FINITE ELEMENT ANALYSIS OF STEEL FRAME FOR HIGH RISE BUILDING STRUCTURE USING ANSYS

LAM CHENG YONG

B. ENG(HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

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 : LAM CHENG YONG

ID Number : AA15275

Date : 31 MAY 2019

FINITE ELEMENT ANALYSIS OF STEEL FRAME FOR HIGH RISE BUILDING STRUCTURE USING ANSYS

LAM CHENG YONG

Report submitted in partial fulfillment of the requirements
for the award of the
B. Eng (Hon.) Civil Engineering

Faculty of Civil Engineering & Earth Resources

UNIVERSITI MALAYSIA PAHANG

MAY 2019

ACKNOWLEDGEMEN

Firstly, I would like to express my gratefulness to the God Almighty for the good health and well-being throughout the entire study.

I would like to address my greatest to my supervisor, Dr. Cheng Hock Tian for the greatest effort in guiding, giving advices, supports and encouraged me through the entire journey of my studies for the final year project.

Next, I wish to thank Dr. Nurul Nadrah Aqilah binti Tukimat for providing me with necessary knowledge in thesis writing. Throughout this final year project, I wish to palce my sincere appreciations to all the lecturers for giving encouragement, support, and attention directly and indirectly in Universiti Malaysia Pahang.

I would also like to take this opportunity to thank my family and friends who encouraged me with their best wishes and support.

Last but not least, I would like to thank all who involved directly and indirectly in ensuring the smoothness of this study. Thank you very much.

ABSTRAK

Dalam kajian ini, bingkai keluli 3D dianalisis dengan pengiraan manual dan juga program ANSYS dengan kaedah probabilistik. Analisis ini adalah untuk menentukan tekanan, ketegangan, pesongan, ubah bentuk struktur dan diperiksa dengan sewajarnya kepada Eurocode 3 juga. Baru-baru ini, bingkai keluli biasanya digunakan dalam bangunan kerana cahaya, ketahanan yang lebih tinggi, didirikan dengan pantas dan ekonomik. A 3D Lin. Rasuk 188 jenis elemen keluli dipilih untuk bingkai keluli 3D. Oleh kerana kriteria reka bentuk dalam reka bentuk untuk tegangan, mampatan dan tenggelam telah diluluskan, maka bahan dan geometri keluli telah dipenuhi juga untuk kajian ini. Keputusan berangka menunjukkan persetujuan yang sangat baik dengan hasil perhitungan manual. Tingkah laku sebenar struktur di bawah beban diterapkan dari hasil simulasi. Hasil fungsi pengedaran kumulatif, plot histogram, plot kepekaan dan plot sejarah mudah oleh 10000 kali simulasi untuk sebarang input dan output dalam analisis probabilistik.

ABSTRACT

In this research, 3D steel frame was analyzed by manual calculation and also ANSYS program with the probabilistic method. This analysis is to determine the stress, strain, deflection, deformation of the structure and checked accordingly to Eurocode 3 as well. Recently, steel frame is normally used in building due to light, greater durability, erected rapidly and economical. A 3D Lin. Beam 188 of steel element type was choose for the 3D steel frame. Due to the design criteria in designing for tensile, compression and buckling were passed, hence the material and geometry of the steel were satisfied as well for this research. The numerical results show very good agreement with manual calculation results. The real behavior of the structure under the applied loads from the results of simulation. The results of cumulative distribution function, histogram plot, sensitivity plot and simple history plot by 10000 times of simulation for any input and output in probabilistic analysis.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

tw Web thickness

tf Flange thickness

hw Height of web

i Height

b Width

d Depth

A Area of section

I Moment of inertia

W_{pl} Plastic modulus

i Radius of gyration

N Axial load

V Shear force

M Moment

α Imperfection factor

$\gamma M0$ 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

[K] Global-coordinate structure stiffness matrix

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 this modern era, we can found that many of the famous building in this world used the high technology especially in civil engineering. For the construction we can see the new technologies in the term of the design, installation of the member and how the project conducted such as IBS system and software that used to analysis. Steel frame can be defined as the building technique with a skeleton frame of the steel columns and horizontal steel beam. The steel frame in the construction are used to support the main element such as roof, floors and walls that attached to the frame. The steel frame can be formed into many types of shape such as square, tabular and I section. According to (john wiley and sons 1984), as a designer they have to select the steel sections from a discrete that contains certain designations of steel profiles. Generally the steel structures are designed to be pinned or fully rigid connections but most of the connections in steel frames are semi rigid in reality. The research for the type of these connections as well as on the numerical techniques of completing an analysis of flexible connected steel frames was developed (A.J numer,1988).

Nowadays, there are many building constructions are failed in the term of steel frames failure and it's brings a lot of concern to the structural engineers and designers. Typically, the steel frame structures are designed and constructed based on the term of the steel frame's strength and rigidity to satisfy the loads and serviceability limitation. The steel frames as we know mostly are ready-made. So, the discrete optimization technique should be proposed (S.A May, R.J Bailing, 1992). As the result, the engineers must initiate their design of these frames by propose the beams, columns and girders to satisfy the total sideways. According to (D.E Grierson ,1994), to ensure the steel frame

is safe to used, approximate optimization techniques are useful for obtaining well balanced which must be checked for strength using the standards.

ANSYS CivilFEM is a most advanced comprehensive, reputable finite element analysis software and it is the design software package available for the structural engineering project. CivilFEM has the advantages which is it can support all types of advanced analysis supported by ANSYS. Finite element analysis is a numerical method of deconstructing a complex system in every part of member that called as element. The analysis that produced was linearly elastic because contact was embodied artificially by attaching and release nodes at each load step. Then, the correlation between 2-Dimensional and 3-Dimensional finite element was established with 2D models (A.R kukreti, T.M Murray, A.Abolmaali ,1987). The ANSYS software implements equations that govern the behavior of the elements and solves them all and create a comprehensive explanation of the system. The result from the analysis was useful in the range for which such validations were performed and finite elements models was developed for stiffened steel tee hanger connections (T.M Murray , A.R Kukreti, M.Ghassemieh 1989).

1.2 PROBLEM STATEMENT

The structure failure can be classified into various type of factors. Normally, the structure will collapse due to their design failure of the structural components. Typically, the construction projects are more focusing on the artistically design of the building compared to the stability and strength of the structures. Most of the developers just want their projects look artistic and ignore the important messages or data from the analysis. In this cases, the design standards will be unable to fulfill the requirement of the building itself. Besides that, the more complicated the structure, the more of time needed to analyze. As an engineer which responsible to the construction it is necessary to calculate, design and make sure the steel frames are safe to use in the construction.

By using the ANSYS software, the designer just have to fill in the data which are required by the software such as the building’s length, area and material to produce the results. In minor cases, the results produced are inaccurate because the designer especially the fresh graduated are not familiar with the software and lack of the

knowledge on the software. This could be a disaster due to the inaccurate analysis data may affect the frame structure stability.

1.3 OBJECTIVES

Objectives can be defined as the goal or mission that we need to achieve when conducting a project or study. Objectives act as the guidance to make sure we can successfully complete our project or study. In my thesis there are a few objectives that related in term of structure and analysis. The objectives are:

1. To model the steel frame structure design for high rise.
2. To determine the behaviour of steel frame structure.
3. To check the steel framed pass all the code checking.
4. To analyse the steel frame structure in term of stress, strain and deflection.

1.4 SCOPE OF STUDY

The research are more focused on the design of steel frame structures by using standard Eurocode 3 that based on the steel design. The data that produced will be used to generate the new equation by proving the parameters in the ANSYS software. Ability to understand the types of the steel frame design and the characteristic of the structure is very important to achieve the objectives of the research.

Exploring the functions and the steps of operating the ANSYS software by practicing the tutorial from the class and internet. Practicing the tutorials and understanding the function can helps to solve problem when the software running the real model that proposed. The results from the analysis can produced the data for the steel frame structures that mention in the objectives.

1.5 EXPECTED OUTCOME

The research is more to put emphasis on the steel frame structure behavior. It claims to analyze the behaviors and passing code checking. The behavior that interested for this research is deflection, deformation, compression checking, stress and strain

REFERENCES

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

BS EN 1993-3-8:. (2005). *Eurode 3: Design of Steel Structures. Design of joints, BSI.*

Caprani, C. (2009/2010). Plastic Analysis Structural Engineering.

Davidson, B., & W.Owens, G. (2012). *Steel Designers' Manual.* BLACKWELL PUBLISHING LTD.

Deakin, U. (2009). Retrieved from Frame Concstruction: http://www.ab.deakin.edu.au/online/vgallery/2004/srt251/team22/Home/navigation/Framing/Frame.htm#top

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.

Goswami, S., Chakraborty, S., & Ghosh, S. (2013). Adaptive Response Surface Method in Structural Approximation Under Uncertainty.

Jing Ji, W. Z. (2012). Research of Seismic Testing and Dynamic Character of High-rise Building Structure Based on ANSYS.

Lenka Lausovaa, I. K. (2017). Numerical Analysis of Steel Frame Exposed to Fire. *Procedia Engineering*.

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

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., & Sha, W. (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: https://www.steelconstruction.info/Continuous_frames

Tvrda, K. (2017). RSM Method in Probabilistic Analysis of the Foundation Plate.

University of Alberta - ANSYS Tutorials. (2010). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys/