Building and Environment 70 (2013) 161–170

Contents lists available at ScienceDirect

Building and Environment

journal homepage: www.elsevier.com/locate/buildenv

Dynamical computational fluid dynamics modeling of the stochastic wind for application of urban studies

Parham A. Mirzaei [a,b,c,*], Jan Carmeliet [a,b]

[a] Laboratory for Building Science and Technology, Swiss Federal Laboratories for Materials Testing and Research (Empa), Überlandstrasse 129, 8600 Dübendorf, Switzerland
[b] Swiss Federal Institute of Technology Zurich (ETHZ), Wolfgang-Pauli-Strasse 15, 8093 Zürich, Switzerland
[c] The University of Nottingham, Room A2a, Lenton Firs House, University Park, Architecture and Built Environment Department, Nottingham, NG7 2RD, UK

ARTICLE INFO

Article history:
Received 15 March 2013
Received in revised form
30 June 2013
Accepted 9 August 2013

Keywords:
Street canyon
Wind
CFD
Urban-scale
Cylindrical domain

ABSTRACT

For decades, the borders of building studies were restricted up to the exterior walls. With better understanding the side effect of urbanization on human health and with excessive progress in new investigation tools, the area of building studies were enlarged to neighborhood environment where mass and air transport vividly interact with the buildings. Unlike the indoor studies, one can emphasize the significance of the stochastic wind in outdoor studies.

Many experimental and simulation research have been conducted to verify the contribution of the wind velocity over/within street canyons/buildings. These works were mainly designed to observe the relations between wind flow and natural ventilation, pollution dispersion, pedestrian thermal/wind comfort, and drag resistance of the buildings. All these efforts continuously proved the significance of vortices caused by upstream wind as they interact on physical phenomena within the street canyons.

However, the main drawback of these studies is not attributed to the extracted results, but the way in which wind, an inherently transient and stochastic phenomenon, is presented by a steady state consideration as applied through the boundary conditions. For example, upstream wind in terms of direction and magnitude is widely considered as a constant and steady state profile induced through the street canyons. However, due to the transient and stochastic nature of encountered wind in the urban environment, these assumptions fail to capture the physics of the problems.

Therefore, the main point of concern in street canyon modeling should be described as constant and steady state reflection of stochastic and transient phenomena through the boundary conditions. Although there are existing studies related to the application of transient boundary conditions, a practical procedure to model the dynamic and stochastic behavior of wind direction is barely addressed in literature.

In light of the lack of stochastic wind modeling, this study intends to introduce for the first time an approach in order to generate dynamic wind. This is followed by a brief discussion of existing approaches in urban-scale wind modeling and their major shortcomings. For this purpose, a computational Fluid dynamics (CFD) model is developed considering a novel cylindrical computational domain. An analytical and a parametric study have been also conducted to obtain suitable sizes for the domain geometries. The advantages of the proposed model comparing to the traditional approaches are depicted using a case study of array of cuboids.

1. Introduction

For decades airflow modeling was the subject of many fundamental, theoretical, and experimental studies as it interacts a broad range of engineering problem. Beyond traditional models, Navier–Stokes (NS) equations was proposed as the most accurate fluid dynamics model. However, their application and validation were limited to very simplified case studies for a long period due to complexity of these equations. After excessive development of computational tools, computational fluid dynamics (CFD) was implemented to a large spectrum of problems from micro-scale to meso-scale. These techniques were also responding to many wind engineering demands.

One of the main applications of airflow modeling is in building energy simulation (BES). At the beginning, it was as paramount of

---

* Corresponding author.
E-mail address: Parham.Mirzaei_Ahranjani@nottingham.ac.uk (P.A. Mirzaei).

0360-1323/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.buildenv.2013.08.014

importance to calculate convective heat transfer from external skin of a building. The knowledge of the heat transfer coefficient is essential to calculate the heat balance of a building in order to simulate the energy consumption, as well as cooling and heating loads [1]. This heat transfer around the building was conventionally approximated with empirical correlations [2]; assuming that the stochastic behavior of wind can be modeled by heat transfer between a node on wall and a reference node in the street canyon in order to represent wind velocity and temperature. Nonetheless, a huge discrepancy in convective heat transport is mainly reported due to such an idealistic assumption.

With the progress in CFD techniques, it was feasible to obtain the heat transfer coefficient with much higher accuracy by including the exterior volume of the buildings, street canyons, in the transport equations. Beyond the application of wind engineering in BES, CFD also emerged as a powerful tool to resolve the transport phenomena in other subjects of urban-scale studies. This includes urban heat island mitigation and prediction studies [17,22,28], natural and/or mechanical improvement of HVAC systems ([1,8,18,24,25]), pedestrian thermal and wind comfort [4,15,23,32], and pollution dispersion [9,11,16,19,20,27,30]. Moreover, one should add the extensive and ongoing works being performed to analyze the drag effect on buildings for structural design purposes.

Despite the above mentioned research, the stochastic behavior of wind in urban-scale studies is barely addressed. Instead, wind is mainly assumed in its prevailing condition with a constant magnitude and direction. Moreover, efforts have been mainly conducted to generate the geometry and mesh based on one desired direction which could produce inaccurate results for other directions. Creation of several geometries and meshes for different orientations was also a solution to overcome discussed drawbacks [20,29] although it requires intensive time for preparation of the meshes and geometries. Furthermore, there would be a considerable discrepancy when it is intended to transfer data from one domain to another as the shape and place of cells can be varied in different domains. Therefore, it can be concluded that the mentioned domain preparation techniques are unsuitable especially in buoyant flow when the temperature distribution of street canyon surfaces (i.e. ground, walls, and roofs) is significantly affected by stochastic wind rather than a wind with constant velocity and magnitude.

The aim of this research is therefore to propose an approach in order to model dynamical stochastic behavior of wind in CFD simulations. A brief overview of the existing approaches to model wind is provided in the next section. Then, a cylindrical domain approach is explained to model the stochastic wind. Consideration on geometry generation of the cylindrical domain as well as its unstructured-structured curvilinear grid is further described. Performance of the proposed domain in comparison with a commonly used domain, i.e. a rectangular domain with structured rectilinear grid, is also investigated.

The novel cylindrical domain provides more realistic surface temperature due to changing of the convection and buoyant flow. It is also more feasible to investigate the dynamic impact of pollution, moisture, and air transport on pedestrian comfort and health. It should be mentioned that this technique can be applied and expanded to other wind engineering subject areas.

2. Approaches to model stochastic wind

Various simulation and experimental approaches have been conducted to represent the direction and magnitude variation of wind in urban-scale studies. As shown in Fig. 2a, a rectangular domain is a widely preferred domain in urban-scale studies. Wind is simulated with a temporal constant magnitude or profile [5,6,9,10,14,16,30], entering from downstream, counteracting studied objects, and exiting the domain towards upstream. Many parametric studies have been reported to obtain corresponding lengths L_1 to L_4 in order to minimize the influence of domain geometry on simulation results following by wind tunnel experiment to support the outcomes [31]. Beyond geometry considerations, grid type and structure were also extensively examined for rectangular domains [31]. Hefny and Ooka [12] studied and approved the effectiveness of hexahedral meshes comparing to tetrahedral meshes despite the considerable time and efforts required to prepare such domains.

In many 2D and 3D studies, such as flow over long street canyon, the worst scenario case is mostly assumed where the wind direction is perpendicular to inflow plane. However, this would be an idealistic assumption when the studied object has a 3D geometry or the influence of the oblique wind above the object is also frequent and significant. To overcome this shortcoming, the concept of “frequency of occurrence” is used [20,21,29]. The magnitude of the approaching wind is mainly simplified with limited numbers according to the existing prevailing and frequent winds on the investigated area. For example Ref. [20], discreted Montreal wind rose to three directions (i.e. 0°, 45°, & 90°) and three magnitudes (i.e. 1, 3, and 7 m/s). In frequency of occurrence approach, wind rose is analyzed to obtain the probability of each wind direction and magnitude with certain accuracy. For example, the illustrated wind rose in Fig. 1 can be divided into 36 directions (e.g. 0°–10°, 10°–20°, 20°–30°, ..., 340°–350°, & 350°–0°) and six magnitudes (e.g. 0–1 m/s, 1–2 m/s, 2–3 m/s, 3–4 m/s, 4–5 m/s, and 5–6 m/s). Thus, it is possible to model the stochastic wind with the creation of 36 different rectangular domains and by 36 times rotating the objects in each domain. Then, six simulations have to be conducted for each domain in order to cover different range of the wind velocity. These frequencies of occurrences are also depicted in Fig. 1.

The limitation of the rectangular domain to model the stochastic wind is related to first the number of generated meshes (e.g. 36 meshes are required for the description of stochastic wind as in Fig. 1). Moreover, when the ground and walls are not in isothermal condition for instance in existence of radiation and other heat fluxes, interpolation has to be used introducing inaccuracy while transferring data from one domain to another, since the type and size of cells on walls and ground would not be identical in different domains of different wind directions.

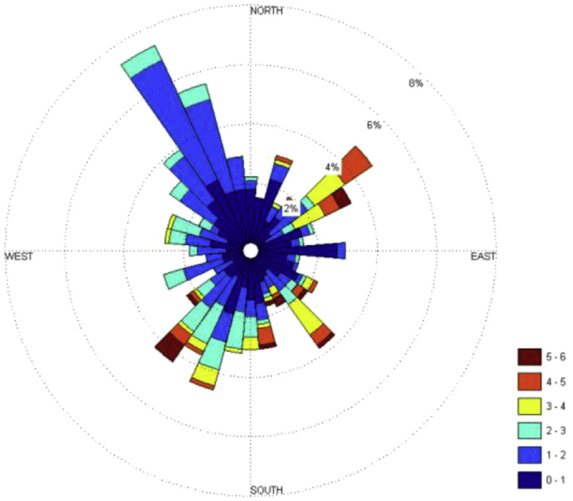

Fig. 1. Percentage of Zurich annual wind direction distribution for discreted Zurich wind to 36 directions (e.g. 0°–10°, 10°–20°, 20°–30°, ..., 340°–350°, & 350°–0°), and six magnitudes (e.g. 0–1 m/s, 1–2 m/s, 2–3 m/s, 3–4 m/s, 4–5 m/s, & 5–6 m/s) [source: FOEN/NABEL].