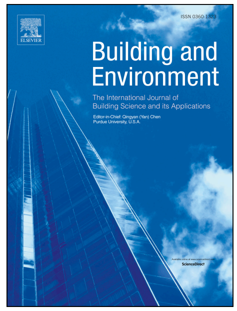
Accepted Manuscript



Influence of surrounding buildings on wind flow around a building predicted by CFD simulations

Sumei Liu, Wuxuan Pan, Xingwang Zhao, Hao Zhang, Xionglei Cheng, Zhengwei Long, Qingyan Chen

PII: S0360-1323(18)30271-3

DOI: 10.1016/j.buildenv.2018.05.011

Reference: BAE 5449

To appear in: *Building and Environment*

Received Date: 18 February 2018

Revised Date: 2 May 2018

Accepted Date: 4 May 2018

Please cite this article as: Liu S, Pan W, Zhao X, Zhang H, Cheng X, Long Z, Chen Q, Influence of surrounding buildings on wind flow around a building predicted by CFD simulations, *Building and Environment* (2018), doi: 10.1016/j.buildenv.2018.05.011.

Influence of surrounding buildings on wind flow around a building predicted by CFD simulations

Sumei Liu[a], Wuxuan Pan[a], Xingwang Zhao[a], Hao Zhang[a], Xionglei Cheng[a], Zhengwei Long[a,*], Qingyan Chen[b,a]

[a]*Tianjin Key Laboratory of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin, China*

[b]*School of Mechanical Engineering, Purdue University, West Lafayette, IN, USA*

ABSTRACT

This investigation used wind information from a meteorological station to conduct a CFD study of wind distribution in an urban configuration. The study treated the computational domain with detailed building structures only in the proximity of the urban configuration, and treated other regions with roughness. The resulting mean wind velocity was 5.5% higher than that computed with the use of detailed building structures between the meteorological station and the urban configuration, which is acceptable for most applications. This investigation then explored the influence of surrounding buildings on the wind flow around the target building with different geometrical models. The models differed in terms of the geometrical details around the building. The results showed that the surrounding buildings had a considerable impact on the wind flow around the target building due to the sheltering and channeling effect. When the details of the surrounding building structures were not taken into account, the wind flow around the building was inaccurate and unacceptable. The predicted wind speed and pressure distributions improved with an increase in the level of detail of the surrounding building structures. To achieve acceptable results, a CFD simulation should use detailed building structures around the target building within a radius of at least three times the length scale, where the length scale is the largest dimension of the target building. The results of the study can serve as a practical guide for predicting airflow around an urban configuration or a building.

Keywords: Urban wind flow, Geometric model, Surface roughness, Experimental verification

1. Introduction

The investigation of wind around buildings in an urban configuration in the lower part of the atmospheric boundary layer (0-200 m) [1] is crucial in many wind environmental problems, including natural ventilation design, pedestrian comfort, and air pollutant dispersion [2, 3]. Solving wind-environment problems requires the study of wind flow into an urban configuration [4, 5]. With the development of computing resources and grid-generation techniques, an increasing number of researchers have adopted computational fluid dynamics (CFD) in their investigation of urban wind environments [6-9]. The accuracy of CFD simulations relies on many factors, and to the knowledge of the authors, computational domain and representation of surroundings are the prerequisite to acquire accurate results [10].