FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE FRAME BY USING ANSYS

PHANG SZE QIAN

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 : DR. CHENG HOCK TIAN

Position : Undergraduate Research Project Supervisor

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 : PHANG SZE QIAN

ID Number : AA15127

Date : 30 May 2019

FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE FRAME BY USING ANSYS

PHANG SZE QIAN

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

ACKNOWLEDGEMENTS

With boundless love and appreciation, I would like to extend my heartfelt gratitude and appreciation to the people who helped me bring this study into reality. I would like to extend my profound gratitude to the following:

My advisor, Dr. Cheng Hock Tian whose expertise, consistent guidance, ample time spent and consistent advices that helped me bring this study into success.

To the panel team for their constructive comments, suggestions and critiquing.

Besides, I would like to express my utmost gratitude to all those persons who have given their valuable support and assistance.

Last but not least, I would like to acknowledge with gratitude, the support and love of my family. It was my parents' unconditional love, care and tolerance which made the hardship of writing the thesis worthwhile.

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 memeriksa struktur mengikut Eurocode 2. Major bangunan yang dibina di Malaysia adalah bangunan konkrit bertetulang berbanding bangunan bingkai keluli. Perisian reka bentuk struktur konkrit bertetulang 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 konkrit bertetulang 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.

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 2. Major of the buildings constructed in Malaysia are reinforced concrete buildings compared to steel frame buildings. Reinforced concrete 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 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 reinforced concrete 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.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

C25/35	Concrete Grade
KN	Kilonewton

LIST OF ABBREVIATIONS

CS	Coordinate System
FEA	Finite Element Analysis
PDF	Probability Density Function
RCC	Reinforced Cement Concrete
RSM	Response Surface Method
2D	2 Dimensional
3D	3 Dimensional

CHAPTER 1

INTRODUCTION

1.1 Introduction

The engineers use different tools and techniques in designing the structure to ensure their safety and stability. However, unexpected foreseen problems always occur. It might be a designing error, construction error or a failure of the structural components. Finite element analysis is a numerical method to provide solution to the problems that would difficult to obtain. This numerical analysis is performed by ANSYS CivilFEM 12.0 software. ANSYS is a powerful software that can perform linear and non-linear analysis on materials behaviours, joint and geometry.

1.2 Problem statement

Reinforced concrete frame building is a most common and modern structure nowadays. Its simplicity of construction that consist of a frame or skeleton of concrete shorten the construction period. To improve and to avoid any unexpected construction failure of the structure, Finite Element Analysis should be introduced in analysing the structure.

By introducing Finite Element Analysis(FEA), the reinforced concrete frame building can be analysed in a more efficient and careful way that are more reality. FEA method can analysis the overall structure with considering different type of failures before being constructed in real world. This study is an attempt to produce an analysis that is more constructability by understanding the concrete nonlinear stress-strain relationship and crack profile of concrete.

1.3 Objectives

To gain familiarity with phenomena and achieve new insight into it, objectives are important for every thesis papers. It helps us to discover new facts and truth. Objectives also help the researchers to study and resolve contradiction in the area of a study.

i. To determine the reinforced concrete frame passes all code checking.

ii. To determine the behaviours of portal frame under surface load and wind load.

1.4 Scope of study

In this research, ANSYS + CivilFEM 12.0 will be used to analysis the reinforced concrete frame building. Eurocode 1 will be used to calculate every applied loading, which include the self-weight of beams, slabs and columns. Deflection and displacement of structural components will be checked by Eurocode 3 in post processing steps.

1.5 Expected outcome

This research aims to find out the structural behaviour of the elements by applied loads on the structure. The stability of the structure will be checked based on Eurocode 3. The materials behaviours of concrete such as deflection and deformation diagrams will be obtained to understand the stress distribution diagram of the structural components. To obtain an illustration diagram of tension crack that may occur in concrete.

1.6 Significance of study

This research is significant for analysing the reinforced concrete frame building by considering the applied loads on the construction. The crack profiles obtained are significantly useful on the identification of the failure modes of the RCC frame. This research can help us to understand more about the actual behaviours of structural members during construction stages. It is important to learn different methods to do analysis on structural buildings. It will definitely improve the safety and stability of the structure.

CHAPTER 2

LITERATURE REVIEW

2.1 Reinforced cement concrete(RCC) frame

2.1.1 Introduction of reinforced cement concrete frame

A reinforced cement concrete frame is an assembly of beams, slabs and columns. They are interconnected with each other to form a unit. The load is transferred from a slab to the beams then to the columns and to the foundation which turns into the soil. The brick walls are constructed after the whole RCC frame finished constructed and unable to carry any loads. The function of brick walls is used to separate the spaces on every floor to create rooms. RCC frame structures can be seen in many urban areas because of its simple and faster construction time.

2.2 Actions

2.2.1 Permanent actions

Permanent loads or dead loads refer to loads that will not change relative to time. They will transfer to the whole structure throughout its lifespan. Example of dead loads such as self-weight of beams, columns, slabs, wall and finishing flooring materials.

2.2.2 Variable actions

Variable actions or live loads is the load that can change over time. Live load can be movable or moving loads, temporary and transferable loads. The examples of live loads are the occupancy of the building such as the furniture, stored materials and humans.

REFERENCES

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

Dundar Cengiz, Tokgoz Serkan, Tanrikulu A. Kamil, Baran Tarik. (2008).. Behavior of reinforced and concrete-encased composite columns subjected to biaxial bending and axial load",Building and Environment, 43,(6),1109-1120

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.

Mohamed, H.-Z., David, W., & M., A. (1983). NONLINEAR FINITE ELEMENT ANALYSIS. J. Struct. Eng., 353-368.

Wight.J.K. and MacGregor.J.G. (2009)."Reinforced Concrete Mechanics & Design". Fifth Edition, New Jersey, USA: Pearson Education, Inc..

Fu, F. (2009). Progressive collapse analysis of high-rise building with 3-D finite element modeling method..

Goswami, S., Ghosh, S., & Chakraborty, S. (2016). Reliability analysis of structures by iterative improved response surface method.

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

University of Alberta - ANSYS Tutorials. (2001). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys/

ANSYS. (January, 2018). 4.188 BEAM188 3-D Linear Finite Strain Beam.

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

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 Sevill

University of Alberta - ANSYS Tutorials. (2001). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys

SHARCNet. (January, 2018). BEAM 188.

Carley, K. M., Kamneva, N. Y., & Reminga, J. (2004). Response surface methodology. CASOS Technical Report