



UNIVERSITI PUTRA MALAYSIA

**FINITE ELEMENT ANALYSIS OF A PORTAL FRAME SUBJECTED TO
CONCENTRATED IMPULSELOADING**

WONG CHOON YAP

ITMA 2009 7

**FINITE ELEMENT ANALYSIS OF A PORTAL FRAME
SUBJECTED TO
CONCENTRATED IMPULSE LOADING**

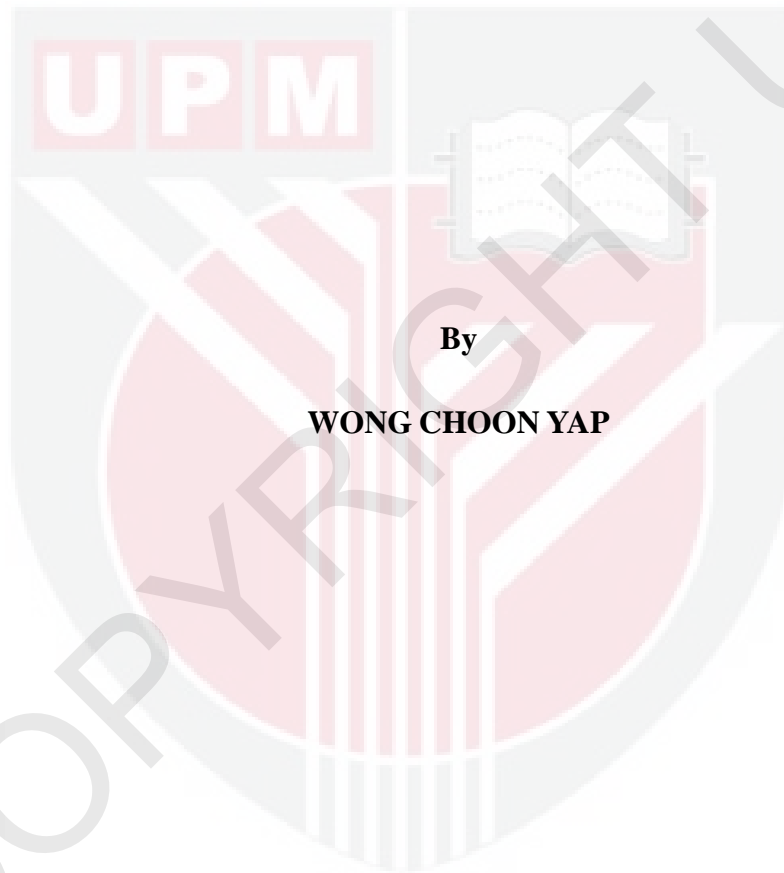
The logo of Universiti Putra Malaysia (UPM) is a shield-shaped emblem. It features a red and white design with a book in the upper right and a stylized 'U' and 'M' in the center. The letters 'UPM' are prominently displayed in a red box at the top left of the shield.

WONG CHOON YAP

**MASTER OF SCIENCE
DEPARTMENT OF MECHANICAL AND
MANUFACTURING ENGINEERING
FACULTY OF ENGINEERING
UNIVERSITI PUTRA MALAYSIA**

2009

**FINITE ELEMENT ANALYSIS OF A PORTAL FRAME SUBJECTED TO
CONCENTRATED IMPULSE LOADING**



By

WONG CHOON YAP

Thesis Submitted to the School of Graduate Studies, Universiti Putra Malaysia,

In Fulfilment of the Requirements for the Master of Science

October 2009

Abstract of thesis presented to the Senate of Universiti Putra Malaysia in
Fulfilment of the requirements for the degree of the Master of Science

**FINITE ELEMENT ANALYSIS OF A PORTAL FRAME
SUBJECTED TO CONCENTRATED IMPULSE LOADING**

By

WONG CHOON YAP

October 2009

Chairman : Wong Shaw Voon, PhD

Faculty : Faculty of Mechanical and Manufacturing Engineering

In general, the portal frame structure which undergoing a large deflection will be studied more on how the structure response upon subjected to dynamic impact loading. In this project we are studying the different material constitutional model to simulate the response effect by using finite element analysis. During the simulation process, the most influent geometry parameters such as width, height, thickness are being simulated by ANSYS LS-DYNA finite element commercial software.

By using the finite element commercial software, the response mode of the portal frame will be captured and compared with experimental results. The output solutions such as stress, strain and displacement effects will be studied and discussed. This finite element impact modeling is being model by generating the SHELL elements as a portal frame and the SOLID element as a projectile. In this project, this portal

frame is made up by a square rectangular frame of aluminium sheet with the portal width of 5.2” and the portal height of 4.6” with both support column were clamp permanently. Meanwhile, the portal wide strip of 0.5” with a thickness of 0.036” will be developed for this finite element modeling analysis.

The portal frame meshing consists of portal frame with 28 elements and 58 nodes, while the impact projectile consists of 143 elements and 245 nodes. Besides, the classifications of impact types also being study by this project in general.

Results obtain in this study shown that by employing a different material model to simulate the impact, we can uses the different material model to predict the large deformation, stress distribution, strain distribution and displacement distribution as comparison to experimental results.

In addition, by adjusting width and height ratio of the portal frame, the results can be used to optimization the best impact resistant structures and this could be used in the more complex portal frame for future study.

Abstrak tesis yang dikemukakan kepada Senat Universiti Putra Malaysia
sebagai memenuhi keperluan untuk Ijazah Sarjana Sains

**ANALISIS UNSUR TERHINGGA RANGKA PORTAL
TERTAKLUK KEPADA DAYA IMPAK TUMPUAN**

Oleh

WONG CHOON YAP

October 2009

Pengerusi : Wong Shaw Voon, PhD

Fakulti : Kejuruteraan Mekanikal dan Pembuatan

Pada umumnya, struktur rangka portal yang melibatkan perubahan bentuk besar akan dikaji dan dibincangkan bagaimana struktur itu berubah akibat daripada daya dinamik impak. Dalam projek ini, kita akan membuat simulasi gerak balas dengan menggunakan analisa unsur terhingga. Dalam proses simulasi ini, parameter yang paling berpengaruh seperti kelebaran, ketinggian, ketebalan akan disimulasikan dengan ANSYS LS-DYNA perisian komputer komersil.

Dengan menggunakan perisian computer komersil unsur terhingga. Gerakbalas rangka portal akan ditangkap dan dibandingkan dengan keputusan eksperimental. Keputusan penyelesaian hasil seperti tekanan, terikan dan jarak sesaran akan dikaji dan dibincangkan. Dalam pemodelan impak ini, unsur SHELL akan digunakan sebagai

rangka portal dan unsur SOLID akan digunakan sebagai peluru impak. Dalam project ini, rangka portal diperbuat daripada gerbang segiempat kepinggan aluminium dengan kelebaran 5.2 inci pada rangka portal dan ketinggian 4.6 inci yang disokong dengan kolum tetap bersebelahan. Ketika itu, lebar jalur dengan 0.5 inci dan ketebalan 0.036 inci akan dibinakan dalam analisa pemodelan unsur terhingga.

Pada rangka portal dibahagi kepada 28 unsur dan 58 nodal, sementara peluru impak mempunyai 143 unsur dan 245 nodal. Selain itu, klasifikasi beberapa impak juga akan dibincangkan secara ringkasnya dalam projek ini.

Keputusan projek menunjukkan bahawa dalam membuat kajian yang melibatkan sesaran besar yang disebabkan oleh impak, kita boleh menggunakan pembolehubah bahan untuk mempengaruhi ubahan bentuk, tekanan, terikan dan sesaran untuk diperbandingkan dengan keputusan eksperimental..

Tambahan pula, dengan mengubah nilai nisbah tinggi and lebar struktur gerbang segiempat, keputusan nisbah boleh digunakan untuk mengkaji struktur rangka yang lebih rumit dan ketahanan impak.

ACKNOWLEDGEMENTS

Author's don't simply go off to their islands to write this technical master thesis. With the help and diverse perspective of Assoc Prof. Dr Wong Shaw Voon the Director of MIROS for Biomekanik & Keselamatan Kenderaan and my co-supervisor Assoc Prof Dr Megat Mohd who provide very a great supervision and guidance.

My gratitude also goes to Assoc Prof Dr Mohd Hamouda and Assoc Prof Dr Yousif who also contribute for this thesis writing and presentation. The author's mission to complete this technical thesis will not be turned from dreams to realistic, without the constructive suggestions and checked the solutions to the problems when draft of this technical thesis.

Author would also like to acknowledge the helpful recommendations and valuable perspective offered by Institute of Advance Technology fellow friends. To who has given a lot of guidance and advises during the publishing of the technical thesis. Author's also thanks to Faculty of Mechanical Engineering staffs for well warming hand during the thesis writing.

Lastly, I would like to appreciate my express in ward and immerse to my parents, my dear wife, and my little sons. Also my brothers and sisters by having great support and motivation during my study of this project. They are granted me to soul and thoughtful mind in completion of my thesis.

This thesis was submitted to the Senate of Universiti Putra Malaysia and has been accepted as fulfilment of the requirement for the degree of Master of Science. The members of the Supervisory Committee were as follows:

Wong Shaw Voon, PhD

Associate Professor
Faculty of Engineering
Universiti Putra Malaysia
(Chairman)

Megat Mohammad Hamdan B Megat Ahmad, PhD

Associate Professor
Faculty of Engineering
Universiti Putra Malaysia
(Member)

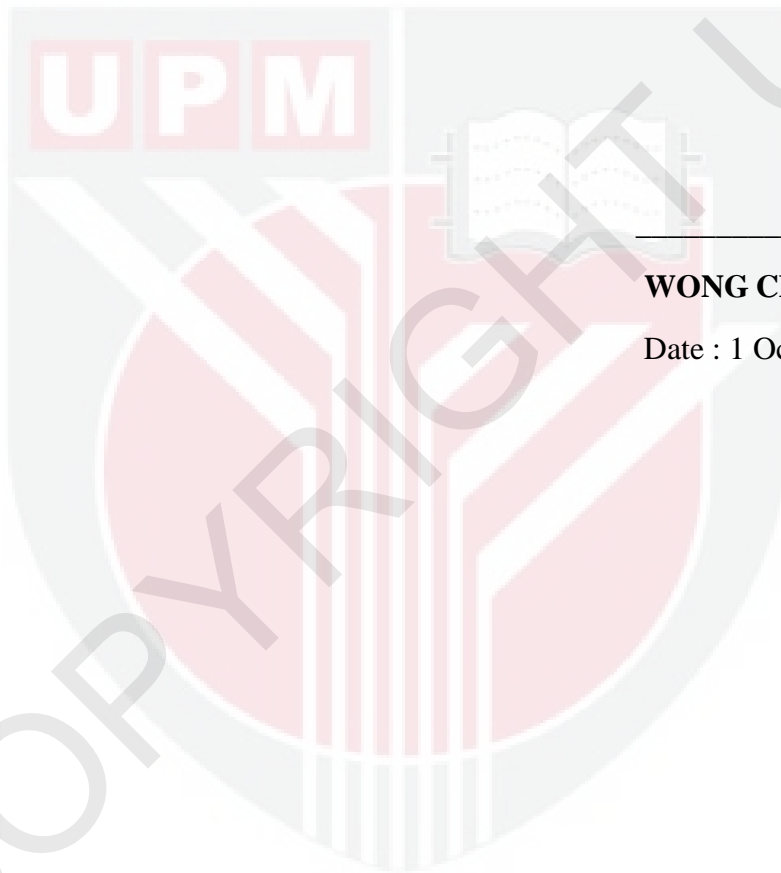


HASANAH MOHD.GHAZALI, PhD

Professor and Dean
School of Graduate Studies
Universiti Putra Malaysia
Date : 11-February-2010

DECLARATION

I declare that the thesis is my original work expect for the quotations and citations which have been duly acknowledged. I also declare that it has not been previously, and is not concurrently, submitted for any other degree at Universiti Putra Malaysia or at any other institution.



WONG CHOON YAP

Date : 1 October 2009



TABLE OF CONTENTS

	Page
ABSTRACT	i
ABSTRAK	iii
AKNOWLEDGEMENTS	v
APPROVAL DEPUTY DEAN	vi
APPROVAL DEAN	vii
DECLARATION	viii
TABLE OF CONTENTS	ix
LIST OF TABLES	xii
LIST OF FIGURES	xiii
LIST OF ABBREVIATIONS	xix
LIST OF SYMBOLS	xxi
CHAPTER	
1 IMPACT AND DEFORMATION	
1.1 Impact and Impulsive	1
1.2 Deflection and Deformation	1
1.3 Impulsive Loading of Structural Frame	2
1.4 Problem Statement	2
1.5 Objective	3
1.6 Why Portal Frame Impact Loading is Important	4
2 LITERATURE REVIEW	
2.1 Dynamic Impact Terminology	6
2.2 Dynamic Impact Loading Classification	8
2.3 Dynamic Impact Loading Comparison	9
2.3.1 Missile Impact	10
2.3.2 Drop Hammer or Power Sledge Impact	10
2.3.3 Explosive Impact	11
2.3.4 Magnetomotive Impact	12
2.4 Dynamic Impact of the Portal Frame	12
2.4.1 Square Portal Frame Subjected to Impact Loading	13
2.4.2 Circular Portal Frame Subjected to Impact Loading	14
2.4.3 Roof Top Portal Frame Subjected to Impact Loading	15
2.5 Plasticity Phenomena of Impact Loading	16
2.5.1 The Plasticity Theory	18
2.5.2 Decomposition of Strain into Elastic and Plastic Model	19
2.5.3 Yield Criterion Model	20
2.5.4 Hardening Rules Model	22
2.5.5 Plastic Flow Rules Model	23
	ix

2.5.6	Bi Linear and Multi Linear Material Model	24
2.5.7	Plastic Kinematic Material Model	26
2.6	Summary	27
3	METHODOLOGY	
3.1	Methodology	28
3.2	Finite Element Software Methodology Flow for ANSYS	30
3.3	Present Portal Frame Methodology	31
3.4	Modeling Methodology of Impact Directions for Portal Frame	32
3.5	Material Properties of the Portal Frame	33
3.6	Methodology Modeling of Material Model for Portal Frame	35
3.7	Methodology Modeling of Pre-Processing for Portal Frame	36
3.8	Methodology of Response Mode for Portal Frame	38
3.8.1	Experimental Response Mode of Directional Impact Loading	40
3.8.2	Influent of Impact Resistant Methodology	41
3.9	Summary	41
4.	FINITE ELEMENT MODELLING AND SIMULATION	
4.1	Finite Element Modelling and Simulation	42
4.2	Pre-Processing Environment and Configuration Modelling	43
4.3	Pre-Processing Creating Nodes for the Portal Frame Modelling	45
4.4	Pre-Processing Creating Elements Type and Material Modelling	47
4.4.1	Material Modelling for the Portal Frame	49
4.5	Pre Processing Mesh Generation for the Portal Frame	52
4.5.1	Boundary Condition Definition for the Portal Frame	55
4.5.2	Contact Definition for the Portal Frame	56
4.5.3	Loading and Initial Velocity Definition for the Portal Frame	58
4.5.4	Time Solution Parameters Modelling	59
4.5.5	Writing Input File	60
4.5.6	Processing Stage	60
4.5.7	Post Processing Procedure	61
4.5.8	Discussion	61
4.6	Editing the Width, Height and Thickness Using High Lighter Editor	62
5.	RESULTS AND DISCUSSION	
5.1	Results and Discussion	64
5.1.1	Portal Frame Mesh Convergence Modelling Discussion	65
5.2	Portal Frame Response Mode Subjected to Directional Impact	68
5.2.1	BKIN Response Mode Subjected to Directional Impact	69
5.2.2	MKIN Response Mode Subjected to Directional Impact	72
5.2.3	BISO and MISO Mode Subjected to Directional	

	Impact Loading	74
5.2.4	Plastic Kinematic Mode Subjected to Directional Impact Loading	75
5.3	Comparison and Mapping Result for BKIN, MKIN and PKIN	77
5.3.1	Displacement Distribution for BKIN at Different Velocity Impact	80
5.3.2	Displacement Distribution for MKIN at Different Velocity Impact	81
5.3.3	Displacement Distribution for PKIN at Different Velocity Impact	82
5.4	Stress Distribution for PKIN at Different Velocity Impact	83
5.5	Strain Distribution for PKIN at Different Velocity Impact	84
5.6	Strain Comparison for PKIN with Experimental Result	86
5.7	Material Model Comparison for PKIN versus Displacement	87
5.7.1	Von-Mises Stress Contour Plot for PKIN Directional Impact	88
5.8	Portal Frame Width, Height and Thickness Influent Factor	89
5.8.1	Influent Factor of Width Ratio for Portal Frame	89
5.8.2	Influent Factor of Height Ratio for Portal Frame	92
5.8.3	Influent Factor of Thickness Ratio for Portal Frame	94
5.8.4	Width Index Ratio Comparison Results	96
5.8.5	Height Index Ratio Comparison Results	97
5.8.6	Height Index Ratio Comparison Results	98
5.8.7	Overall Influent Factor Results for Width, Height and Thickness	99
5.8.8	Overall Results Discussion	101
6.	CONCLUSION AND RECOMMENDATION FOR FUTURE WORK	
6.1	Conclusion	103
6.2	Recommendation for Future Works	104
	REFERENCES	105
	APPENDIX A-1 Typical Engineering Material Properties	109
	APPENDIX A-2 Typical Material Model Properties	110
	APPENDIX A-3 Typical English to Units S.I Conversion Table	111
	APPENDIX A-4 Typical Metric to English Conversion Table	112
	APPENDIX B Numerical Modeling Input File for ANSYS LS-DYNA	113
	APPENDIX C Finite Element Simulation Software References	120
	APPENDIX D Fundamental of Finite Element Method	121
	BIODATA OF THE STUDENT	130