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

LIST OF TABLES

LIST OF FIGURES

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 /MAX_DEFLECTION	Maximum 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 Signicance 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., Gălăţanu, 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