FINITE ANALYSIS OF STEEL FRAME MULTI-STOREY BUILDING BY USING ANSYS

LIM YEE SIANG

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 : LIM YEE SIANG

ID Number : AA15241

Date : 30 May 2019

FINITE ANALYSIS OF STEEL FRAME
MULTI-STOREY BUILDING
BY USING ANSYS

LIM YEE SIANG

Thesis submitted in fulfillment of the requirements
for the award of the
B.Eng (Hons) in 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 multi-storey building 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. 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 bangunan bertingkat telah dianalisis menggunakan perisian ANSYS. Kajian ini adalah untuk menentukan ketegangan, tekanan, pesongan maksimum, ubah bentuk dan juga menyemak struktur mengikut Eurocode 3. Major bangunan yang dibina di Malaysia adalah bangunan konkrit bertetulang berbanding bangunan 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

LIST OF TABLES

LIST OF FIGURES

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
Ø	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

In modern days now, new technologies have been found to help engineers to conduct projects. These technologies ease the work of the engineers and also speed up the progress of the project to ensure the project is done in limited time.

Major of the buildings constructed in Malaysia are reinforced concrete buildings compared to steel frame buildings. Steel structure design software is very important to help civil engineers in analyzing of the steel structure. Thus, the software of ANSYS CivilFEM software has invented to bring an improvement for faster analysis and design of the steel arch structure. It is very unpractical and not effective to do manual calculation in every structure of the building in industry. Hence, it is very important to use the ANSYS software to analyze and solve the practical problem.

In this project, the software used, ANSYS and Civil FEM is one of the well-known and advanced finite element analysis and design software package available for the structural engineering projects. This software combines the state-of-the art general purpose structural analysis features of ANSYS (ISO-9001) with high-end civil engineering-specific structural analysis capabilities of Civil FEM, which make it usable in very wide range of civil engineering projects. Since both ANSYS and Civil FEM are completely integrated, Civil FEM supports all types of advanced analysis supported by ANSYS running as a unique software and executable. ANSYS and Civil FEM are going to be technological leader which is capable to analyze the construction/civil engineering design and help in other technological process in the civil engineering projects.

1.2 Problem Statement

Analyzing of a 3D steel frame structure building is a very complicated process. This is a very time consuming and difficult to analyze the structure behavior. Delay of time in analyzing the structure might cause a delay in the construction work. This delay of work will cause a huge risk to the contractor especially for those very time-limited construction project. Besides, it is not practical to analyze the 3D steel frame structure. Software are invented to solve this problem, for example ANSYS is one of them which is a very effective software to do complicated analyzing of steel frame structure. By using ANSYS, it helps to reduce the time consumed in designing phase of construction.

In this project, ANSYS + CIVILFEM 12.0 program is used for modelling and simulation of the characteristic behavior of the steel frame in this research. CivilFEM, is a civil engineering special software that comes in package with ANSYS, is taking base on the structure of civil engineering for a variety of simulation of design and checking (Moreno, Monteagudo, Maia, & Ingeciber, 2001).

1.3 Objective

This study is to analyze the influence line of bending moment. Hence, there are the following objectives to be achievable

i. To check the stability of the structure by code checking process.
ii. To determine the behaviors of portal frame under surface load and wind load.

1.4 Scope of Study

In this project, ANSYS + CIVILFEM 12.0 will be used to carry out 3D portal frame analyzation and modelling. Loading acting on every beam will be calculated based on Eurocode 3, by calculating the loading of slabs and brick walls, and will be applied as pressure on beam in CivilFEM. Besides, wind load will be calculated based on Malaysia Standard, while the wind load will be only applied on just one side of the structure. Lastly, deflection, deformed shape and Eurocode 3 checking will be done in postprocessing step.

1.5 Expected Outcome

From this project, it is expected to determine the structural behaviors of the 3D portal frame of the high rise building. Code checking of the building structure should be passed. The structural behaviors include deformation, deflection, tension checking, compression checking and lateral torsional buckling checking based on the standard of Eurocode 3.

1.6 Significance of Study

This project is very important in finding out the results of analyzing the 3D portal frame by using ANSYS software. From this project, maximum deformation, deflection and moments under different load can be determined. Analyzing of 3D portal frame is very important before construction because it can avoid any failure of the structure. From this project, it is easy to determine the best dimension to be chosen for the portal frame . This can help the contractors to calculate the bill of quantity and cost of the project before construction. Not only save time, cost can also be controlled to prevent over budget.

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