MODELING OF DYNAMIC RESPONSE OF DESKTOP COMPUTER'S MOTHERBOARD DUE TO DROP IMPACT

MOHD MAARIF BIN ABDUL MALIK

UNIVERSITI TEKNOLOGI MALAYSIA

MODELING OF DYNAMIC RESPONSE OF DESKTOP COMPUTER'S MOTHERBOARD DUE TO DROP IMPACT

MOHD MAARIF BIN ABDUL MALIK

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

MAY 2010

A project report submitted in partial fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Dedication to my beloved father and mother

(Abdul Malik Tan and Hasmah Taib), sisters,

brothers, special friend and all my friends.

Thanks for everything

.....

ACKNOWLEDGEMENTS

Dengan nama Allah Yang Maha Pemurah Lagi Maha Mengasihani. Selawat dan salam ke atas Junjungan Besar Nabi Muhammad S.A.W.

First and foremost I wish to give all the praise to Almighty God for giving me the strength and time to complete this research. With His blessings may this work be beneficial for the whole of humanity.

I'm deeply indebted to my supervisor, Dr. Nazri bin Kamsah, for his help, guidance and encouragement throughout this work. He has thought me his professionalism and the profound art of research which inevitably is reflected in this thesis. For all these, and for innumerable friendly discussions we have had, I am very grateful. Thanks also for his advice, guidance and motivation. Without his continue support and interest, this thesis would not have been the same as presented here.

My sincere appreciations to Haslina Abdullah and Farizana Jaswadi as their guidance in installing the ABAQUS software and modeling procedure that had made the process become ease to complete. This appreciations also extend to all my colleagues and friends at Universiti Teknologi Malaysia and Jabatan Kerja Raya Malaysia for their co-operation and assistance. I would also like to acknowledge the support of my parents, sisters, brothers and girl friend for their kindly good encouragement in the course of my study.

Finally, I am also indebted to Majlis Amanah Rakyat (MARA) for the financial support in my Master study. Without this financial support, this project would not be possible.

ABSTRACT

Electronic products fit into the customer and variable market segments around the world. Reliability of products such as computers, laptop computers, digital cameras, calculators, mobile phones, and other electronic products have become a major concern recently. In this research, detailed drop test simulations are performed using ABAQUS 6.8 finite element software to investigate transient dynamic response of the motherboard based on specifications obtained from the JEDEC Standard. The aim is to evaluate the deformation of the portion motherboard and distribution of stresses in the solder balls interconnection. These connections are the weakest link in the system so realibility of the entire system depends on them. The effects of drop orientations and position of package on board. are also studied. The accuracy of finite element simulation results usually depends on element size control. Therefore, mesh sensitivity analysis was conducted on the finite element model. The simulation results show that at specific time, the model impact to the rigid floor and the stress begins to develop at the lower corner of the board. The board experiences small bending because the lower corner is suddenly impact brought to rest while the other part on the board is still travelling downwards with maximum velocity. At the end of the simulation, the model up, bounce away from the floor under a free vibration condition. Package located at lower corner of the board experince the greatest stress wave transmision than the those at the centre and upper part in 45^0 drop orientation. For vertical drop direction, package at the upper edge undergoes the largest stress compared to the others. Varying the drop orientation angle from 45^0 to 90^0 will produce great stress wave transmission into the solder joints through the board.

ABSTRAK

Produk elektrik pada masa kini perlulah memenuhi citarasa pelanggan dan segmen pelbagai pasaran. Kemampuan produk seperti komputer, kamera digital, kalkulator, telefon bimbit dan produk lain yang kerap digunakan kini menjadi tumpuan. Dalam kajian ini, ujian jatuh secara simulasi telah dilaksanakan dengan menggunakan perisian ABAQUS 6.8 untuk menyiasat tindakbalas dinamik ke atas papan utama berdasarkan spesifikasi jatuhan daripada piawaian JEDEC. Tujuannya adalah untuk menilai kesan perubahan bentuk papan utama dan peredaran tegasan di dalam penyambungan bebola pateri. Penyambungan ini adalah rangkaian hubungan yang paling lemah di mana kemampuan sistem adalah berdasarkan penyambungan tersebut. Parameter yang akan dikaji secara kaedah matematik adalah kesan perubahan orientasi jatuhan dan posisi komponen di atas papan. Ketepatan penyelesaian kaedah secara simulasi adalah bergantung kepada kawalan saiz elemen yang digunakan. Oleh itu, analisis terhadap sensitiviti jejaring telah dijalankan ke atas model unsur terhingga. Keputusan ujian simulasi yang telah dijalankan menunjukkan bahawa pada masa yang tertentu, model akan menghentam permukaan lantai yang kasar di mana ketika ini tegasan akan mula terbentuk di penghujung sebelum mengembang ke keseluruhan model. Papan akan menghasilkan lenturan kecil di penjuru bawah ataupun sisi papan ketika hentaman dan tegasan tersebut akan dipindahkan kepada komponen lain yang mana masih menurun ke bawah dengan halaju yang maksimum. Di akhir ujian simulasi, model akan melantun ke atas dalam keadaan getaran bebas. Komponen paling hampir dengan titik hentaman akan menghasilkan tegasan yang lebih tinggi berbanding dengan komponen yang lain untuk orientasi jatuhan pada 45^0. Untuk orientasi jatuhan 90^0, komponen di kedudukan paling atas akan menghasilkan tegasan yang paling tinggi. Perubahan sudut orientasi jatuhan daripada 45^0 kepada 90^0 telah menghasilkan tegasan yang tinggi di dalam bebola pateri.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

A	-	Area
E	-	Modulus of elasticity
FEA	-	Finite element analysis
FEM	-	Finite element method
FR-4	-	Flame Retardant 4
g	-	Gravity
G	-	Acceleration peak
h	-	Height
I	-	Internal forces in the structure
ISO	-	International Standards Organization
JEDEC	-	Joint Electron Device Engineering Council
l	-	Length
M	-	Mass of the structure
P	-	Applied external forces
PCB	-	Printed circuit board
t	-	Time
$\ddot{u}$	-	Acceleration of the structure
v	-	Impact velocity
w	-	Width
σ	-	Stress
ε	-	Strain
μ	-	Poisson's ratio
ρ	-	Mass density

LIST OF APPENDICES

CHAPTER I

INTRODUCTION

1.1 Introduction

Electronic products fit into the consumer and variable market segments around the world. Included in the electronic products are computers, digital cameras, calculators, mobile phones, pagers, palm size PCs, smart cards, personal digital assistants (PDAs) and other electronic products. Reliability of these products has become a major concern recently.

These electronic products have the tendencies to being dropped during transportation or customer usage that may cause failure, which leads to malfunction of the products. This dropping event not only caused mechanical failures in the housing of the device but also create electrical failures in the printed circuit board (PCB) assemblies mounted inside the housing due to transfer of energy through PCB supports. The failure mechanism may result from various failure modes such as cracking of circuit board, trace cracking on the board, cracking of solder interconnections between the components and the board, and the component cracks. The primary driver of these failures is excessive flexing of circuit board due to input acceleration to the board created from dropping the electronic product. This flexing of the board causes relative motion between the board and the component mounted on it, resulting in component, interconnects, or board failures. The failure is a strong function of the combination of the board design, construction, material, thickness, and surface finish; interconnect material and standoff height; and component size. Therefore, it is important to design a more robust package and board with better reliability performance due to drop impact.

This project will focus on the dynamic response of a desktop computer's motherboard. For this investigation, the parameters that will be considered are effect of drop orientations and placement of package on the motherboard under the impact loading.

1.2 Objectives of This Study

Dynamic response of motherboard are crucial because they reflect the mechanical behaviour of motherboard, which are closely related to the material properties during drop impact. Before its application as a main component in the computer, it is important to understand its dynamic response as this will affect all the components within the motherboard; solder balls, package and die and to have a modeling tools for simulating the response of the motherboard under the impact load.

Dynamic response in this study is focused on the manner of the motherboard during the impact and after the impact. It also concern the stress propagation in the PCB and acceleration at a selected point in the finite element model. The objectives of this study are:

i. To predict a dynamic response of a computer's motherboard under drop impact loading condition using Finite Element Method (FEM).

ii. To evaluate the state of stresses in the solder balls interconnection which are connect the electronic package to the PCB.

iii. To investigate the effects of drop orientation and package placement on the dynamic response of the motherboard and the state of stresses in the solder ball interconnection.

1.3 Scopes of Study

The aims of this study is to identify dynamic phenomenon in term of free fall drop by modeling the component using the finite element method. The scopes of this study are :

i. Modeling the portion of a motherboard that consists of microelectronic package that is attached to the PCB using solder balls interconnection.

ii. Performing drop impact simulation based on specifications obtained from Joint Electron Device Engineering Council (JEDEC) Standard.

iii. Commercial finite element software is used to model the board and simulate the drop transient of the motherboard from 1 meter height.

iv. Assess the effects of drop orientations of 45^0 and vertical 90^0 relate to the floor and package position at the centre, upper and lower of the motherboard.

1.4 Research Questions

Generally, there are some problems that still remain untouched previously. The problems are:

i. For a given electronics product, how severe would the dynamic loading be in a typical drop impact, and how will the motherboard as a whole respond to the drop impact load?

ii. How will the impact load be transmitted to the electronic package attached to the motherboard?

iii. What would be the state of stresses in the solder balls connects the package to the PCB?

1.5 Rational and Assumption of Study

Before this study is done, there are some rationals and assumptions condition that are used to justify the significant of this work. There are :

i. Motherboards are more prone to being accidentally dropped during transportation and during services. Normally, motherboard is designed to withstand drop to floor.

ii. Drop impact can cause excessive bending of the PCB leading to cracking, and also failures of solder balls interconnection between packages and the motherboard. In this event, the reliability of that product becomes the major concern by the consumer.

iii. Performing actual drop test is quite difficult due to the facilities are very expensive which require high speed camera and sensors. Then, labor extensive need to consider because it requires a lot of manpower in measurement and failure analysis. Not all labor can setup the experiment procedures. Beside that, the experimental setup and analyze data consume a lot of time due to difficulty of that works. That is why experimental work is not included in this work.

1.6 Summary Outline

Chapter 2 provides a review of the literature related to the aims and scope of this study. Topics reviewed include material properties, standard procedure requirement, analysis of few research works and publications related to drop test and simulation of its solder joint realibility.

Chapter 3 describes the steps that are used to develop the finite element model. Techniques to simulate dynamic impact loading conditions also covered in this chapter. Development of an appropriate material model for the motherboard will discussed detail in this chapter.

Chapter 4 performs an analysis of the dynamic response with the various input parameters involved under drop impact loading. The parametric study examines the relative effect of various geometry parameters on the impact loading condition and dynamic response of the motherboard.

Chapter 5 discuss the results of analysis done using FE model developed in Chapter 3. The findings are used to make the conclusions for overall results and finally summarises the main conclusions of this research and for the future applications. Also include about the recommendation to improve and extends this study later on.