FINITE ELEMENT ANALYSIS OF FACTORY PORTAL FRAME BY USING ANSYS

PRISCILLIA CHAI LI HSIA

B. ENG (HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this thesis and in my opinion, this thesis is adequate in terms of scope and quality for the award of the Bachelor Degree of Civil Engineering

_____

(Supervisor's Signature)

Full Name : CHENG HOCK TIAN

Position :

Date : 30 May 2019

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is based on my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously or concurrently submitted for any other degree at Universiti Malaysia Pahang or any other institutions.

_____

(Student's Signature)

Full Name : PRISCILLIA CHAI LI HSIA

ID Number : AA15232

Date : 30 May 2019

FINITE ELEMENT ANALYSIS OF FACTORY PORTAL FRAME
BY USING ANSYS

PRISCILLIA CHAI LI HSIA

Thesis submitted in fulfillment of the requirements
for the award of the
B.Eng (Hons) Civil Engineering

Faculty of Civil Engineering and Earth Resources
UNIVERSITI MALAYSIA PAHANG

MAY 2019

This study is especially dedicated to my beloved family, project supervisor, and my friends for their continuous support and care throughout my studies.

ACKNOWLEDGEMENTS

First of all, I would like to give sincere gratitude to my Supervisor, Dr. Cheng Hock Tian. Along my research project, he had provided a lot of guidance and help to me in order to finish my research in time. Besides, he had also provided several examples and tutorials for me to learn about the ways to use the ANSYS software.

Furthermore, I truly appreciated the supports which have been given by my family members throughout the final year project. I would also like to give a million thanks to my course mates for helping me out by providing some suggestions and advices, so that I am able to complete my final year project. Without their help, I am unable to complete my final year project smoothly.

In addition, I am thankful to UNIVERSITY MALAYSIA PAHANG for providing me an opportunity to do my research and final year project. I feel contented to be able to learn using ANSYS software right before graduation. This is a very useful experience and skill which I can apply in my future career.

ABSTRACT

In this research, a factory portal frame has been analysed by using ANSYS software. This research is to determine the strain, stress, maximum deflection, deformation and also checking the structure according to Eurocode 3. Major of the buildings constructed in Malaysia are reinforced concrete buildings compared to steel frame buildings. However, steel is commonly used in portal frame design. Steel structure design software is very important to help civil engineers in finite element analysis. The type of material and the geometry for the structure was satisfied in in cases it passed all the designing for tensile, buckling and compression. The values for the input variables are generated randomly by using Monte Carlo Simulation with given mean values and standard deviation or as prescribed samples using Response Surface Method. From the result of simulation, we can know the behaviour of the steel frame structure under the input parameter that applied. Then, from the probabilistic analysis, we collect the result of cumulative distribution function, the histogram plot for input and output parameter, sensitivity plot and simple history plot for all parameter.

ABSTRAK

Dalam kajian ini, sebuah bingkai keluli kilang telah dianalisis menggunakan perisian ANSYS. Kajian ini adalah untuk menentukan ketegangan, tekanan, pesongan maksimum, ubah bentuk dan juga menyemak struktur mengikut Eurocode 3. Kebanyakan bangunan yang dibina di Malaysia adalah bangunan konkrit bertetulang berbanding bangunan bingkai keluli. Manakala, keluli selalu digunakan dalam reka bentuk bingkai keluli. Perisian reka bentuk struktur keluli adalah sangat penting untuk membantu jurutera awam dalam analisis unsur terhingga. Jenis bahan dan geometri untuk struktur itu berpuas hati dalam kes-kes yang melepasi semua reka bentuk untuk tegangan, geseran dan pemampatan. Nilai-nilai untuk pemboleh ubah masukan dijana secara rawak dengan menggunakan Simulasi Monte Carlo dengan nilai min dan sisihan piawai atau sampel yang ditetapkan menggunakan Kaedah Surface Response. Dari hasil simulasi, kita dapat mengetahui kelakuan struktur bingkai keluli di bawah parameter input yang digunakan. Kemudian, dari analisis probabilistik, kami mengumpul hasil fungsi pengedaran kumulatif, plot histogram untuk input dan output parameter, plot kepekaan dan plot sejarah mudah untuk semua parameter.

TABLE OF CONTENTS

DECLARATION OF THESIS AND COPYRIGHT i

SUPERVISOR'S DECLARATION ii

STUDENT'S DECLARATION iii

THESIS TITLE iv

DEDICATION v

ACKNOWLEDGEMENTS vi

ABSTRACT vii

ABSTRAK viii

TABLE OF CONTENTS ix

LIST OF TABLES xiv

LIST OF FIGURES xiv

LIST OF SYMBOLS xix

LIST OF ABBREVIATIONS xx

CHAPTER 1 INTRODUCTION

1.1 Introduction 1

1.2 Problem Statement 2

1.3 Objective 2

1.4 Scope of Study 2

1.5 Expected Outcome 3

1.6 Significance of Study 3

CHAPTER 2 LITERATURE REVIEW

2.1 Portal Frame 4

2.1.1 Introduction of Portal Frames 4

2.1.2 Anatomy of Typical Portal Frame 4

2.1.3 Types of Portal Frames 5

2.1.4 Frame Analysis at Ultimate Limit State 5

2.1.4.1 Plastic Analysis 6

2.1.4.2 Elastic Analysis 6

2.1.5 Actions 6

2.1.5.1 Permanent Actions 7

2.1.5.2 Variable Actions 7

2.1.5.2.1 Wind Load 7

2.1.5.2.2 Roof Loads 7

2.1.6 Connections 7

2.2 ANSYS Program 8

2.2.1 Introduction 8

2.2.2 Portal Frame ANSYS Analysis 9

2.3 Analysis Method 9

2.3.1 Finite Element Analysis 9

CHAPTER 3 METHODOLOGY

3.1 Introduction 11

3.2 Preprocessing 13

3.2.1 Title of Project 13

3.2.2 Set Codes and Units 13

3.2.3 Defining Element Types 13

3.2.4 Defining Material 15
3.2.5 Defining Section 16
3.2.6 Defining Member Properties 17
3.2.7 Defining Beam & Shell Properties 17
3.2.8 Defining Keypoints and Elements 18
3.2.9 Creating of Model 19
3.3 Solution Phase 20
3.3.1 Define Analysis Type 20
3.3.2 Apply Constraints 21
3.3.3 Apply Loads 22
3.3.4 Solving 23
3.4 Postprocessing 23
3.4.1 Read Results 23
3.4.2 Code Checking 24
3.4.2.1 Tension 24
3.4.2.2 Compression 25
3.4.2.3 Bending Moment 26
3.4.2.4 Shear 27
3.4.2.5 Bending and Shear 28
3.4.2.6 Bending and Axial Force 29
3.4.2.7 Bending, Axial and Shear 30
3.4.2.8 Compression Buckling 31
3.4.2.9 Lateral Buckling 32
3.4.2.10Bending and Compression Buckling 33
3.4.3 Result Obtained 34
3.4.3.1 Force and Moment 34

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1 Introduction 39
4.1.1 Determination of Process Parameters 41

4.2 Random Input Variables 42

4.2.1 PDF & CDF of Input Randon Variable Applied Load (Loading 1) 42

4.2.2 PDF & CDF of Input Variable Density 43

4.2.3 PDF & CDF of Input Random Variable Elastic Modulus 43

4.2.4 PDF & CDF of Input Random Variable Poisson's Ratio 44

4.2.5 PDF & CDF of Input Random Variable Temperature 45

4.2.6 PDF & CDF of Input Random Variable Wind Load (Loading 2) 46

4.3 PROBABILISTIC ANALYSIS RESULT 47

4.3.1 Statistic of the Probabilistic Result 47

4.3.2 Sample History Plots 49

4.3.3 Histogram Plots 53

4.3.3.1 Histogram of Input Parameter 54

4.3.3.2 Histogram of Output Parameter 57

4.3.4 Cumulative Distribution Function Plots 59

4.3.5 Sensitivity Plots 62

4.3.6 Linear Correlation Coefficients 65

4.3.7 Spearman Rank Order Correlation Coefficients 67

CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS

5.1 Introduction 69

5.2 Conclusion 69

5.3 Recommendation 70

REFERENCES 71

APPENDICES

A1 Log File 73

LIST OF TABLES

Table No.	Title	Page
3.1	Specification of factory portal frame	11
4.1	Statistical analysis of various input random variables for Probabilistic Design	40
4.2	Statistical of Random Input Variables	47
4.3	Statistical of Random Output Variables	48
4.4	Linear Correlation Coefficients between Input Variables	65
4.5	Linear Correlation Coefficients between Input and Output Variables	66
4.6	Linear Correlation Coefficients between Output Variables	66
4.7	Spearman Rank Order Correlation Coefficients between Input Variables	67
4.8	Spearman Rank Order Correlation Coefficients between Input And Output Variables	67
4.9	Spearman Rank Order Correlation Coefficients between Output Variables	67

LIST OF FIGURES

Figure No.	Title	Page
2.1	Components of Portal Frame	5
2.2	Various Types of Portal Frames	5
2.3	Bending Moment Diagram Resulting from the Plastic Analysis of A Symmetrical Portal Frame under Symmetrical Loading	6
2.4	Eaves Connections	8
2.5	Apex Connections	8
3.1	Flow Chart of the Research Process	12
3.2	Utility Bar	13
3.3	Change Title	13
3.4	Activate CIVILFEM	13
3.5	Select Eurocode 3	14
3.6	Select International System Units	14
3.7	Selection of Element Type	15
3.8	New Material	16
3.9	Steel Cross Section	16
3.10	Member Properties I	17
3.11	Beam 1	18
3.12	Create Keypoints in Active Coordinate System	19
3.13	Coordinates of Total 65 Keypoints	19
3.14	Element Attributes	20
3.15	Model of Factory Portal Frame	20
3.16	Type of Analysis	21

3.17	Apply U, Rot on Notes	21
3.18	Displacement Constraint Applied to the Model	22
3.19	Point Loads and Wind Load Applied to Model	22
3.20	Solve Current Load Step	23
3.21	Read Results by Load Step Number	23
3.22	Check Model Tension Results by Eurocode 3	24
3.23	Graph Steel Results for Tension	24
3.24	Tension Checking Results	25
3.25	Check Model Compression Results by Eurocode 3	25
3.26	Graph Steel Results for Compression	25
3.27	Compression Checking Results	25
3.28	Check Model Bending Moment Results by Eurocode 3	26
3.29	Graph Steel Results for Bending Moment	26
3.30	Bending Moment Checking Results	26
3.31	Check Model Shear Results by Eurocode 3	27
3.32	Graph Steel Results for Shear	27
3.33	Shear Checking Results	27
3.34	Check Model Bending and Shear Results by Eurocode 3	28
3.35	Graph Steel Results for Bending and Shear	28
3.36	Analysis of Compression Buckling	28
3.37	Compression Buckling Graph Result	29
3.38	Bending and Shear Checking Results	29
3.39	Lateral Buckling Graph Result	29
3.40	Check Model Bending, Axial and Shear Results by Eurocode 3	30
3.41	Graph Steel Results for Bending and Axial Force	30

3.42	Bending, Axial and Shear Checking Results	30
3.43	Check Compression Buckling Results by Eurocode 3	31
3.44	Graph Steel Results for Compression Buckling	31
3.45	Compression Buckling Checking Results	31
3.46	Check Lateral Buckling Results by Eurocode 3	32
3.47	Graph Steel Results for Lateral Buckling	32
3.48	Lateral Buckling Checking Results	32
3.49	Check Bending and Compression Buckling Results by Eurocode 3	33
3.50	Graph Steel Results for Bending and Compression Buckling	33
3.51	Bending and Compression Buckling Checking Results	33
3.52	Graph Force and Moment Results for Axial Force X	34
3.53	Axial Force X Diagram	34
3.54	Graph Force and Moment Results for Shear Force Y	34
3.55	Shear Force Y Diagram	35
3.56	Graph Force and Moment Results for Shear Force Z	35
3.57	Shear Force Z Diagram	35
3.58	Graph Force and Moment Results for Torsion Moment X	35
3.59	Torsion Moment X Diagram	36
3.60	Graph Force and Moment Results for Bending Moment Y	36
3.61	Bending Moment Y Diagram	36
3.62	Graph Force and Moment Results for Bending Moment Z	37
3.63	Bending Moment Z Diagram	37
4.1	Model Geometry and Finite Element Mesh	39
4.2	PDF & CDF of Input Random Variable Applied Load	41
4.3	PDF & CDF of Input Random Variable Density	42

4.4	PDF & CDF of Input Random Variable Elastic Modulus	43
4.5	PDF & CDF of Input Random Variable Poisson's Ratio	44
4.6	PDF & CDF of Input Random Variable Temperature	45
4.7	PDF & CDF of Input Random Variable Wind Load	46
4.8	Mean Values History for Output Parameter MAXIMUMDEFLECTION	49
4.9	Standard Deviation History for Output Parameter MAX_DEFLECTION	49
4.10	Mean Value History for Output Parameter MAX_DEFLECTION	50
4.11	Standard Deviation History for Output Parameter MAXI_DEFLECTION	51
4.12	Sample Values for Output Parameter MAXIMUM DEFLECTION	51
4.13	Sample Values for Output Parameter MAX_DEFLECTION	52
4.14	Histogram of Input Variable Applied Load (Loading 2)	53
4.15	Histogram of Input Variable Density	54
4.16	Histogram of Input Variable Elastic	54
4.17	Histogram of Input Variable POISSON	55
4.18	Histogram of Input Variable TEMP	55
4.19	Histogram of Input Variable Wind Load	56
4.20	Histogram of Output Parameter MAXIMUMDEFLECTION	57
4.21	Histogram of Output Parameter MAX_DEFLECTION	57
4.22	CDF of Input Variable Applied Load (Loading 1)	59
4.23	CDF of Input Variable Density	59
4.24	CDF of Input Variable POISSON	59
4.25	CDF of Input Variable Temperature	60
4.26	CDF of Input Variable Windload (Loading 2)	61

4.27	Linear Correlation Sensitivity Plot for MAXIMUMDEFLECTION	62
4.28	Rank-Order Correlation Sensitivity Plot for MAXIMUMDEFLECTION	63
4.29	Sensitivity Plot for MAX_DEFELCTION	64

LIST OF SYMBOLS

d	Outside Diameter
t	Thickness
d/t	Ratio for Local Buckling
A	Area of section
I	Moment of inertia
W_{pl}	Plastic modulus
i	Radius of gyration
N	Axial load
V	Shear force
M	Moment
I_T	Torsional Constants
$\gamma M0$	Partial factor for resistance of cross-sections whatever the class is
$\gamma M1$	Partial factor for resistance of members to instability assessed by member checks
λ	Slenderness value
$\emptyset$	Value to determine the reduction factor
X	Reduction factor
L_{cr}	Buckling Length
K_{zy}	Interaction factor

LIST OF ABBREVIATIONS

2D	Two Dimensional
3D	Three Dimensional
CIVIFEM	Civil Finite Element Method
LatBuck	Lateral Buckling
ChckAxis	Check Axis
BMSHPRO	Beam and Shell Properties
CS	Coordinate System
LS	Load Step
DOF	Degree of Freedom
PRES	Pressure
GAUS	Gaussian
DENS	Density
ELASTIC	Elastic modulus
POISON	Poison ratio
LOAD	Point load
WINDLOAD	Wind load
TEMP	Temperature
PDF	Probabilistic density function
CDF	Cumulative distribution function
MAXIMUMDEFLECTION /MAX_DEFLECTION	Maximum Deflection

CHAPTER 1

INTRODUCTION

1.1 General

Multiple types of frames are used in steel construction industries. It is noted that steel portal frame is favoured frame that is used to build single-storey industrial buildings. There are many advantages of steel portal frame to be used compared to structural portal frame. One of it is steel portal frame can be erected and built easily. The steel structural elements of portal frame can be fabricated before being transported to construction site. Besides, it is more cost effective compared to structural portal frame, thus, saving the clients lots of money.

In this research, a model of factory steel portal frame will be proposed and modelled by using the software ANSYS+CivilFEM 12.0 in three dimensions, a finite element analysis calculation will be performed through the software to obtain the deflection, deformation and moments of the factory steel portal frame. The calculation of a factory steel portal frame can be done easily by manual calculation. However, a more complicated factory steel portal frame requires the help of software. Otherwise, it would be too time consuming to analyse the factory steel portal frame by manual calculation.

CivilFEM for ANSYS is an advanced, comprehensive and notable finite element analysis and design software that is suitable for civil engineering related projects. Apart from the software ANSYS, there are other software such as STAAD PRO, TEKLA, SAP 2000 and many more which can be used for civil engineering related projects.

1.2 Problem Statement

In modern construction, steel structure is used for almost all type of structure such as heavy industrial buildings, single-storey buildings and many more. Steel portal frame is a metal structure which is made of structural steel components connected with each other to carry loads and provide rigidity. There are several methods that can be used to analyse a factory steel portal frame. However, in order to make the procedure easier, the finite element method is used through ANSYS software.

Before construction process, it is necessary to make sure the model passes the designing phase by using the software ANSYS. In this research, the deformation, deflection and moments of the model will have to be determined from the software ANSYS. The model will also have to pass the code checking according to Eurocode 3.

1.3 Objective

Research objectives play an important role for every project in order for the research to be conducted well. It serves as a guideline for researchers to follow and achieve the final results required. The objectives of this research are :-

i. To determine the behavior of factory steel portal frame.
ii. To determine the factory steel portal frame passes all the code checking.

1.4 Scope of Study

The software ANSYS will be used to analyse this research of factory steel portal frame. Modeling of the design will be done by using ANSYS. The uniformly distributed load will be assumed as point loads as the load will be transferred to the modes as point loads. The point loads include permanent actions and variable actions. However, wind load will be applied to one side of the model. The wind load will be applied on the main structure. The main steel work of the structure is the portal frame while the secondary steel works of the structure includes purlins, bracings and side rails. The deformation, deflection and code checking will be assessed.

1.5 Expected Outcome

This research claims to find out the behaviors and passing of code checking of factory portal frame. The behaviors are deformation, deflection, tension checking, compression checking and lateral torsional buckling checking according to Eurocode 3.

1.6 Significance of Study

This research will provide information regarding the steps and results of analyzing a factory steel portal frame by using ANSYS. The maximum deformation, deflection and moment of the structure can be determined. It is important to do analysis on the structure before constructing it to ensure the structure is safe against failure. Besides, it also helps us to save cost and time in terms of designing and selecting suitable dimension of material to be used for construction. Thus, it is important to learn on how to design steel portal frame in Malaysia as it is being practiced widely by developed countries in building single-storey buildings. With this skill, it will improve the development of Malaysia.

REFERENCES

ANSYS. (January, 2018). 4.188 BEAM188 3-D Linear Finite Strain Beam. Retrieved from ANSYS: http://www.ansys.stuba.sk/html/elem_55/chapter4/ES4-188.htm

BS EN 1991-1-1:. (2002). Eurocode 1: Actions on structures. General actions. Densities, self-weight, imposed loads for buildings , BSI.

Davidson, B., & W. Owens, G. (2012). STEEL DESIGNERS' MANUAL. BLACKWELL PUBLISHING LTD.

Duoc, T., James B.P., L., Tiku T. , T., R. Mark, L., Yixiang, X., Steven, M., & Wei, S. (2013). Effect of serviceability limits on optimal design of steel portal frames. Journal of Constructional Steel Research, 74-84.

Elsayed, M., Mohamed, E.-H., Hamdy, A.-E., & Mohamed, O. (2010). Finite element analysis of beam-to-column joints. Alexandria Engineering Journal, 91-104.

G Lackshmi, N. (2009). Finite Element Analysis. BS Publications.

M.T., R.-L., & Jose, S.-S. (2014). Analysis of wind action on unique structures with application to Seville. Engineering Structure.

Madsen, J. J. (6 January, 2005). Which is the better building material? Concrete or Steel? Retrieved from Buildings Smarter Facility Management: https://www.buildings.com/article-details/articleid/2511/title/which-is-thebetter-building-material-concrete-or-steel-/viewall/true

P.J., M., R.P., D., M.W. , B., & A.H. , B. (2008). Design of steel portal frame buildings for fire safety. Journal of Constructional Steel Research, 1216-1224.

Ross, M., James, B., Tiku, T., Duoc, T., & Wei Sha. (2014). Optimal design of longspan steel portal frames using fabricated beams. Journal of Constructional Steel Research, 104-114.

SHARCNet. (January, 2018). BEAM 188. Retrieved from SHARCNet: https://www.sharcnet.ca/Software/Ansys/16.2.3/enus/help/ans_elem/Hlp_E_BEAM188.html

Steel Construction. (2014). Retrieved from Steel Construction: http://www.steelconstruction.info/Portal_frames

University of Alberta - ANSYS Tutorials. (2001). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys/

-, J. J., -, F. X., -, W. Z., -, D. X., & -, Q. D. (2012). Static Performance Analysis of Large Span Portal Frame with Variable Section. International Journal of Digital Content Technology and Its Applications, 6(12), 73–82. https://doi.org/10.4156/jdcta.vol6.issue12.9

Caprani, C. (2010). Plastic Analysis 3rd Year Structural Engineering, (January), 1–129.

Carley, K. M., Kamneva, N. Y., & Reminga, J. (2004). Response surface methodology. CASOS Technical Report, (October), 1–26. https://doi.org/10.1002/wics.73

Ding, Y., Song, X., & Zhu, H. (2017). Probabilistic progressive collapse analysis of steel frame structures against blast loads. Engineering Structures, 147, 679–691. https://doi.org/10.1016/j.engstruct.2017.05.063

El-Heweity, M. M. (2012). Behavior of portal frames of steel hollow sections exposed to fire. Alexandria Engineering Journal, 51(2), 95–107. https://doi.org/10.1016/j.aej.2012.06.004

Fu, F. (2009). Progressive collapse analysis of high-rise building with 3-D finite element modeling method. Journal of Constructional Steel Research, 65(6), 1269– 1278. https://doi.org/10.1016/j.jcsr.2009.02.001

Goswami, S., Ghosh, S., & Chakraborty, S. (2016). Reliability analysis of structures by iterative improved response surface method. Structural Safety, 60, 56–66. https://doi.org/10.1016/j.strusafe.2016.02.002