FINITE ELEMENT ANALYSIS OF SLAB BRIDGE WITH BEAM ELEMENT USING ANSYS

HO KIT SHENG

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 : HO KIT SHENG

ID Number : AA15239

Date : 31st May 2019

FINITE ELEMENT ANALYSIS OF SLAB BRIDGE WITH BEAM ELEMENT USING ANSYS

HO KIT SHENG

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

Faculty of Civil Engineering & Earth Resources

UNIVERSITI MALAYSIA PAHANG

MAY 2019

ACKNOWLEDGEMENTS

First and foremost I offer my sincerest gratitude to my supervisor, Dr Cheng Hock Tian, who has supported me throughout my thesis with his patience and knowledge whilst allowing me the room to work in my own way. I attribute the level of my degree to his encouragement and effort and without him this thesis, too, would not have been completed or written. One simply could not wish for a better or friendlier supervisor. He always teaches me very patiently throughout the process of probabilistic analysis and gives me good suggestion when I have problem. Without advice from him, I think I would be unable to complete this project report.

Moreover, I would also like to thanks to the project panel team for the recommendations and comment about this research. With the comment and suggestion, it has improved the outcome of this project. Rather than that, I'm also want to thanks a lot to research team that had gave a lot of ideas to improve my shortage. It is a pleasure to thank to all my fellow undergraduate friends, who are under supervision of Dr Cheng Hock Tian. The moment that we share our knowledge and helping each other to solve the problem is meaningful and will always keep inside my heart.

Besides that, it is an honor for me to take this opportunity to thank the authority of University Malaysia Pahang (UMP) for providing me a good environment and facilities. Besides, thanks to all lectures of Civil Engineering & Earth Resources faculty. Thank you very much.

Finally, again I would like to express my thanks to all who directly and indirectly involved in completing this research.

ABSTRAK

Dalam kajian ini, model jambatan rasuk telah dianalisis dengan menggunakan ANSYS. Kajian ini adalah untuk mengkaji kecukupan pengukuhan dan juga mengkaji tingkah laku struktur mengikut Eurocode 2. Struktur ini dianalisa menggunakan analisis unsur terhingga menggunakan ANSYS untuk memeriksa kecukupan pengukuhan. Dari hasil simulasi, kita dapat mengetahui tingkah laku struktur di bawah parameter masuakan yang digunakan. Di samping itu, dengan menjalankan analisis probabilistik, kami mengumpul hasil fungsi agihan kumulatif, graf histogram, graf kepekaan dan graf sejarah untuk semua parameter. Graf tersebut menunjukkan maklumat seperti nilai min, sisihan piawai, nilai minima, nilai maksimum, kecondongan dan kurtosis parameter. Nilai bagi parameter keluaran dihasilkan dengan simulasi Monte Carlo secara rawak. Parameter keluaran dihasilkan berasaskan parameter masukan yang dihasilkan melalui analisis probabilistik dengan sistem reka bentuk probabilistik ANSYS. Nilai min, sisihan piawai, nilai minimum nilai sampel, nilai maksimum, kecondongan dan kurtosis parameter keluaran ditunjukkan melalui graf sejarah sampel. Sampel 5000 simulasi analisis probabilistik dilakukan, untuk memastikan hasil analisisnya tepat dan seragam. Panjang model ialah 20m. Pada akhirnya, beberapa cadangan diberikan untuk memperbaiki perisian ANSYS untuk memastikan pengalaman pengguna yang terbaik.

ABSTRACT

In this research, a beam bridge model has been analysed by using ANSYS. This research is to investigate the adequacy of reinforcement and also checking the structural behaviour of the structure according to Eurocode 2. The structure is analyse using finite element analysis using ANSYS to check the adequacy of the reinforcement. From the result of simulation, we can know the behaviour of the structure under the input parameter that applied. Beside these, by carrying out the probabilistic analysis, we collected the result of cumulative distribution function, the histogram plot, sensitivity plot and simple history plot for all parameter. These plots shows the information such as mean value, standard deviation, minimum value, maximum value, skewness and kurtosis of the parameters. The values for the output parameter were generated with Monte Carlo simulation randomly. The output parameter is generated based the input parameter generated through probabilistic analysis with ANSYS probabilistic design system. The mean value, standard deviation, sample values minimum value, maximum value, skewness and kurtosis of the output parameter is presented through sample history plot. 5000 loops of probabilistic analysis simulation is done, to ensure that the result of analysis is precise and uniform. The length of the model is 20m. At last, some suggestions is given to improve the ANSYS software to make sure the best user experience.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

αcc	Coefficient of long term effects on compressive strength
β	Centroid ratio
r	Radius
h	Height
b	Width
d	Effective of section
β	Centroid ratio
fyk	Characteristic yield stress of the reinforcement
fav	Average stress
As	Area of steel reinforcement
Fs	Force of steel
M_{Rd}	Moment equilibrium
γ_s	Partial safety factor for steel
γ_c	Partial safety factor for concrete
∅	Diameter
π	Pi
x	Distance
Es	Modulus of Elasticity
εcu2	Ultimate concrete compressive strains
$>$	Greater than
$<$	Lesser than

LIST OF ABBREVIATIONS

BBH	Distance between holes
BBOT	Bottom width
BCNL	Bridge and Civil Non Linearities
BM	Middle width
BTOP	Top width
BS	British Standard
CDF	Cumulative distribution function
CivilFEM	Civil Finite Element Method
DOF	Degree of freedom
EC2	Eurocode 2
E view	Elevation view
FEM	Finite element method
GAUS	Gaussian
LS	Load step
MP	Mileage point
MZ	Moment from Z direction
NH	Number of holes
Num	Number
PDS	Probabilistic Design System
PDF	Probablistic Density Function
P View	Plan view
REINFACT	Reinforcement Factor
SS	Substep
TBOT	Bottom thickness
TM	Middle Thickness
TTOP	Top Thickness

CHAPTER 1

INTRODUCTION

1.1 General

Finite element method is a method of solving a complicated problem by replacing a complicated problems with relatively simpler problems. The study or analysis of phenomenon with finite element method is often called as finite element analysis. This method is to generate approximate values of the unknown at discrete number of points over a body. This is done with a process called discretization which model a body by separating it into an equivalent system of smaller part interconnected at nodes. Instead of solving the body at once, FEM solves it by formulating equation for each part then them to get solution of whole. (Daryl L Logan, 2011).

Bridge are designed in many types such as beam bridge, truss bridge, cantilever bridge, arch bridge etc. Meanwhile, beam bridge is considered the simplest and oldest type bridge as it is just bridge made of a horizontal beam supported by vertical piers.

Despite its simplicity in design, length of beam bridge rarely exceed 76m in a single span as the flexural stresses is proportional to the square of length with deflection is proportional to length to power of 4. In other words, the bridge can be failed on its own weight if it's too long. In this research, we will take voided concrete slab bridge as the example of slab bridge with beam element.

1.2 Problem Statement

In the case of this, the bridge structure have to be well designed to ensure that it can support the stress and safe for usage over long period of time. In order to carry out the design process easily, ANSYS software is used do the finite element analysis to

study the structural behaviour of the bridge despite there are several methods. This is because by using computer software, more complicated criteria in bridge design such as curve can be easily analysis compared to manual calculation which has higher percentage of mistake and more time-consuming.

1.3 Objective

Every study and project comes with objectives as they are important for the research to be done well. Objective acts as reminder for the researcher of the initial criteria and testing that should be conducted in the research. As for this project, the main objectives are:

1. To make a model of beam bridge with adequate reinforcement.
2. To check on the structural behaviour of the structure in term of force and moment.
3. To carry out probabilistic analysis of the structure.

1.4 Scope of Study

This research focus mainly on the generation of model of bridge through the CivilFEM integrated ANSYS software. Eurocode 2 is use as the guide code on the anlaysis and the parameters are applied to obtain the result. This research will be analysing applying different parameter on the structure. The comparison of the result of will be made to show the mechanism of the design. Do exploration on ANSYS through the tutorial to get familiar with the operation.

1.5 Research Outcomes

Obtain the shear force diagram, bending moment diagram, deformed shape diagram and reinforcement factor diagram from ANSYS. Perform probabilistic analysis on the structure and obtain statistical data of input parameters and output parameter.

1.6 Research Question

1. What is the structural behaviour of voided concrete bridge slab when subjected to loading?

2. What is the critical status of voided concrete bridge slab when subjected to loading? How does it affect the structure when come in real situation?

REFERENCES

Chen, Z. and Zhou, Y. 2011. Dynamic analysis of hinge joint voided slab wide bridge. Abstract. 2011 International Conference on Electric Technology and Civil Engineering (ICETCE). Lushan, China: 22-24 April.

Chung, J.H.., Jung, H.S., Bae, B. et al. 2018. Two-Way Flexural Behavior of Donut-Type Voided Slabs. International Journal of Concrete Structures and Materials. 12: 26. https://doi.org/10.1186/s40069-018-0247-6 (20 March 2018)

Courant, R., 1943, Variational methods for the solution of problems of equilibrium and vibrations. Bulletin of the American Mathematical Society. 49: 1–23.

Gao, L., & Zeng, X. Y., & Wang, Y. (2011). Research on Several Problems about Nonlinear Analysis of RC Beams Based on ANSYS. Journal of Water Resources and Architectural Engineering, 92-96

Giuseppe Quaranta, 2011, Finite element analysis with uncertain probabilities, Computer Methods in Applied Mechanics and Engineering, Volume 200, Issue 1, pp.114-129.

G.Stefanou, 2009, The stochastic finite element method: past, present and future,Computer Methods in Applied Mechanics and Engineering, Volume 198, pp.1031-1051.

Hasan, M., Lu, M., Yi, C. and Zheng, C. 2018. Synthesizing engineering design, material takeoff and simulation-based estimating on a bridge deck reinforcement case. Abstract. 2017 Winter Simulation Conference (WSC). Las Vegas, NV, USA: 3-6 Dec.

Hock Tian, Cheng, Bashar S. Mohammed and Kamal Nasharuddin Mustapha, 2009, Experimental and analytical analysis of pretensioned inverted T-beam with circular web openings, International Journal of Mechanics and Materials in Design, Volume 5, No. 2; pp.203-215.

Hrennikoff, Alexander, 1941, Solution of problems of elasticity by the framework method. Journal of Applied Mechanics. 8.4: 169–175.

L.Figiel and M.Kaminski, 2009, Numerical probabilistic approach to sensitivityanalysis in a fatigue delamination problem of a two layer composite, Applied Mathematics and Computation, Volume 209, pp.75-90.

Lei, Z. and Sun, W. 2011. Effect of superposed deck on the static performance of a prestressed concrete hollow slab girder bridge. Abstract. 2011 International Conference on Electric Technology and Civil Engineering (ICETCE). Lushan, China: 22-24 April.

Li, B. M. (2012). Mechanical Property of Reinforced Concrete Beams with Openings. Fu Jian Construction Science and Technology, 35-38

Liu, Y. and Wu, X. 2011. Study on reinforce prestressed hollow slab by CFRP. Abstract. 2011 International Conference on Electric Technology and Civil Engineering (ICETCE). Lushan, China: 22-24 April.

Naito, Clay; Sause, Richard; Hodgson, Ian; Pessiki, Stephen; Macioce, Thomas, 2010, Forensic Examination of a Noncomposite Adjacent Precast Prestressed Concrete Box Beam Bridge. Journal of Bridge Engineering. 15(4): 408–418

Nan Zhang and Chung C.Fu, Experimental and theoretical studies on composite steel – concrete box beams with external tendons, Engineering Structures, Volume 31, 2009, pp.275-283.

Purushottam, Y. J., & Hemantkumar, T. Y., 2016, Analytical Study of Solid flat Slab and Voided Slab using ANSYS Workbench. International Research Journal of Engineering and Technology (IRJET), 1000-1003

Sadeghi, J. and Fathali M. 2012. Grillage analogy applications in analysis of bridge decks. Australian Journal of Civil Engineering. 10(1):23-36.

SAS, ANSYS 11.0 Finite Element Analysis System, 2009, ANSYS Manual Set, ANSYS Inc., Southpoint, 275 Technology Drive, Canonsburg, PA 15317, USA.

Tie Ming, 2011, Application of Probabilistic Analysis in Finite Element Modeling of Prestressed Inverted T-Beam with Web Openings, Vol. 4, (04): 350-366

Wang, L and Wang, S. 2011. A study of Jia-Musi highway bridge reinforcing materials test. Abstract. 2011 International Conference on Electric Technology and Civil Engineering (ICETCE). Lushan, China: 22-24 April.

Wu, S., Zhang, G., Han, J. et al. 2009. Fatigue performance of bridge deck pavement materials. Journal of Wuhan University of Technology-Mater. Sci. Ed. 24: 318. https://doi.org/10.1007/s11595-009-2318-7. (9 July 2009)

Yang J. 2010. Disease analysis and reinforcement methods discussion of several sorts of arch bridges. Abstract. 2010 International Conference on Mechanic Automation and Control Engineering. Wuhan, China: 26-28 June.

Yue, J., Wu, X., Huang, X. 2014. Effects of Live Load on the Deflection of Long-Span PC Continuous Rigid-Frame Highway Bridge Based on Midas. Abstract. 2014 7th International Conference on Intelligent Computation Technology and Automation. Changsha, China: 25-26 Oct