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

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.

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 menglibatkan 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 except 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

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>i</td>
</tr>
<tr>
<td>ABSTRAK</td>
<td>iii</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>v</td>
</tr>
<tr>
<td>APPROVAL DEPUTY DEAN</td>
<td>vi</td>
</tr>
<tr>
<td>APPROVAL DEAN</td>
<td>vii</td>
</tr>
<tr>
<td>DECLARATION</td>
<td>viii</td>
</tr>
<tr>
<td>TABLE OF CONTENTS</td>
<td>ix</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>xii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>xiii</td>
</tr>
<tr>
<td>LIST OF ABBREVIATIONS</td>
<td>xix</td>
</tr>
<tr>
<td>LIST OF SYMBOLS</td>
<td>xxi</td>
</tr>
</tbody>
</table>

## 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
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 75
Impact Loading

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