NUMERIACAL SIMULATION OF MEDIUM RISE BUILDINGS (SUBJECTED TO WIND LOADS) USING COMPUTATIONAL FLUID DYANAMICS (CFD)

By

W.H. DODANGODA

199191M

Master of Science Degree in Structural Engineering Design

Supervised by

Prof. M.T.R. JAYASINGHE

Department of Civil Engineering

Faculty of Engineering, University of Moratuwa

Moratuwa 10400, Sri Lanka.

April, 2024

DECLARATION

I hereby attest that the work I have submitted here is authentic and was created by myself for the partial fulfillment of master's degree in Structural Engineering at University of Moratuwa and has not been published or submitted elsewhere for the requirement of a degree Programme. Any literature, data or works done by others and cited within this dissertation has been given due acknowledgement and listed in the reference section.

…………………………………..…

W.H. Dodangoda,
Department of Civil Engineering,
University of Moratuwa,
Sri Lanka.

The above candidate has carried out research for the Masters Dissertation under my supervision.

……………………………………

Prof. M.T.R. Jayasinghe,
Senior Lecturer,
Department of Civil Engineering,
University of Moratuwa.

ABSTRACT

High-rise building industry has become more popular with the urbanization all over the world. Tall & medium rise buildings are more sensitive to the wind force than low-rise buildings. Wind action is more significant on the building when the building is getting taller. Therefore, wind induced action should be accurately predicted by considering the building height.

Traditional techniques, like code-based wind predictions, are unreliable on wind action when the height of the building increases. However, wind tunnel testing is widely used to predict the wind induced effect using a prototype with the geometric and dynamic similarity for all types of buildings and structures. But wind tunnel testing requires large, expensive equipment and time.

Numerical methods also can be used to simulate the wind flow behavior around the building. Computational Fluid Dynamic (CFD) method is a reliable option for wind analysis of tall buildings. Direct Numerical Simulation (DNS) is not possible because of insufficient computational power. Using turbulence models, solutions utilizing the Reynolds Averaged Navier-Stokes (RANS) equation can serve as a substitute for wind analysis. Various types of turbulent models can be developed in Midas NFX software for simulating the wind flow.

Nine rectangular-shaped buildings were subjected to rigid body modeling analysis using Midas NFX software inside a well-defined computational area. Obtained results were compared with the code base prediction results & wind tunnel testing results for all buildings. 2-equation k-ω (SST) turbulent model was used to predict the wind pressure distribution around the building and the normalized base overturning moment with the reasonable accuracy with 1m mesh size.

ACKNOWLEDGEMENT

I would like to show my sincere gratitude to my thesis supervisor Prof. M T R Jayasinghe for giving me the opportunity to do research under his supervision and providing valuable guidance.

I would like to express my gratitude to all of the other instructors in the Department of Civil Engineering at the Faculty of Engineering, as well as the panel of examiners who patiently revised my thesis, for your kind assistance in helping me to complete my master's and bachelor's degrees to this point.

I am extending my thanks to the Post Graduate Unit, Faculty of Engineering, University of Moratuwa to giving this opportunity to conduct research with expertise in this research area. I would also like to acknowledge to my batch mates who supported me to carry out this research.

I would like to thank my wife for encouraging me to complete the MSc. She was always supporting me with her best wishes. Also, I take this opportunity to thank Eng. Shiroshi Gunathilaka who assisted me in working with MIDAS NFX software required for this research.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES