

**FINITE ELEMENT ANALYSIS OF WAREHOUSE STEEL
STRUCTURE USING ANSYS**

MUHAMMAD SHAFIQ BIN NOOR ZAMMIZI

**B.ENG (HONS.) CIVIL ENGINEERING
UNIVERSITY MALAYSIA PAHANG**

SUPERVISOR'S DECLARATION

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

Signature :

Name of Supervisor : DR. CHENG HOCK TIAN

Position : Undergraduate Research Project Supervisor

Date : 28 DECEMBER 2016

FINITE ELEMENT ANALYSIS OF WAREHOUSE STEEL STRUCTURE
USING ANSYS

MUHAMMAD SHAFIQ BIN NOOR ZAMMIZI

Report submitted in partial fulfillment of the requirements
For the award of degree of
B.Eng. (Hons.) Civil Engineering

Faculty of Civil Engineering & Earth Resources
UNIVERSITI MALAYSIA PAHANG

JANUARY 2017

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name : MUHAMMAD SHAFIQ BIN NOOR ZAMMIZI

ID Number : AA 12193

Date : 28 DECEMBER 2016

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

ACKNOWLEDGEMENT

My most gratitude to Allah S.W.T, the Almighty for giving me this gave the great chance to enhance my knowledge and to complete this research. May the peace and blessings be upon prophet Muhammad (SAW).

First and foremost, I would love to offer and convey my sincere gratitude to my supervisor, Dr. Cheng Hock Tian for his endless support and guidance for me to finish this study. It is a whole new experience for me as I get to learn in handling ANSYS software, something that I never encounter before. Being able to finish this thesis is indeed an achievement and it is something that I will cherish forever in my life.

Next, to my parents for their endless support too. Without them, there is no way that I would be able to finish this thesis as it requires materials, money and so on. It is my parents who provide them all and I am truly grateful to be awarded with such blessings.

I am grateful to thank UNIVERISITY MALAYSIA PAHANG that provided me a chance to complete my study and final year project. I also want to thank you to my course mates and seniors that helped me a lot in using ANSYS software, sharing knowledge to me. Just to let them know, I appreciate all of it.

Lastly, I feel happy to complete this research as I had used a lot of time in completing this research. The hard work is paid off after I see the results come out is satisfied. Thank you very much.

TABLE OF CONTENTS

DECLARATION OF THESIS AND COPYRIGHT	i
SUPERVISOR'S DECLARATION	ii
STUDENT'S DECLARATION	iii
DECICATION	v
ACKNOWLEDGEMENTS	iv
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xii
LIST OF FIGURES	xiii
LIST OF SYMBOLS	xvii
LIST OF ABBREVIATIONS	xviii

CHAPTER 1 INTRODUCTION

1.1	Introduction	1
1.2	Problem Statement	2
1.3	Objective	2
1.4	Scope of Study	3
1.5	Expected Outcome	3
1.6	Significance of Study	3

CHAPTER 2 LITERATURE REVIEW

2.1	Introduction	4
2.2	Model of Warehouse Steel Structure	5
2.3	Probabilistic Analysis in Finite Element Modelling	6
2.4	Finite Element Analysis	7
2.5	Probabilistic Design	7
2.5.1	General	8
2.5.2	Probabilistic Design System	8
2.5.3	Distribution of Probability Design Variables	9
2.5.4	Monte Carlo Simulation	10
2.5.5	Monte Carlo Simulation and Latin Hypercube Simulation	11

CHAPTER 3 METHODOLOGY

3.1	Introduction	12
3.2	Preprocessing: Defining the Problem	13
3.2.1	Modelling 3D warehouse steel structure	13
3.2.2	Entering Title	14
3.2.3	Defining CivilFEM	14
3.2.4	Defining Beam Element Types for Code Checking	15
3.2.5	Defining Material Properties	16
3.2.6	Defining Cross sections and Member Properties	16
3.2.7	Entering Key Points	18
3.2.8	Setup All Force, Wind Load and Support	20
3.2.9	Result by Load Step Number	21
3.2.10	Reaction Forces	22
3.2.11	Deflection	23
3.2.12	Read Result	25

3.2.13	Force & Moment	25
3.2.14	Result by Eurocode 3	27
3.2.15	Force and Moment List by Load Step	34

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1	Introduction	37
4.1.1	Determination of Process Parameters	39
4.2	Random Input Variables	40
4.2.1	Random Input Variables	40
4.2.2	PDF & CDF of Input Variable Density	41
4.2.3	PDF & CDF of Input Random Variable Elastic Modulus	42
4.2.4	PDF & CDF of Input Random Variable Poisson's Ratio	43
4.2.5	PDF & CDF of Input Random Variable Temperature	44
4.2.6	PDF & CDF of Input Random Variable Wind Load	45
4.3	PROBABILISTIC ANALYSIS RESULT	46
4.3.1	Statistic of the Probabilistic Result	47
4.3.2	Sample History Plots	48
4.3.3	Histogram Plots	52
4.3.3.1	Histogram of Input Parameter	53
4.3.3.2	Histogram of Output Parameter	57
4.3.4	Cumulative Distribution Function Plots	58
4.3.5	Sensitivity Plots	62
4.3.6	Linear Correlation Coefficients	65
4.3.7	Spearman Rank Order Correlation Coefficients	66

CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS

5.1	Introduction	68
5.2	Conclusion	68
5.3	Recommendation	70

REFERENCES	71
-------------------	----

APPENDICES

A1	Stiffness Matrix and Displacement of Portal Frame	73
A2	Element Forces of Steel Warehouse	80
A3	Eurocode Checking of Steel Warehouse Frame	83
B1	Log File	87

LIST OF TABLES

Table No.	Title	Page
3.1	Coordinate of node	18
3.2	Beam Element Force and Moment	35
3.3	Stress and Strain	36
4.1	Statistical analysis of various input random variables for Probabilistic Design	39
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
1.1	Side View and Dimension of 3D Portal Frame	2
1.2	Top View and Dimension of 3D Portal Frame	2
2.1	Real model warehouse steel frame	5
2.2	Example of structure warehouse steel frame by ANSYS software	6
2.3	Example of model for design steel warehouse	7
2.4	Probabilistic Analysis Procedures	10
3.0	Flow Chart of Research Process	13
3.1	Change Title	14
3.2	Activate CIVILFEM	14
3.3	CIVILFEM Setup Options	15
3.4	Beam Element Types for Code Checking	15
3.5	New Material	16
3.6	Steel Cross Section	17
3.7	Member Properties 1	17
3.8	Beam and Shell Properties	18
3.9	Creates Nodes in Active Coordinate System	19
3.10	Apply U, ROT on Nodes	19
3.11	Apply F/M on Nodes	20
3.12	Apply PRESS on Nodes	20
3.13	Model with load and support and solution is done	21
3.14	Load step number	21
3.15	Graph Force and Moment Results	22

3.16	Plot Deformed Shape	22
3.17	Deformed Shape	23
3.18	Contour Nodal Solution Data	24
3.19	Contour Plot of Deflection	24
3.20	Read Results by Load Step Number	25
3.21	Graph Force and Moment Results	25
3.22	Axial Force Diagram	26
3.23	Shear Force Diagram	26
3.24	Bending Moment Diagram	27
3.25	Check Model Results by Eurocode 3	28
3.26	Graph Steel Results	28
3.27	Tension Checking Results	29
3.28	Compression Checking Results	30
3.29	Bending Moment Checking Results	30
3.30	Shear Checking Results	31
3.31	Bending Shear Checking Results	31
3.32	Bending and Axial Checking Results	32
3.33	Bending Axial and Shear Checking Results	32
3.34	Bending Buckling Checking Results	33
3.35	Compression Buckling Checking Results	33
3.36	Bending & Compression Buckling Checking Results	34
3.37	List Force and Moment	34
3.38	List Stress & Strain Result	36
4.1	Model Geometry and Finite Element Mesh	38

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 Value History for Output Parameter MAXIMUMDEFLECTION	49
4.9	Standard Deviation History for Output Parameter MAXIMUMDEFLECTION	49
4.10	Mean Value History for Output Parameter MAX_DEFLECTION	50
4.11	Standard Deviation History for Output Parameter MAX_DEFLECTION	51
4.12	Sampled Values for Output Parameter MAXIMUMDEFLECTION	51
4.13	Sample Values for Output Parameter MAX_DEFLECTION	52
4.14	Histogram of Input Variable APPLIEDLOAD	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	56
4.19	Histogram of Input Variable WINDLOAD	56
4.20	Histogram for Output Parameter MAXIMUMDEFLECTION	57
4.21	Histogram for Output Parameter MAX_DEFLECTION	58
4.22	CDF of Input Variable APPLIEDLOAD	59
4.23	CDF of Input Variable DENSITY	60
4.24	CDF of Input Variable POISON	60

4.25	CDF of Input Variable TEMPERATURE	61
4.26	CDF of Input Variable WINDLOAD	61
4.27	Linear Correlation Sensitivity Plot for MAXIMUMDEFLECTION	62
4.28	Rank- Order Correlation Sensitivity Plot for MAX_DEFLECTION	63
4.29	Sensitivity Plot for MAX_DEFLECTION	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
γM_0	Partial factor for resistance of cross-sections whatever the class is
γM_1	Partial factor for resistance of members to instability assessed by member checks
λ	Slenderness value
ϕ	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

FINITE ELEMENT ANALYSIS OF WAREHOUSE STEEL STRUCTURE
USING ANSYS

MUHAMMAD SHAFIQ BIN NOOR ZAMMIZI

Report submitted in partial fulfillment of the requirements
For the award of degree of
B.Eng. (Hons.) Civil Engineering

Faculty of Civil Engineering & Earth Resources
UNIVERSITI MALAYSIA PAHANG

JANUARY 2017

ABSTRACT

Frame is one of the important elements of this structure. Unfit or improper design can lead to the failure of a structure. The warehouse steel frame was analyzed by manual calculation and also design in ANSYS software. There are several methods can be used to analyses the behavior of steel warehouse frame. However, among all these methods, finite element method and probabilistic method will be a very effective method to obtain the strength and behavior of steel structure. This analysis is to determine the behavior of frame design for the warehouse structure. This research will mainly focus on the steel frame of warehouse structure. The yield strength of steel jacket that used is 500 N/mm^2 which is the strongest steel. The value for the input variables are generated randomly by using Monte Carlo Simulation or as prescribed samples using Response Surface Methods. 1000 simulations had been made to make sure the analysis is more precise and accurate. Here, the study was conducted to prove that a steel warehouse under the existence of randomness and uncertainty can be analyzed by apply probabilistic finite element analysis using ANSYS.

ABSTRAK

Kerangka adalah salah satu elemen penting dalam struktur ini. Reka bentuk tidak layak atau tidak betul boleh menyebabkan kegagalan struktur. Rangka gudang keluli dianalisis dengan pengiraan manual dan juga mereka bentuk dalam perisian ANSYS. Terdapat beberapa kaedah yang boleh digunakan untuk menganalisis kelakuan kerangka gudang keluli. Walau bagaimanapun, di kalangan semua kaedah ini, kaedah unsur terhingga dan kaedah kebarangkalian akan menjadi kaedah yang sangat berkesan untuk mendapatkan kekuatan dan kelakuan struktur keluli. Analisis ini adalah untuk menentukan kelakuan reka bentuk bingkai untuk struktur gudang. Kajian ini akan memberi tumpuan pada rangka keluli gudang. Kekuatan alah gudang keluli yang digunakan ialah 500 N/mm^2 yang keluli yang paling kuat. Nilai untuk pemboleh ubah input yang dijana secara rawak dengan menggunakan Monte Carlo Simulasi atau sebagai sampel ditetapkan menggunakan Kaedah Respon Permukaan. 1000 simulasi telah dibuat untuk memastikan analisis yang lebih tepat dan tepat. Di sini, kajian ini dijalankan bagi membuktikan bahawa keluli jacket yang mengalami masalah kerawakan dan ketidakpastian boleh dianalisis dengan menggunakan cara analisis terhingga dan analisis kebarangkalian di bawah penolongan perisian komputer ANSYS.

CHAPTER 1

INTRODUCTION

1.1 GENERAL

It is undeniably true that nowadays, the evolution of technology has made it possible to finish any works so much easier. In civil engineering for example, most engineers now are using software to design structures. Software like ANSYS, ESTEEM, AutoCAD, Civil 3D and such are some of the tools that have been used by civil designers to finish their desired design. But of course, certain precautions should be well taken to make sure that the structure designed is safe and sound. The finite element method is used to solve as it is generally not possible to obtain analytical mathematical solutions due to complicated geometries, loadings and material properties.

This method is also capable of analyzing structural problems such as stress analysis including truss and frame analysis and buckling in columns and frames. It is also possible to obtain displacements for typical load cases as required by design codes. There are many advantages of the finite element method as it models irregularly shaped bodies easily. It is also possible to handle general load cases without encountering difficulties and handles unlimited numbers and kinds of boundary conditions.

1.2 PROBLEM STATEMENT

There are many reasons that contribute to design failures. One of them is lack of attentions given to stability and strength when designing the structures. Besides, complicated structures also can give problems to engineers who are tasked to finish the design. The structures may require a lot of time to be analyzed and some of the engineers may find it hard to finish the design within the period.

Nowadays, with the evolution of technology, many software is available to be used to carry out the design process. One of them is ANSYS software and it is very useful as it also can be used to design steel frame design. Just fill in the required data and the result will be produced. A thorough understanding of the software must be considered because some of the engineers do not fully familiar with the software and the result produced might be inaccurate. Inaccurate analysis data then may lead to affect the frame structure as a whole.

1.3 OBJECTIVE

Objectives are very important as it can be used as a guidance when finishing the task given. It is also as to briefly explain what the task is all about. For this thesis, there are a few objectives that have been set.

1. To model the steel frame structure design for warehouse.
2. To check the behavior of steel frame structure.
3. To check whether the steel frame pass all the codechecking.
4. To analyse the steel frame structure in terms of stress, strain and deflection.

1.4 SCOPE OF STUDY

This research focuses more in designing steel structure according to Eurocode 3. The results produced will then be used in the ANSYS software and necessary checking can then be carried out. Design characteristics of the steel structure must be ensured so that the main objective of the research can be achieved.

ANSYS software is used to conduct the analysis and the methodology of the software is learned through tutorials in class and internet. Practicing the software and understanding more the software will help to solve the problems and the results produced will be able to meet the objectives.

1.5 EXPECTED OUTCOME

This research claims to find out the behaviors and passing code checking of 3D 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

The research that is being done focuses more on warehouse steel frames behavior. The analysis of warehouse steel frames behavior and passing code checking are also being carried out. All deformation, deflection, stress and strain tension checking, compression checking and lateral torsional buckling checking are according to Eurocode 3.

REFERENCES

- 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*.
- Mohamed, H.-Z., David, W., & M., A. (1983). NONLINEAR FINITE ELEMENT ANALYSIS. *J. Struct. Eng.*, 353-368.
- 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 long-span steel portal frames using fabricated beams. *Journal of Constructional Steel Research*, 104-114.
- Steel Construction*. (2014). Retrieved from Steel Construction:
http://www.steelconstruction.info/Portal_frames
- University Deakin. (2004). Retrieved from Frame Construction:
<http://www.ab.deakin.edu.au/online/vgallery/2004/srt251/team22/Home/navigation/Framing/Frame.htm#top>
- University of Alberta - ANSYS Tutorials*. (2001). Retrieved from
<http://www.mece.ualberta.ca/tutorials/ansys/>