

**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

DECLARATION	
TITLE PAGE	
ACKNOWLEDGEMEN	ii
ABSTRAK	iii
ABSTRACT	iv
TABLE OF CONTENT	v
LIST OF TABLES	viii
LIST OF FIGURES	ix
LIST OF SYMBOLS	xii
LIST OF ABBREVIATIONS	xiii
CHAPTER 1 INTRODUCTION	1
1.1 GENERAL	1
1.2 PROBLEM STATEMENT	2
1.3 OBJECTIVES	3
1.4 SCOPE OF STUDY	3
1.5 EXPECTED OUTCOME	3
1.6 SIGNIFICANCE OF STUDY	4
CHAPTER 2 LITERATURE REVIEW	5
2.1 INTRODUCTION	5
2.2 LOADS ON STRUCTURE	6
2.2.1 Static Force	6

2.2.2	Wind Loads	7
2.3	FRAME	7
2.3.1	Steel Frame	7
2.4	FINITE ELEMENTS ANALYSIS	8
2.4.1	Finite Element Analysis Function	8
2.4.2	Advantage of Parametric Design	9
CHAPTER 3 METHODOLOGY		10
3.1	Introduction	10
3.2	Preprocessing	11
3.2.1	Entering Title	12
3.2.2	Set Codes and Units	12
3.2.3	Defining Element Types	14
3.2.4	Defining Material	14
3.2.5	Defining Section	15
3.2.6	Defining Member Properties	16
3.2.7	Defining Beam and Shell Properties	17
3.2.8	Defining Nodes and Elements	18
3.2.9	Creating of Model	20
3.3	Solution Phase	22
3.3.1	Define Analysis Type	22
3.3.2	Apply Constraints	22
3.3.3	Apply loads	23
3.3.4	Solving	25
3.4	Postprocessing	26
3.4.1	Reaction Forces	26

3.4.2	Deflection	27
3.4.3	Read Results	29
3.4.4	Forces & Moments	29
3.4.5	Code Checking	31
CHAPTER 4 RESULTS AND DATA ANALYSIS		38
4.1	Introduction	38
4.2	Probabilistic Results from ANSYS	38
4.2.1	Random Input Variables	38
4.2.2	Statistics of The Probabilistic Results	43
4.2.3	Sample History Plots	44
4.2.4	Histogram Plots of Output Parameters	52
4.2.5	Histogram Plots of Input Parameters	54
4.2.6	Cumulative Distribution Function Plots	59
4.2.7	Sensitivity Plots	65
4.2.8	Linear Correlation Coefficients	66
4.2.9	Spearman Rank Order Correlation Coefficients	67
CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS		69
5.1	Introduction	69
5.2	Conclusion	69
5.3	Recommendation	70
REFERENCES		70

LIST OF TABLES

Table 3.1	Member Properties	16
Table 3.2	Beam & Shell Properties	18
Table 4.1	Random Input Variables Specifications	39
Table 4.2	Statistical of Random Input Variables	44
Table 4.3	Statistical of Random Output Variables	44
Table 4.4	Linear Correlation Coefficients between Input Variables	67
Table 4.5	Linear Correlation Coefficients between Input and Output Variables	67
Table 4.6	Linear Correlation Coefficients between Output Variables	67
Table 4.7	Spearman Rank Order Correlation Coefficients between Input Variables	68
Table 4.8	Spearman Rank Order Correlation Coefficients between Input and Output Variables	68
Table 4.9	Spearman Rank Order Correlation Coefficients between Output Variables	68

LIST OF FIGURES

Figure 2.1	show the typical types of loading condition standards that must be follow when design process.	6
Figure 3.1	Summarize research methodologies for the study	11
Figure 3.2	Activate CIVILFEM	12
Figure 3.3	CIVILFEM setup options for units	13
Figure 3.4	CIVILFEM setup options for codes	13
Figure 3.5	Selecting element type	14
Figure 3.6	New Material	15
Figure 3.7	Steel cross section	16
Figure 3.8	Member Properties	17
Figure 3.9	Beam	17
Figure 3.10	Create Nodes in Active Coordinate System	18
Figure 3.11	Keypoints with Coordinates	19
Figure 3.12	Total of 48 Keypoints	20
Figure 3.13	Element Attributes	21
Figure 3.14	Model of 3D steel frame	21
Figure 3.15	Type of Analysis	22
Figure 3.16	Apply U.ROT on nodes	23
Figure 3.17	Displacement constraint applied on model	23
Figure 3.18	Apply Force/Moment on Nodes	24
Figure 3.19	Apply PRES on Beams	24
Figure 3.20	Model that applied by point and wind loads	25
Figure 3.21	Solve Current Load Step	25
Figure 3.22	Note	26
Figure 3.23	Plot Deformed Shape	26
Figure 3.24	Deformed Shape of the Steel Frame	27
Figure 3.25	Contour Nodal Solution Data	28
Figure 3.26	Contour Plot of Deflection	28
Figure 3.27	Read Results by Load Step Number	29
Figure 3.28	Graph Force and Moment Results	29
Figure 3.29	Axial Force Diagram	30
Figure 3.30	Shear Force Diagram	30
Figure 3.31	Bending Moment Diagram	31

Figure 3.32	Check Model Results By Eurocode 3	32
Figure 3.33	Graph Steel Results	32
Figure 3.34	Tension Checking Results	32
Figure 3.35	Compression Checking Results	33
Figure 3.36	Shear Resistance Checking Results	33
Figure 3.37	Bending Moment Resistance Checking Results	34
Figure 3.38	Compression Buckling Checking Results	34
Figure 3.39	Lateral Torsional Buckling Checking Results	35
Figure 3.40	Bending + Axial Force Checking Results	35
Figure 3.41	Bending + Shear Force Checking Results	36
Figure 3.42	Bending + Axial Force + Shear Force Checking Results	37
Figure 4.1	PDF & CDF of Input Random Variable DENS	40
Figure 4.2	PDF & CDF of Input Random Variable ELASTIC	40
Figure 4.3	PDF & CDF of Input Random Variable POISON	41
Figure 4.4	PDF & CDF of Input Random Variable LOAD	42
Figure 4.5	PDF & CDF of Input Random Variable WINDLOAD	42
Figure 4.6	PDF & CDF of Input Random Variable TEMP	43
Figure 4.7	Sampled Values for Output Parameter MAXIMUMDEFLECTION	45
Figure 4.8	Mean Value History for Output Parameter MAXIMUMDEFLECTION	45
Figure 4.9	Standard Deviation History for Output Parameter MAXIMUMDEFLECTION	46
Figure 4.10	Minimum Value for Output Parameter MAXIMUMDEFLECTION	47
Figure 4.11	Maximum Value for Output Parameter of MAXIMUMDEFLECTION	47
Figure 4.12	Sample Values for Output Parameter MAX_DEFLECTION	48
Figure 4.13	Mean Value History for Output Parameter MAX_DEFLECTION	48
Figure 4.14	Standard Deviation History for Output Parameter MAX_DEFLECTION	49
Figure 4.15	Minimum Value for Output Parameter MAX_DEFLECTION	49
Figure 4.16	Maximum Value for Output Parameter of MAX_DEFLECTION	50
Figure 4.17	Simulation Sample Values for Output Parameter MAXIMUMDEFLECTION	51
Figure 4.18	Simulation Sample Values for Output Parameter MAX_DEFLECTION	51
Figure 4.19	Histogram of Output Parameter for MAXIMUMDEFLECTION	52
Figure 4.20	Histogram of Output Parameter for MAX_DEFLECTION	53

Figure 4.21	Histogram of Input Parameter for DENS	54
Figure 4.22	Histogram of Input Parameter for ELASTIC	55
Figure 4.23	Histogram of Input Parameter for POISON	56
Figure 4.24	Histogram of Input Parameter for LOAD	57
Figure 4.25	Histogram of Input Parameter for WINDLOAD	58
Figure 4.26	Histogram of Input Parameter for TEMP	59
Figure 4.27	CDF of Input Variable DENS	60
Figure 4.28	CDF of Input Variable ELASTIC	60
Figure 4.29	CDF of Input Variable POISON	61
Figure 4.30	CDF of Input Variable LOAD	62
Figure 4.31	CDF of Input Variable WINDLOAD	62
Figure 4.32	CDF of Input Variable TEMP	63
Figure 4.33	CDF of Output Variable MAXIMUMDEFLECTION	64
Figure 4.34	CDF of Output Variable MAX_DEFLECTION	64
Figure 4.35	Sensitivity Plot for MAXIMUMDEFLECTION	65
Figure 4.36	Sensitivity Plot for MAX_DEFLECTION	66

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
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
\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 find that many of the famous buildings in this world used high technology especially in civil engineering. For construction, we can see new technologies in terms of design, installation of members and how the project is conducted such as IBS systems and software used for analysis. Steel frames can be defined as a building technique with a skeleton frame of steel columns and horizontal steel beams. Steel frames in construction are used to support main elements such as roofs, floors, and walls attached to the frame. Steel frames can be formed into many shapes such as square, tubular, and I-section. According to (John Wiley and Sons, 1984), as a designer, they have to select steel sections from a discrete set that contains certain designations of steel profiles. Generally, steel structures are designed to be pinned or fully rigid connections, but most connections in steel frames are semi-rigid in reality. Research on the type of these connections as well as on the numerical techniques for completing an analysis of flexible connected steel frames was developed (A.J. Numer, 1988).

Nowadays, many building constructions have failed in terms of steel frame failure, which brings a lot of concern to structural engineers and designers. Typically, steel frame structures are designed and constructed based on the strength and rigidity of the steel frame to satisfy load and serviceability limitations. Steel frames as we know are mostly ready-made. So, discrete optimization techniques should be proposed (S.A. May, R.J. Bailing, 1992). As a result, engineers must initiate their design of these frames by proposing 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). *Eurocode 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 Construction:
<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/>