Applied Energy 182 (2016) 625–633

Contents lists available at ScienceDirect

Applied Energy

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

Defining the Influence Region in neighborhood-scale CFD simulations for natural ventilation design

Zheming Tong *, Yujiao Chen, Ali Malkawi

Center for Green Buildings and Cities, Harvard University, Cambridge, MA 02138, USA
Graduate School of Design, Harvard University, Cambridge, MA 02138, USA

HIGHLIGHTS

- The accuracy of natural ventilation analysis relies largely on how the Influence Region is chosen.
- Only including the adjacent layer of surrounding buildings is not sufficient.
- Three layers of surrounding buildings are typically required for modeling low-rise neighborhoods.
- Fewer surrounding buildings are required for wide canyons and high-rise landscapes.
- Downstream buildings can be moderately excluded in the Influence Region.

ARTICLE INFO

Article history:
Received 31 March 2016
Received in revised form 27 June 2016
Accepted 17 August 2016
Available online 7 September 2016

Keywords:
Natural ventilation
Influence Region
Street canyon
Air change rate (ACH)
CFD

ABSTRACT

Natural ventilation is one of the most important design options for green buildings, which reduces energy use and improves thermal comfort. Computational Fluid Dynamics (CFD) simulations have been used increasingly for natural ventilation design in urban neighborhoods. The accuracy of such simulations relies largely on how the CFD domain is chosen. In the domain, we define the Influence Region as the area where the surrounding buildings must be modeled explicitly to predict the ventilation flow rate accurately. This study presents the early efforts to determine the adequate size of the Influence Region in the CFD domain using a coupled indoor-outdoor CFD simulation, in which the air change rate (ACH) no longer varies noticeably with increasing number of surrounding obstacles. Convergence charts of ACH as a function of an increasing number of surrounding building layers are generated using various urban parameters (e.g., wind condition, aspect ratio, building height relative to surroundings, downstream obstacles, and non-idealized surroundings). Our analysis demonstrated that only including the adjacent layer of surrounding obstacles is not sufficient for predicting correctly the ACH because of the artificial channeling effect between buildings. For both normal and oblique wind directions, three layers of surroundings are required for regular street canyons with an aspect ratio H/W = 1. In the case of wide canyons (H/W = 1/3), two layers of surroundings are needed because there is less flow interference between upstream and downstream obstacles. For the urban configuration, where the target building is significantly taller than nearby structures, the ACH on higher floors does not vary much with increasing amount of surroundings, which significantly reduces the required number of buildings in the Influence Region. In addition, buildings at the side and downstream of the target building can be moderately excluded in the Influence Region as long as the most adjacent downstream layer of obstacles is modeled. A real urban configuration with non-uniform spacing among buildings is evaluated. We showed that the required size of the Influence Region that is derived from uniform building arrays still generally applies to non-idealized landscapes. This study demonstrates the importance of assessing the sensitivity of the selected Influence Region in CFD simulations to reduce unintended modeling errors and computing expense.

1. Introduction

Natural ventilation has become an increasingly attractive design option in the building industry because of the recent focus

* Corresponding author at: Center for Green Buildings and Cities, Graduate School of Design, Harvard University, Cambridge, MA 02138, USA.
E-mail address: ztong@gsd.harvard.edu (Z. Tong).

http://dx.doi.org/10.1016/j.apenergy.2016.08.098
0306-2619/

on sustainable design [1–3]. It is the process that utilizes natural outside air movement to both cool and ventilate a building passively without the use of any mechanical system. Studies have demonstrated that the cooling energy consumption of naturally ventilated buildings can be reduced by as much as 40–50% in comparison with an air-conditioned building [4–15].

With the aid of the recent advance in computing technology, CFD is widely used to understand the urban physics in the lower atmospheric boundary layer, due to the relatively low cost, high data resolution, and adequate validation with experimental data [16–26]. Neighborhood-scale simulation is critical to study various physical mechanisms such as local energy demand, pollutant transport, and thermal comfort [23,27–34]. Successful designs of natural ventilation system rely largely on the configuration of surrounding neighborhood and the airflow over and through naturally ventilated buildings [35]. Therefore, CFD has been shown as a valuable tool for natural ventilation designs [3,36–46]. Although CFD is a promising tool, appropriate implementation is required to obtain reliable simulation results. The existing literature has provided best practice guidelines to determine the size of the CFD domain, boundary conditions, and proper mesh resolution [17,47,48].

In Fig. 1, the size of the CFD domain is determined based on the blockage ratio that avoids artificial acceleration by non-physical boundaries. In particular, the inlet, lateral, and top boundaries should be at least 5H away from the Influence Region where buildings are explicitly modeled (H is the height of the target building). The outflow boundary is recommended to be at least 15H away from the Influence Region. The obstacles in the upstream, downstream, and lateral areas of the domain are not included explicitly but their effect on the flow can be parameterized in terms of surface roughness in the wall function. In the highlighted Influence Region (Fig. 1), obstacles such as individual buildings and streets that are in close vicinity to the target building must be modeled explicitly with their geometrical shapes. Researchers have demonstrated that the pressure distribution on a building is greatly influenced by surrounding structures, i.e., the sheltering effect, through both CFD simulation and wind tunnel experiments [35,49–54].

In an urban environment, buildings are grouped closely together. To predict the ventilation flow rate accurately, it is essential to explicitly model a sufficient number of surrounding buildings in the Influence Region. A key question that has not been addressed thoroughly in the literature is how many surrounding obstacles should be included in the Influence Region of the CFD domain. In general, if the modeled Influence Region is larger, the CFD domain needs to be larger, and, therefore, more computational time is required. Conversely, underrepresenting the surroundings leads to incorrect prediction of the ventilation rate at the target building. To fully address this question regarding the required size of the Influence Region, we chose ACH as a primary indicator to derive the proper size of the Influence Region that would capture the effect of surroundings sufficiently, yet maintain a reasonable computational cost.

In practice, ACH is often considered as a target criterion to assess the adequacy of natural ventilation for acceptable indoor air quality and thermal comfort. It is a straightforward measure for building engineers and it is often used as a "rule of thumb" in ventilation design. There are two common approaches to determine ACH in the design stage. The most common strategy is based on the orifice equation as described in Eq. (1).

$$ACH = \frac{C_D \cdot A \cdot u_{ref} \cdot \sqrt{\Delta C_p}}{\sqrt{2} \cdot V} \tag{1}$$

where C_D is the discharge coefficient. A is the area of the opening. u_{ref} is the reference wind speed. ΔC_p is the difference in surface-averaged wind pressure coefficient between windward and leeward walls, and V is the volume of the cross-ventilated room. The literature suggests that C_D is in the range of 0.60–0.65 for sharp-edged openings. The value of C_p can be obtained from either direct measurement (i.e., field experiment and wind tunnel), or indirect measurement that includes sources such as existing C_p database (AIVC, ASHRAE) or regression models that are based on a large amount of empirical data from wind tunnel studies (e.g., TNO C_p generator) [55–58]. Although the sheltering effect by buildings can be partially considered in these databases using various approximations, de Wit and Augenbroe [59] pointed out that these methods, which are based on interpolation or extrapolation of existing wind pressure coefficients, can introduce considerable uncertainties. Overall, the orifice equation is derived based on Bernoulli's assumption. However, in reality, flow through window openings is never laminar. In addition, past studies have shown that the use of C_D involves many uncertainties and C_D also varies with several flow variables [60–62]. In wind tunnel studies, the volumetric flow rate through cross-ventilated buildings cannot be

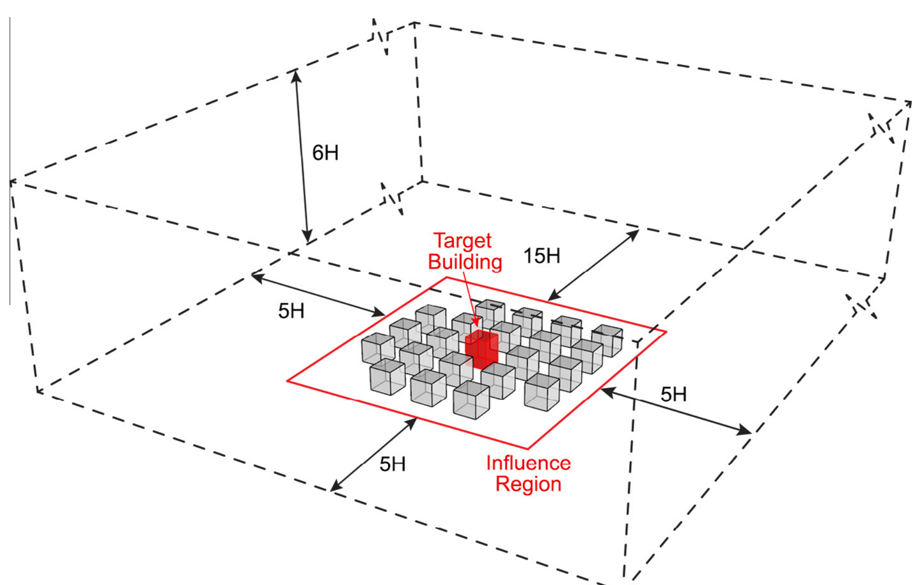

Fig. 1. Graphic illustration of the CFD domain and Influence Region. H is the height of target building in the Influence Region.