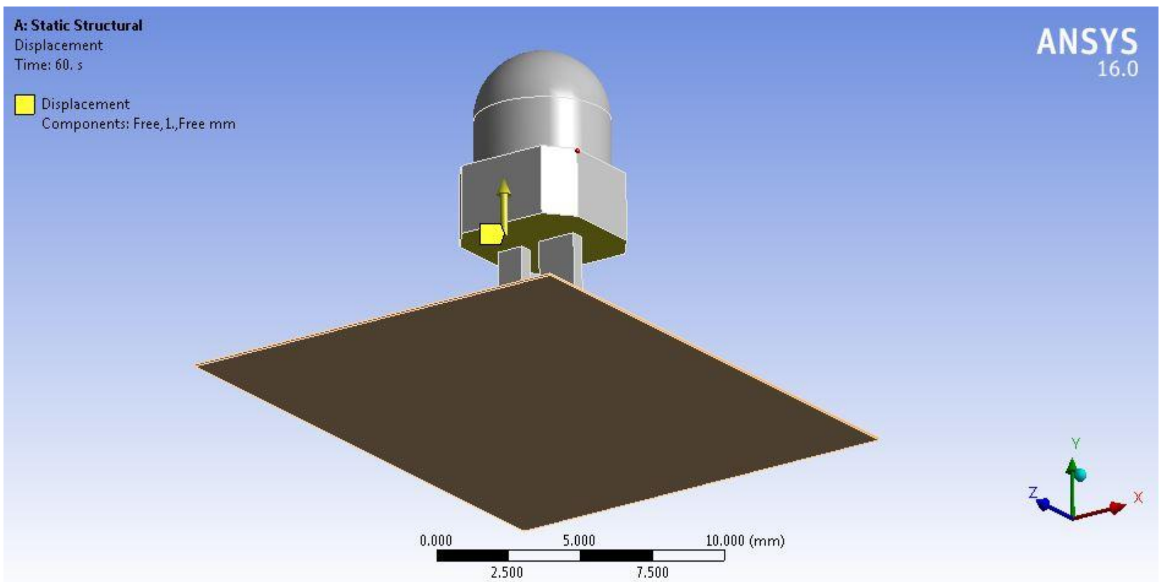
Structural analysis was done in ANSYS 16.0 to analyze the conductive adhesive joint structural and its strength. The analysis was run for both models and with different joint material strength, the new material mechanical properties applied are as in Table 3.4.

Table 3.5 Mechanical properties of material 2

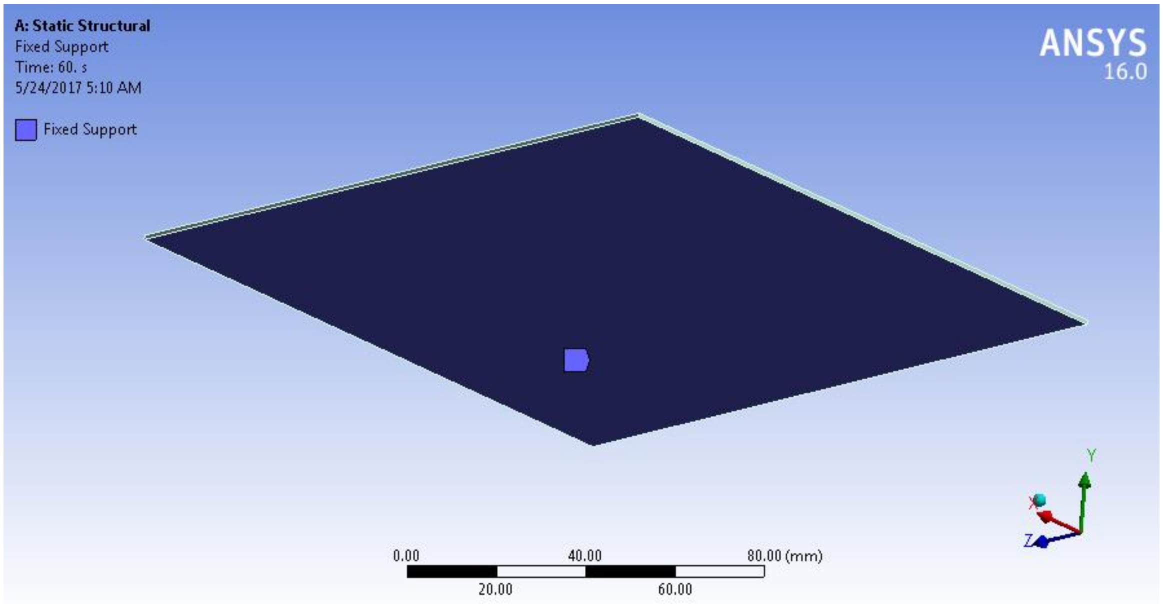
Materials	Density (kg/m³)	Young's modulus (MPa)	Poisson ratio	Yield strength (MPa)	Tangent modulus (MPa)	Ultimate tensile Strength (MPa)
Silver epoxy [19]	4500	345	0.32	6.89	6.55	3.22

3.7.1 BOUNDARY CONDITION

Boundary condition was set up after materials and mesh was applied. In structural analysis only two boundary conditions was applied, fixed support and displacement. Both boundary conditions were applied by referring to the pull strength test experiment. Fixed support was applied at the bottom of the polycarbonate base surface and the displacement was applied below the led capsule surface at Y-axis direction for 1mm/minutes as shown in Figure 3.5 (a) and (b).



(a)



(b)

Figure 3.5: The boundary condition applied (a) displacement (b) fixed support

3.8 RANDOM VIBRATION

Random vibration analysis was done in ANSYS 16.0 to analyze the failure occur on the model under random vibration loading. The analysis was run for both model 1 and 2. Before random vibration was done, models need to be analyzed in modal analysis to determine the dynamic response.

3.8.1 MODAL ANALYSIS

Materials and mesh model used in this analysis were as in structural analysis. Before proceed to random vibration analysis, modal analysis need to be run first to identify the dynamic response on the model. Only fixed support was applied as boundary condition. The fixed support was applied at the two edge of the polycarbonate base as shown in Figure 3.6. From this analysis, 20 mode shapes were identified, and 12 mode shapes were identified were in the frequency range 3Hz-500Hz. The 12 mode shapes then would be used in the random vibration for further analysis.

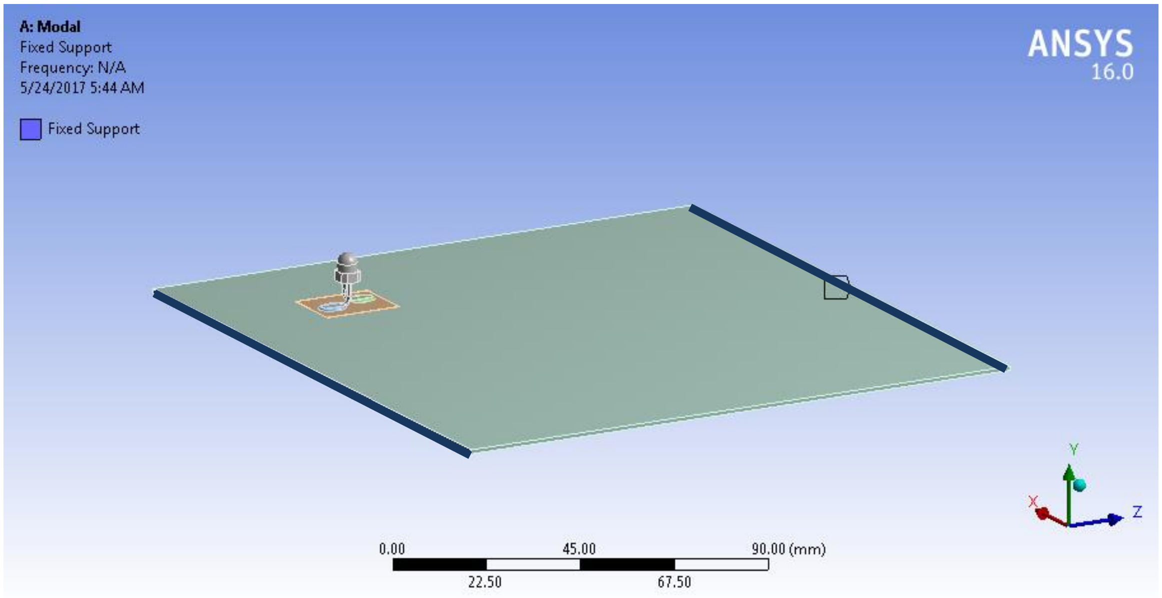
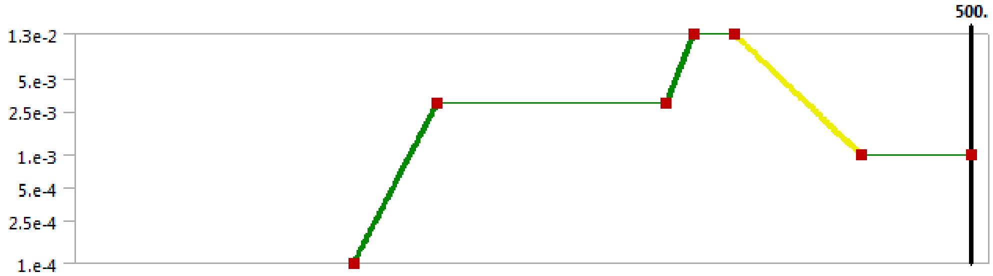


Figure 3.6: Fixed support applied on the model for modal analysis

3.8.2 RANDOM VIBRATION ANALYSIS

The solution data history from the modal analysis was used in random vibration analysis. The number of 12 mode shapes used and 0.02 of constant damping ratio need to be set up in the analysis settings. Random vibration is a non-deterministic motion. The vibration pattern would be varied, to quantify the frequency excitation 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 3.7. Frequency range from 3-500Hz would be 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 [20]. 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.



	Frequency [Hz]	G Acceleration [G²/Hz]
1	3.	1.e-004
2	6.	3.e-003
3	40.	3.e-003
4	50.	1.3e-002
5	70.	1.3e-002
6	200.	1.e-003
7	500.	1.e-003
*		

Figure 3.7: Power Spectral Density input curve and values

CHAPTER 4

RESULT & DISCUSSION

4.1 INTRODUCTION

In this chapter, the results of finite element analyses obtained would be discussed in detailed. The discussions were divided into two sections, results of structural and random vibration analysis. From the analyses, the structural and reliability of electronics adhesive conductive joint is obtained. Deformation results from structural analysis would be compared with pull strength test experiment results to verify the simulation. Next, the stress contour results on the model are analyzed to acquire the structural analysis, the stress results would be compared with two different models developed and different joint material. Second part of this chapter would discussed the random vibration analysis which discussion would focused on the stress contour results on model and the maximum equivalent stress value for each part.

4.2 STATIC STRUCTURAL ANALYSIS

4.2.1 VERIFICATION OF FINITE ELEMENT ANALYSIS MODEL

A simulation method used need to be verified with the experimental results. The verification is analyzed to determine the reliability of simulation results and whether method used can be accepted or not. In this research, verification is done by compare the maximum Y-axis directional deformation results of simulation and the pull strength test in a range of period. Table4.1 and Table4.2 shows that there is slightly difference between simulations and experimental for both model with average of percentage difference 5.07% for model1 and 7.30% for model 2, although the material properties applied and joints dimension are not exactly same as in experiment. Both deformation results for simulation and experiment are consistently increases and do not have large deviation. Therefore, Finite Element Analysis model is verified and the result is reliable for further analysis.

Table 4.1 Model 1 percentage difference of maximum deformation

Time (sec)	Maximum Deformation (mm)		Percentage Difference (%)
	Simulation	Experiment	
9.1116	0.15948	0.15155	5.09
16.862	0.29555	0.28093	5.07
24.613	0.43131	0.41004	5.05
33.605	0.58889	0.55977	5.07

Table 4.2 Model 2 percentage difference of maximum deformation

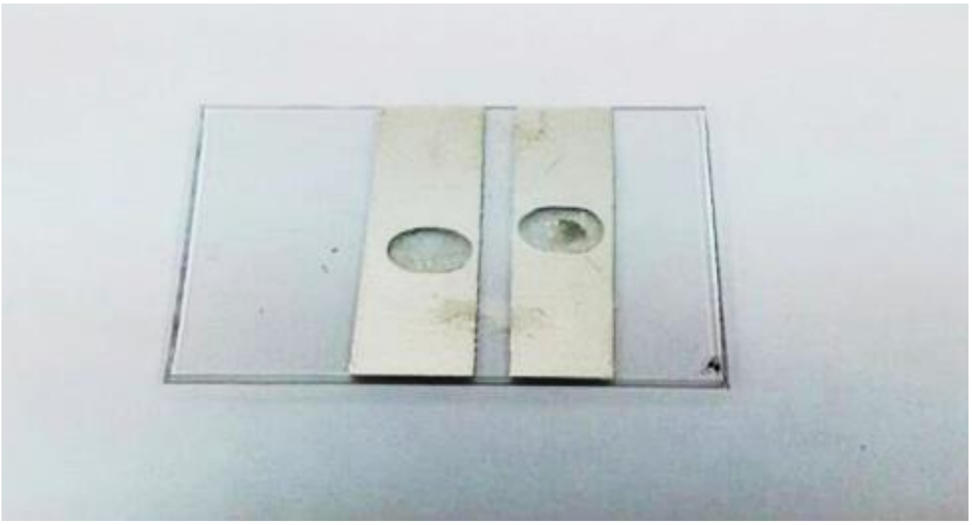
Time (sec)	Maximum Deformation (mm)		Percentage Difference (%)
	Simulation	Experiment	
12	0.21308	0.19982	6.42
24	0.43002	0.39991	7.26
36	0.64746	0.59995	7.62
52.34	0.94388	0.87209	7.91

4.2.2 PULL STRENGTH TEST EXPERIMENTAL

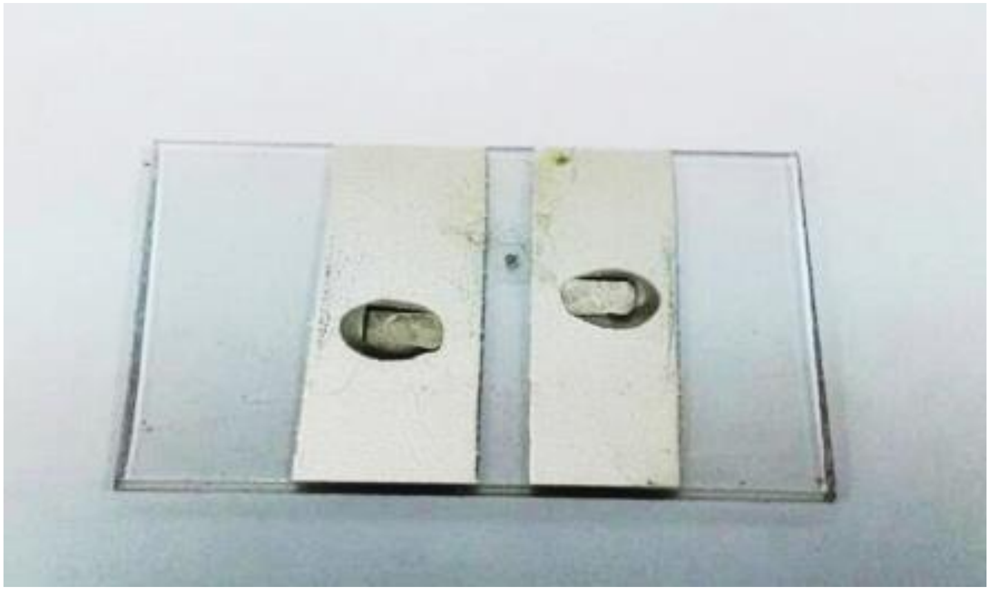
A pull strength test was conducted to determine the strength of conductive adhesive joint with different dispensing method and to verify the reliability of simulation results. Two models with different dispensing methods model 1 and model 2 is compared, from Table 4.3 the average pull strength experiment result is showed which the strength of model 1 is higher than model 2, and also it has been identified from the experiment that the failure is most likely occurred either at the connection of led lead with adhesive joint or adhesive joint with copper pad circuit as in Figure 4.1 (a) and (b).

Table 4.3 Average pull strength of model 1 and 2

Model	Average pull strength (N)
1	13.1580575
2	10.37892714



(a)



(b)

Figure 4.1: (a) adhesive joint pull off with copper pad circuit, (b) lead pull off from adhesive joint

4.2.3 EFFECT OF CONDUCTIVE ADHESIVE JOINT MODEL STRUCTURE

In this analysis, two different conductive adhesive joint model structures that replicate the experiment were developed. Stress analysis would be done and discussed in this section to know the maximum stress concentration exerted on the structure at maximum deformation. The stress analysis was done for each component of the model structure to identify which part is the most likely to fail for both model, which joint structures have high strength and to determine the effect of material strength with the different of joint structure.

Stress concentration as shown in Figure 4.2 (a) and (b), shows that the maximum equivalent stress concentrated at the same spot which is at the corner of led lead for both model, the comparison of stress value are as in Table 4.4. Stress value of model 1 is higher than model 2 which means that model 1 is most likely to fail at the corner of led lead. Compared to the ultimate tensile strength (UTS) of led lead material 491MPa,