

CERTIFICATION OF APPROVAL

Finite Element Analysis of Unstringed Squash Racket Frame to Improve its Impact Strength

by

Noorul Ameen Bin Nasron

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

(AP DR BAMBANG ARIWAHJOEDI)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2009

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

NOORUL AMEEN BIN NASRON

6937

ABSTRACT

This report basically discusses the preliminary research done and basic understanding of the chosen topic, which is **Finite Element Analysis of Unstringed Squash Racket Frame to Improve its Impact Strength**. The objective of the project is to conduct a Finite Element Analysis of the squash racket using the software ANSYS® and to predict the possible failure mode. The challenge in this project is to establish the sites of stress concentration and stress concentration. Research and analysis will be done for the squash racket designed to optimize and improve the performance for the current squash racket. Also the project will analyze the potential modes of failure by undergoing a Finite Element Analysis of the squash racket by using ANSYS® software. This will require design and simulation of the racket to get the perfect condition and to get the improvement in its impact strength.

ACKNOWLEDGEMENTS

First and foremost, the author would like to thank God for guiding him throughout this project and in going through all the challenges and the hurdles.

Next, the author would like to express his heartfelt gratitude to the author's supervisor, AP Dr. Bambang Ariwahjoedi for monitoring and guiding him throughout this project. Without his guidance and patience, the author would not succeed to complete the project.

The author is also grateful towards all lecturers for their help and guidance in their area of specialty. Their experience, knowledge and view on the project have given the author a better outlook in this project.

To all technicians and lab instructors in Mechanical Engineering Departments, thank you for assisting the author in the technical work. Without the cooperation of these individuals, the author would have faced complications throughout the course.

Last but not least, the author is indebted to all individuals who have contributed and have been a great aid to the completion of the project. Hence, the author would like to take the opportunity to express his sincere appreciation and gratitude from the bottom of his heart.

TABLE OF CONTENT

CERTIFICATION OF APPROVAL	i
CERTIFICATION OF ORIGINALITY	ii
ABSTRACT	iii
ACKNOWLEDGEMENTS	iv
TABLE OF CONTENT	v
LIST OF FIGURES	viii
LIST OF TABLES	ix

CHAPTER 1 INTRODUCTION

1.1 Background of Study	1
1.2 Problem Statement	2
1.3 Objectives & Scope of Study	3

CHAPTER 2 LITERATURE REVIEW

2.1 Finite Element Analysis	4
2.2 ANSYS	5
2.3 CATIA	6
2.4 Force	6
2.4.1 The Impulse-Momentum Theorem	6
2.4.1.1 Newton's second law	7
2.4.1.2 Impulse	7
2.4.1.3 Momentum	8
2.4.1.3 Impulse-Momentum Theorem	8
2.5 Stress Concentration	9
2.5.1 Causes	9
2.5.2 Prevention	10
2.5.3 Stress concentration factor	10
2.5.4 Concentration factor calculation	11
2.6 Von-Mises Stress	11

2.7	Aluminium Alloys	12
2.7.1	Flexibility	13
2.7.2	Heat Sensitivity	14

CHAPTER 3 METHODOLOGY

3.1	Project Flow	15
3.2	Gantt Chart	16
3.3	Project Identification	18
3.4	Material Identification	18
3.5	Project Requirement	18
3.6	Project Designation	19
3.7	Squash Racket Modeling	19
3.8	Force Identification	20
3.9	Finite Element Analysis	20
3.9.1	Building the Model	20
3.9.1.1	Geometry	21
3.9.1.2	Material Properties	21
3.9.1.3	Loading	21
3.9.2	Meshing	22
3.9.3	FEA Solution and Output Data	22

CHAPTER 4 RESULT AND DISCUSSION

4.1	Material Identification	23
4.1.1	Flexibility	24
4.1.2	Heat Sensitivity	24
4.2	Design and Modeling	25
4.2.1	Dimensions and Isometric Drawing	25
4.3	Force	27
4.4	Finite Element Analysis	28
4.4.1	Geometry	28

4.4.2	Material Properties	28
4.4.3	Loading.....	28
4.4.4	Meshing.....	29
4.4.5	Static Structural Analysis.....	29
	4.4.5.1 First Attempt.....	30
	4.4.5.2 Second Attempt.....	32
	4.4.5.3 Third Attempt.....	34
4.4.6	Stress Data.....	36
4.5	FEA Data Analysis.....	36

CHAPTER 5 SUMMARY AND CONCLUSION

5.1	Conclusion.....	37
5.2	Recommendation.....	38

REFERENCES	x
-------------------------	----------

LIST OF FIGURES

Figure 3.1: Project flow planning.....	15
Figure 4.1: Promark® PS-S 701	25
Figure 4.2: Schematic diagram of explosion tube.....	26
Figure 4.3: the side view, front view and isometric view of squash racket frame.....	26
Figure 4.4: Meshing for squash racket frame.....	29
Figure 4.5: First attempt for static structural analysis	30
Figure 4.6: Equivalent (von-Mises) stress for squash racket frame.....	31
Figure 4.7: Maximum stress's location in equivalent (von-Mises) stress.....	31
Figure 4.8: Second attempt for static structural analysis.....	32
Figure 4.9: Equivalent (von-Mises) stress for squash racket frame.....	33
Figure 4.10: Maximum stress's location in equivalent (von-Mises) stress.....	33
Figure 4.11: Third attempt for static structural analysis.....	34
Figure 4.12: Equivalent (von-Mises) stress for squash racket frame.....	35
Figure 4.13: Maximum stress's location in equivalent (von-Mises) stress.....	35

LIST OF TABLES

Table 3.1: The Gantt chart for Semester 1.....	16
Table 3.2: The Gantt chart for Semester 2.....	17
Table 4.1: Aluminium Alloy Properties.....	23
Table 4.2: Minimum and Maximum Stress.....	36

REFERENCES

Ian Mckenzie, (1998) The squash workshop. A complete game guide. Seventh Edition

Jack L.groppel., PHD, (1992) High Tech Tennis. Second edition

Paul M. Kurowski, (1999) Finite Element Analysis for Design,

Jianming Jin and Douglas Riley, (2008) Finite Element Analysis of Antennas and Arrays, John Wiley

A Unified approach to The Finite Element Method and Error Analysis Procedure, Academic Press 1999.

D. L. Wang, Z. X. Tian, (2002) Safety Analysis of a Class 3 Component by ANSYS

Benjamin D. Craig, (2005) Material Failure Mode, Part I

John R. Brauer, (1993) What Every Engineer Should Know About Finite Element Analysis

Mike Stanko, Jeffrey M. Steele, (2008) Ansys Advantage, Volume II, Issue 1, Rotating Machinery

Robert A Adey, John M. W. Baynham, Sharon Mellings, Tom Curtin, (2003) Fatigue Life and Crack Growth Prediction Using FEM Data

Cutnell & Johnson, (2007)Physics 7th edition Wiley Asia Student Edition