

**FINITE ELEMENT ANALYSIS OF OFFICE
BUILDING STEEL STRUCTURE BY USING
ANSYS**

**FATHIN NUR AMMALINA BINTI
NORHASIM**

B. ENG(HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG



SUPERVISOR'S DECLARATION

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

(Supervisor's Signature)

Full Name : DR. CHENG HOCK TIAN

Position :

Date :



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 : FATHIN NUR AMMALINA BINTI NORHASIM

ID Number : AA14135

Date : 8 June 2018

FINITE ELEMENT ANALYSIS OF OFFICE BUILDING STEEL STRUCTURE BY
USING ANSYS

FATHIN NUR AMMALINA BINTI NORHASIM

Thesis submitted in fulfillment of the requirements
for the award of the
Bachelor Degree in Civil Engineering

Faculty of Civil Engineering and Earth Resources
UNIVERSITI MALAYSIA PAHANG

JUNE 2018

ACKNOWLEDGEMENTS

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 S.A.W.

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

I am grateful to thank UNIVERSITI 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 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.

ABSTRAK

Bingkai adalah salah satu elemen penting dalam struktur ini. Reka bentuk yang tidak sesuai atau tidak wajar boleh menyebabkan kegagalan sesuatu struktur. Bingkai keluli gudang dianalisis dengan pengiraan manual dan juga reka bentuk dalam perisian ANSYS. Ada beberapa kaedah yang boleh digunakan untuk menganalisis kelakuan binaan bangunan keluli. Walau bagaimanapun, di antara semua kaedah ini, kaedah unsur terhingga dan kaedah probabilistik akan menjadi kaedah yang sangat berkesan untuk mendapatkan kekuatan dan kelakuan struktur keluli. Analisis ini adalah untuk menentukan reka bentuk rangka tingkah laku bagi struktur bangunan pejabat. Penyelidikan ini akan memberi tumpuan utama kepada rangka binaan struktur bangunan pejabat. Kekuatan hasil jaket keluli yang digunakan adalah 500N/mm^2 , yang merupakan keluli terkuat. Nilai untuk pemboleh ubah masukan dijana secara rawak dengan menggunakan Simulasi Monte Carlo atau sebagai contoh yang ditetapkan menggunakan Kaedah Surface Response. 1000 Simulasi telah dibuat untuk memastikan analisis lebih tepat dan tepat. Di sini, kajian ini dijalankan untuk membuktikan bahawa gudang keluli di bawah kewujudan kekangan dan ketidakpastian boleh dianalisis dengan menggunakan analisis elemen terhingga probabilistik menggunakan ANSYS.

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 office building frame. However, among all these methods, finite element method and probabilistic method will be a very effective method to obtain the strength and the behavior of steel structure. This analysis is to determine the behavior frame design for the office building structure. This research will mainly focus on the steel frame of office building structure. The yield strength of steel jacket that used is 500N/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 were 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.

TABLE OF CONTENT

DECLARATION	
TITLE PAGE	
ACKNOWLEDGEMENTS	ii
ABSTRAK	iii
ABSTRACT	iv
TABLE OF CONTENT	v
LIST OF TABLES	ix
LIST OF FIGURES	x
LIST OF SYMBOLS	xiii
LIST OF ABBREVIATIONS	xiv
CHAPTER 1 INTRODUCTION	1
1.1 GENERAL	1
1.2 Problem Statement	1
1.3 Objectives	2
1.4 Scope of Study	2
1.5 Expected Outcome	3
1.6 Signicance of Study	3
CHAPTER 2 LITERATURE REVIEW	4
2.1 INTRODUCTION	4
2.1.1 Introduction to Structural Framing System for Office Building	4
2.2 PROBABILISTIC ANALYSIS IN FINITE ELEMENT MODELLING	5

2.3	FINITE ELEMENT ANALYSIS	6
2.4	PROBABILISTIC DESIGN	6
2.4.1	General	6
2.4.2	Probabilistic Design System	7
2.4.3	Distribution of Probability Design Variables	8
2.4.4	Monte Carlo Simulation	8
2.5	ACTIONS	10
2.5.1	Dead Load	10
2.5.2	Wind Load	10
2.5.3	Temperature and Fire	10
	CHAPTER 3 METHODOLOGY	12
3.1	INTRODUCTION	12
3.2	PREPROCESSING	14
3.2.1	Entering Level	14
3.2.2	Set Codes and Units	15
3.2.3	Defining Element Types	17
3.2.4	Defining Material	18
3.2.5	Defining Section	19
3.2.6	Defining Member Properties	20
3.2.7	Defining Beam & Shell Properties	21
3.2.8	Defining Nodes and Elements	22
3.2.9	Creating of Model	26
3.3	SOLUTION PHASE	28
3.3.1	Define Analysis Type	28
3.3.2	Apply Constraints	29

3.3.3	Apply Loads	31
3.3.4	Solving	33
3.4	POSTPROCESSING	33
3.4.1	Reaction Forces	34
3.4.2	Deflection	35
3.4.3	Read Results	37
3.4.4	Forces & Moments	38
3.4.5	Code Checking	41
3.4.6	Force and Moment List by Load Step	46
 CHAPTER 4 RESULTS AND DISCUSSION		 50
4.1	INTRODUCTION	50
4.2	PROBABILISTIC RESULTS FROM ANSYS	52
4.2.1	Random Input Variables	52
4.2.2	PDF & CDF of Input Random Variable APPLIED LOAD	52
4.3	PROBABILISTIC ANALYSIS RESULT	57
4.3.1	Statistic of the Probabilistic Result	58
4.3.2	Sample History Plots	59
4.3.3	Histogram Plots	65
4.3.4	Linear Correlation Coefficients	80
4.3.5	Spearman Rank Order Correlation Coefficients	81
 CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS		 83
5.1	INTRODUCTION	83
5.2	CONCLUSION	83
5.3	RECOMMENDATIONS	84

REFERENCES	86
APPENDIX A SAMPLE APPENDIX 1	87
APPENDIX B SAMPLE APPENDIX 2	97

LIST OF TABLES

Table 2.1	Sample table 1	Error! Bookmark not defined.
Table 2.2	Sample table 2	Error! Bookmark not defined.
Table 3.1	Specification of 3D portal frame	14
Table 3.2	Member Properties	21
Table 3.3	Keypoints with Coordinates	23
Table 3.4	Beams elements and moments	47
Table 3.5	Member Properties	49
Table 4.1	Statistical analysis of various input random variables for probabilistic	52
Table 4.2	Statistical of Random Input Variables	58
Table 4.3	Statistical of Random Output Variables	59
Table 4.4	Linear Correlation Coefficients between Input Variables	80
Table 4.5	Linear Correlation Coefficients between Input and Output Variables	80
Table 4.6	Linear Correlation Coefficients between Output Variables	80
Table 4.7	Spearman Rank Order Correlation Coefficients between Input Variables	81
Table 4.8	Spearman Rank Order Correlation Coefficients between Output Variables	81
Table 4.9	Spearman Rank Order Correlation Coefficients between Output Variables	82

LIST OF FIGURES

Figure 1.1	Steel frame structure	3
Figure 2.1	Steel frame structure	5
Figure 3.1	Summary of research process	13
Figure 3.2	Change Title	14
Figure 3.3	Activate CivilFEM	15
Figure 3.4	CIVILFEM setup options for codes	16
Figure 3.5	CIVILFEM setup options for units	17
Figure 3.6	Selecting element type	18
Figure 3.7	New material	19
Figure 3.8	Steel cross section	20
Figure 3.9	Member properties 1	21
Figure 3.10	Beam 1	22
Figure 3.11	Create Nodes in Active Coordinate System	23
Figure 3.12	Total of 168 nodes	26
Figure 3.13	Element Attributes	27
Figure 3.14	Model of 3D portal frame	28
Figure 3.15	Type of Analysis	29
Figure 3.16	Apply U,ROT on Nodes	30
Figure 3.17	Displacement constraint applied on model	30
Figure 3.18	Apply F/M on nodes	31
Figure 3.19	Apply PRES on Beams	32
Figure 3.20	Model that has been applied point loads and point load	32
Figure 3.21	Solve Current Load Step	33
Figure 3.22	Note	33
Figure 3.23	Plot Deformed Shape	34
Figure 3.24	Deformed Shape	35
Figure 3.25	Contour Nodal Solution Data	36
Figure 3.26	Contour Plot of Deflection	37
Figure 3.27	Read Results by Load Step Number	38
Figure 3.28	Graph Force and Moment Results	38
Figure 3.29	Axial Force Diagram	39
Figure 3.30	Shear Force Diagram	40
Figure 3.31	Bending Moment Diagram	41

Figure 3.32	Check Model Results By Eurocode 3	42
Figure 3.33	Graph Steel Results	42
Figure 3.34	Graph Steel Results	42
Figure 3.35	Compression Checking Results	43
Figure 3.36	Shear Resistance Checking Results	44
Figure 3.37	Bending Moment Resistance Checking Results	45
Figure 3.38	Compression Buckling Checking Results	45
Figure 3.39	Lateral Torsional Buckling Checking Results	46
Figure 3.40	List Force and Moment	47
Figure 3.41	List Stress & Strain Result	48
Figure 4.1	Model Geometry	51
Figure 4.2	PDF & CDF of Input Random Variable DENSITY	53
Figure 4.3	PDF & CDF of Input Random Variable MODULUS ELASTICITY	54
Figure 4.4	PDF & CDF of Input Random Variable APPLIED LOAD	54
Figure 4.5	PDF & CDF of Input Random Variable WINDLOAD	55
Figure 4.6	PDF & CDF of Input Random Variable TEMPERATURE	56
Figure 4.7	PDF & CDF of Input Random Variable POISSON RATIO	57
Figure 4.8	Mean Values History for Output Parameter MAXIMUMDEFLECTION	60
Figure 4.9	Standard Deviation History for Output Parameter MAXDEFLECTION	61
Figure 4.10	Mean Value History for Output Parameter MAX_DEFLECTION	62
Figure 4.11	Standard Deviation History for Output Parameter MAX_DEFLECTION	63
Figure 4.12	Sample Values for Output Parameter MAXIMUMDEFLECTION	64
Figure 4.13	Sample Values for Output Parameter MAX_DEFLECTION	65
Figure 4.14	Histogram of Input Variable APPLIEDLOAD	66
Figure 4.15	Histogram of Input Variable DENSITY	67
Figure 4.16	Histogram of Input Variable ELASTIC	68
Figure 4.17	Histogram of Input Variable POISSON	69
Figure 4.18	Histogram of Input Variable TEMP	70
Figure 4.19	Histogram of Input Variable WINDLOAD	70
Figure 4.20	Histogram of Output Parameter MAXIMUMDEFLECTION	71
Figure 4.21	Histogram of Output Parameter MAX_DEFLECTION	72
Figure 4.22	CDF of Input Variable APPLIEDLOAD	73
Figure 4.23	CDF of Input Variable DENSITY	74

Figure 4.24	CDF of Input Variable POISSON	75
Figure 4.25	CDF of Input Variable TEMPERATURE	75
Figure 4.26	CDF of Input Variable WINDLOAD	76
Figure 4.27	Linear Correlation Sensitivity Plot for MAXIMUMDEFLECTION	77
Figure 4.28	Rank- Order Correlation Sensitivity Plot for MAXIMUMDEFLECTION	78
Figure 4.29	Sensitivity Plot for MAX_DEFLECTION	79

LIST OF SYMBOLS

LIST OF ABBREVIATIONS

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

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, most of engineers now are using software to design structures. Software like ANSYS, ESTEEM, AutoCAD, Civil 3D, Revit 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 structured designed is safe a 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 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 condition.

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, lots of 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 have to 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 than may lead to affect the frame structure as a whole.

1.3 Objectives

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

1. To model the steel frame structure design for the office building.
2. To check the behavior of steel structure.
3. To check whether the steel frame pass all the code checking.
4. To analyze the steel frame structure in terms of stress, strain and deflection.

1.4 Scope of Study

This research is more in designing steel structure according to Eurocode 3. The results produces 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.



Figure 1.1 Steel frame structure

Source: <http://www.understandconstruction.com/steel-frame-structures.html>

1.5 Expected Outcome

This research claims to find out the behaviors and passing code checking of 3D steel frame of office building. 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 a result in term of step and results of analyzing office building steel frames behavior. The analyses of office building 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

- AIJ. (2004). Wind loads. *AIJ Recommendations for Loads on Buildings*, 3(1), 1–56.
- Băetu, G., Gălățanu, T. F., & Băetu, S. A. (2017). Behavior of Steel Structures under Elevated Temperature. *Procedia Engineering*, 181, 265–272.
<https://doi.org/10.1016/j.proeng.2017.02.388>
- Bhandari, N.M, Prem Krishna, Krishen Kumar, A. G. (2016). An Explanatory Handbook on Proposed IS 875 (Part 3) Wind Loads on Buildings and Structures, 1–110.
- Canonsburg, T. D. (2009). Structural Analysis Guide. *ANSYS, Inc.*, 15317(November), 724–746.
- Ming, T., Tian, C. H., Zakaria, Z. bin, & Zakaria, I. bin. (2012). Application of probabilistic analysis in finite element modeling of prestressed inverted T-beam with web openings. *Research Journal of Applied Sciences, Engineering and Technology*, 4(4), 350–366.
- Multi-storey_office_buildings @ www.steelconstruction.info. (2014). Retrieved from https://www.steelconstruction.info/Multi-storey_office_buildings
- Narasaiah, G. L. (2008). *Finite Element Analysis*.
- Niwa, N., Akiyama, S., & Nagai, S. (2004). HIGH-RISE OFFICE BUILDINGS, (53).
- Parisi, F., & Augenti, N. (2017). *Structural failure investigations through probabilistic nonlinear finite element analysis: Methodology and application. Engineering Failure Analysis* (Vol. 80). Elsevier Ltd.
<https://doi.org/10.1016/j.engfailanal.2017.07.004>
- Quaranta, G. (2011). Finite element analysis with uncertain probabilities. *Computer Methods in Applied Mechanics and Engineering*, 200(1–4), 114–129.
<https://doi.org/10.1016/j.cma.2010.07.018>
- Rao, S. S., & Rao, S. S. (2018). Finite Element Analysis Using ANSYS * *ANSYS FEM software is marketed by ANSYS, Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317. *The Finite Element Method in Engineering*, 703–721.
<https://doi.org/10.1016/B978-0-12-811768-2.00022-5>
- Santra, S. B. (2009). Monte Carlo Simulation technique.
- SCI. (2015). *Steel Building Design: Design Data*.
- Solari, G. (2017). Wind Loading of Structures: Framework, Phenomena, Tools and Codification. *Structures*, 12, 265–285. <https://doi.org/10.1016/j.istruc.2017.09.008>
- Steel Structures 1. (2013), 1–39.
- Watts, G. a. (2011). Structural loads., 2, 1–8.
<https://doi.org/10.1061/9780784412251.ch04>