

Studying Wind Flow-Field Around a Triangular Building by CFD and Wind Tunnel



Do N. Ngo, Y. Q. Nguyen, Anh T. Tran, and Hien T. Le

1 Introduction

Studying wind effects on tall building provides essential information for the purposes of designs of structures or natural ventilation of the building. Interaction between wind and building can be studied by experiments in wind tunnels [3, 8, 9], by numerical simulations [4, 10, 11, 13] or by both of them [7]. Those previous studies reported that the cross-sectional shape of the building is one of the most important factors affecting the pressure distribution on the surfaces of the building and the flow structure around the building.

In this studying, we investigated flowfield around a triangular cross-sectional building with a light well on one side and chamfered roof. Baghaei Daemei et al. [3] studied a triangular shaped building with seven different forms to determine the smallest drag coefficient for the design. However, none of their models has a light well.

The flowfield was investigated by numerical simulation with CFD (Computational Fluid Dynamics) technique and compared with results of experiments in a wind tunnel.

D. N. Ngo (✉)

Faculty of Civil Engineering, Ho Chi Minh City University of Technology, VNU-HCM, Ho Chi Minh City, Vietnam

e-mail: 1870018@hcmut.edu.vn

Y. Q. Nguyen

Van Lang University, Ho Chi Minh City, Vietnam

A. T. Tran · H. T. Le

Faculty of Transportation Engineering, Ho Chi Minh City University of Technology, VNU-HCM, Ho Chi Minh City, Vietnam