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.

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.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

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

\[
ACH = \frac{C_D \cdot A \cdot \frac{u_{ref}}{C_1} \cdot \sqrt{\Delta C_p}}{\sqrt{2V}}
\]  

(1)

where \( C_D \) is the discharge coefficient. \( A \) is the area of the opening. \( u_{ref} \) is the reference wind speed. \( \Delta C_p \) 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.](image-url)
measured directly. Instead, $C_{p}$ is measured on solid buildings surfaces, which assumes the flow field around the building without openings is equivalent to that with openings. It also implies that the approaching flow stagnates at the window and the turbulent kinetic energy is dissipated at the window opening. For cross-ventilated buildings with large openings, this approach can introduce significant errors [44,63]. The second approach is to compute $ACH$ through a coupled indoor-outdoor CFD simulation, which directly calculates the volumetric flow rate through openings. The coupled indoor-outdoor CFD simulation resolves the outdoor and indoor wind flow within the same computational domain and is able to capture the dynamic interaction between the outdoor and indoor environment at the window openings [44,46]. This method, in contrast to the first one, eliminates assumptions and uncertainties in the orifice equation such as the estimation of $C_{D}$ and $C_{p}$.

In this study, we present the early efforts to determine the required size of the Influence Region in CFD simulation in which $ACH$ no longer varies noticeably with increasing number of surrounding obstacles using a coupled indoor-outdoor CFD simulation. Convergence charts of $ACH$ as a function of number of surrounding building layers are generated using various urban parameters (e.g., wind condition, aspect ratio, relative building height to surroundings, downstream obstacles, and non-idealized building configuration). Our goal is to address an important, but often overlooked, issue in the setup of neighborhood-scale CFD simulations, and complement existing best practice guidelines for the design of natural ventilation in dense urban landscapes. The paper is organized as follows. We first describe the setup of the CFD simulation. The CFD model is then evaluated against measurements from a wind tunnel experiment. Secondly, we derive the required size of the Influence Region in CFD simulation in which $ACH$ no longer varies noticeably with increasing number of surrounding building layers. A summary of the findings is provided in the conclusion.

2. Model description

2.1. Turbulence modeling

Large Eddy Simulation (LES) with a dynamic subgrid model is employed here. It is the appropriate turbulent model for simulating highly unsteady and three dimensional flows in urban street canyons. LES resolves large-scale unsteady motions and requires modeling only the small-scale, unresolvable turbulent motion that is less influenced by physical boundary conditions [26]. Although LES is more computationally demanding, it shows superior performance for modeling flows that are highly three dimensional in urban street canyons, and general modeling guidelines were developed [23,64–66].

In LES, a low-pass filtering operation is performed so that the resulting velocity field $\tilde{u}$ can be resolved on a relatively coarse grid. A dynamic subgrid model is chosen, which allows the Smagorinsky constant to vary in space and time [67]. The constant $C_{0}$ is modeled according to Eq.(5) [67].

$$C_{0} = \frac{1}{3} \delta_{kk} C_{0} = \frac{1}{2} \mu \frac{S_{ij}}{l_{r}}$$

$S_{ij}$ is the filtered stress tensor, and $\sigma_{ij}$ is the subgrid-scale Reynolds stress, which is modeled using the Boussinesq hypothesis in Eq. (4).

$$\sigma_{ij} = \frac{1}{3} \delta_{kk} \sigma_{kk} = -2 \mu \frac{\partial u_{i}}{\partial x_{j}}$$

$S_{ij}$ is the rate of strain tensor of the resolved scale. The subgrid viscosity $\mu_{t}$ is modeled according to Eq. (5) [67].

$$\mu_{t} = \rho \langle l_{r} \rangle^{3} \langle S \rangle$$

$L_{r}$ is the mixing length scale that depends on the size of the computational cell and the dynamically computed Smagorinsky constant.

2.2. Model setup

The coupled indoor-outdoor CFD simulation resolves the outdoor and indoor wind flow within the same computational domain and is able to capture the dynamic interaction between the outdoor and indoor environment at the window openings [44,46]. Depending on the simulated scenarios in Table 1, the dimensions of the computational domain are 210–1100 m in length.

Table 1

<table>
<thead>
<tr>
<th>Urban parameters selected to determine the size of Influence Region.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Variables</strong></td>
</tr>
<tr>
<td>(1) Array Size</td>
</tr>
<tr>
<td>(2) Wind Direction</td>
</tr>
<tr>
<td>(3) Aspect Ratio</td>
</tr>
<tr>
<td>(4) Target Building Height</td>
</tr>
<tr>
<td>(5) Downstream Buildings</td>
</tr>
</tbody>
</table>
210–1100 m in width, and 60–180 m in height, which satisfy the directional blockage ratio suggested in the guidelines. The computational domain contains 2–11 million unstructured cells depending on the modeling scenario. Although a grid-independent solution is not available for LES, grid sensitivity studies are conducted to ensure the time-averaged results of the selected grid do not vary noticeably with finer grid resolution. The grids are more refined near the solid cube. Prism layer mesh is applied near the wall in order to reduce numerical diffusion. The horizontal homogeneity is evaluated by creating a flow simulation in an empty domain of the same size as suggested by Blocken et al. [18], which ensures that there is no unintended difference between the inlet and incident profiles.

The numerical setup of the boundary conditions in this study follows the CFD guidelines for simulating flows in the urban neighborhood [17,47,48]. For the inlet, a logarithmic inlet velocity profile was employed according to Eq. (6).

\[
u(z) = \frac{u'_*}{k} \ln \frac{z + z_0}{z_0}
\]

where \(k\) is von Karman constant (0.42), and \(z_0\) is aerodynamic roughness height (0.025 mm). \(u'_*\) is the friction velocity. Turbulent kinetic energy \(k\) is estimated according to Eq. (7) [48]. The turbulence dissipation rate is derived from Eq. (8).

\[k(z) = \left(l(z)u(z)\right)^2\]
\[
\epsilon(z) = \frac{u'^3}{k(z + z_0)}
\]

The time-dependent feature of the inlet turbulence profile is simulated with the vortex method, where random vortices in the inlet flow plane for the wall-normal components are generated, providing a spatial correlation [23,69,70]. Symmetry boundary conditions are applied for the two sides as slip walls with zero shear. Outflow boundary condition is specified at the end of the domain. Additional details regarding the model setup are presented in our previous study [46]. The inlet boundary condition is obtained based on measured wind speed and turbulent intensity. Symmetry boundary conditions are applied for the two sides as slip walls with zero shear. Outflow boundary condition is specified at the end of the domain. Additional details regarding the model setup are presented in our previous study [46].

Validation is an essential part of model evaluation that provides quantitative views of the model performance. The accuracy of the simulations was examined by comparing the numerical results with the results of wind tunnel experiments conducted by Karava et al. [73]. The flow field through and over a scaled cross-ventilated building model in a boundary layer wind tunnel was measured with Particle image velocimetry (PIV). Building models with different openings were built from 2 mm cast transparent polymethylmethacrylate sheet at a scale of 1:200. The modeled buildings have a dimension of \(L \times W \times H = 100 \times 100 \times 80 \text{ mm}^3\) at wind-tunnel scale and \(20 \times 20 \times 16 \text{ m}^3\) at real scale. The thickness of the wall is 2 mm. Configurations with both openings at the center of the two opposite walls and with Window-to-Wall ratio (WWR) of 5% and 10% are shown in Fig. 2a and b. The height of the opening is 18 mm for both configurations while the width is varied to provide the corresponding porosity. The PIV measurements were conducted in the vertical plane of symmetry through the openings.

The time-dependent feature of the inlet turbulence profile is simulated with the vortex method, where random vortices in the inlet flow plane for the wall-normal components are generated, providing a spatial correlation [23,69,70]. Symmetry boundary conditions are applied for the two sides as slip walls with zero shear. Outflow boundary condition is specified at the end of the domain. Additional details regarding the model setup are presented in our previous study [46].

The performance of the LES model is satisfactory. For the 5% WWR case, the model was able to accurately capture the flow acceleration near both openings, as well as deceleration inside the building. The 10% WWR case presents a less accurate agreement. This is likely due to the size and position of the upstream vortex in the PIV experiment not being able to be reproduced in the LES simulation. Discrepancies are observed at the inlet and outlet openings \((x/D \sim 0)\) and floor \((x/D \sim 1.1)\). As discussed by Karava et al. [73], the experimental data is not reliable in the immediate vicinity of the openings due to the effects of shadows or reflections that might have produced uncertainties in the PIV measurements at these positions.

3. Determining the Influence Region

The Influence Region is defined here as the area where the structures near the location of interest must be explicitly modeled.
in order to accurately predict the ventilation flow rate. To determine the required size of Influence Region, we generated convergence charts of time-averaged ACH as a function of increasing number of surrounding layers under various parameters. Table 1 presents key urban parameters including wind condition, aspect ratio, building height relative to surroundings, and number of downstream obstacles. The selection of these parameters are based on existing $C_p$ generators derived from a large amount of wind measurement [56,57]. As shown in Table 1, the target cross-ventilated building has a dimension of $10 \times 10 \times 10$ m$^3$ with a WWR of 10%. To reach certain generalizability, matrices of cubical buildings are employed to represent idealized urban geometries. We selected a square matrix of buildings with a centered target building (height $H = 10$ m and aspect ratio $H/W = 1$) under normal wind direction as the baseline. A non-idealized building configuration with non-uniform spacing based on a real urban neighborhood in Massachusetts, USA was also examined at the end (Section 4.5).

4. Results and discussion

Fig. 4 presents the ACH computed directly from CFD as a function of the number of surrounding layers ($n$) for the baseline condition. The ACH declines substantially with increasing size of the Influence Region due to the sheltering effect. In the case of a single building without surroundings, the wind strikes the building and induces a positive pressure on the windward face and negative pressure on the leeward face, which then creates a large air flow through the openings. The maximum pressure occurs at the local stagnation point where the oncoming flow impinges the building surface. transitioning from an isolated building configuration to the configuration surrounded by layers of buildings, the ACH declines significantly due to the addition of upwind buildings. This is because the target building at the center is no longer struck directly by the approaching flow. As visualized in Fig. 5a, a low-pressure wake region with counter-rotating vortices is formed behind each building. The approaching flow is channeled into the street canyon between buildings. The flow is accelerated in parallel to the oncoming wind direction in the canyon, and decelerates gradually with increasing distance from the leading edge of the first building row. This observed flow pattern is consistent with the literature on flow passing a matrix of cubical obstacles [26,74–77]. The magnitude of flow acceleration or the channeling effect is strongest in the canyons at the first building row that are directly attacked by the approaching flow. As a result, the pressure at the wake zone behind the first row is lower than that behind the next building row, which creates a reverse flow from downwind to upwind direction. Our analysis shows that the influence of each additional surrounding layer on ACH decreases with increasing distance from the target building at the center. The channeling effect around the target building attenuates as the
upwind distance increases. The effect becomes negligible near the target building, and ACH reaches a convergence when the number of surrounding building layers increases to three (n = 3).

4.1. Wind direction and speed

The effect of wind direction (or attack angle) is very important to the flow pattern through and around the target building. In contrast to normal wind direction from the baseline condition, the flow field around the building is more “streamlined” under the oblique wind direction (45°). Instead of striking the windward face and creating a stagnation zone, the flow initially diverges at the windward corner and travels along the surface until a separation occurs at the leeward sharp corner (visualized in Fig. 5b). One stream of the flow feeds into two symmetric counter-rotating vortices formed with respect to the 45° diagonal line behind the building. The other stream channeled into the far side of the street re-converges with the one from the symmetric side before hitting the downstream building. As a result, the ACH for the oblique wind direction is noticeably greater than that of the normal direction except for the isolated configuration where the sheltering effect of surroundings is absent. Similar to the normal wind direction, ACH reaches a plateau at n = 3 (three surrounding layers).

4.2. Aspect ratio H/W

The aspect ratio (H/W) of surrounding street canyons is shown to have a great impact on the flow pattern around buildings [78]. In addition to a regular canyon (H/W = 1, H = 10 m), a wide canyon (H/W = 1/3, H = 10 m) is investigated, which is observed with less flow interference between upstream and downstream buildings in the literature [79]. In contrast with the regular canyons, the ACH for the wide street canyons is noticeably greater because there

In general, increasing the wind speed increases the cross ventilation and ACH at the target building. However, the increase in ACH attenuates with a growing number of surrounding layers due to the sheltering effect. For the single building configuration without any surroundings, doubling the wind speed doubles the ACH under both wind directions. However, in the case of multi-layer configurations, the percentage increase in ACH falls with increasing number of surrounding layers under normal wind direction. In comparison, the percentage increase in ACH remains nearly unchanged with increasing number of layers under oblique wind direction. This is because the high windward pressure created by the stagnation effect is greatly reduced when the attack wind angle is oblique (Fig. 5b). With regard to the convergence of ACH, our analysis suggests that three layers of surrounding buildings (n = 3) are still necessary regardless of attack wind angle.

Fig. 5. Velocity contour plot of the flow field with arrows indicating the flow pattern of a 5 by 5 matrix under (a) normal and (b) oblique (45°) wind directions.

Fig. 6. (a) Convergence chart of ACH as a function of increasing number of surrounding layers (n) for baseline wind speed (u) and twice the baseline wind speed (2u); (b) the convergence chart of ACH under two street canyon aspect ratios. n is the number of surrounding layers.
is a sufficient distance between obstacles that allows the airflow to re-converge before encountering the next obstacle. In Fig. 6b, a dip in ACH is observed at the \( n = 1 \) due to strong channeling effect and low pressure wake zone behind the first row of buildings, which diminishes when the number of surrounding layers increases. In the case of a wide canyon, two layers of surrounding buildings (\( n = 2 \)) are required to be modeled explicitly in the Influence Region.

### 4.3. High-rise building

If the relative height of the target building is significantly greater than the surroundings, the required size of the Influence region will be affected. As shown in Table 1 (4), we designed another configuration where a cross-ventilated high-rise is surrounded by low-rise. Fig. 7 depicts the convergence charts of ACH at each floor with increasing number of layers. The ACH at the ground level is mostly affected by the size of the surroundings. On the other hand, the ACH of the third floor does not vary much with increasing number of surrounding layers, because the portion of the building that is taller than the surroundings acts essentially as an isolated building. This suggests that cross-ventilated high-rise is less influenced by nearby obstacles and therefore requires only two layers of surroundings (\( n = 2 \)).

### 4.4. Downstream and lateral obstacles

The presence of downstream obstacles may play a less important role in ACH than the upstream obstacles. Here, we explored the impact of downstream buildings on ACH of the target building by removing one downstream layer at a time as displayed in Table 1 (5). This sensitivity study is based on a three-layer configuration derived previously. The resulting convergence chart of ACH as a function of the number of layers removed is presented in Fig. 8. Instead of modeling the downstream area with the same amount of upwind surrounding buildings, we demonstrate that only the most adjacent downstream layer is required under both normal and oblique wind direction, which to a large extent, reduces the computational cost.

### 4.5. Non-idealized landscape

In addition to idealized building matrices with uniform spacing employed previously, we chose a non-idealized landscape with non-uniform spacing and unequal building dimensions based on a real urban neighborhood in Massachusetts, USA to examine the size of the Influence Region. In Fig. 9, each surrounding layer is color coded. Similar to the uniform building arrangement, the convergence chart of ACH is shown in Fig. 10 where ACH falls with increasing number of layers. The same channeling effect is
observed at the one-layer configuration where there is a drop in ACH at n = 1 (Fig. 10). For both wind directions, the convergence is found at n = 3, which is equivalent to that from uniform building arrays. Although the irregularity alters the flow pattern to certain extent, the substantial decrease in wind speed towards the center of the building matrix reduces the ACH greatly regardless of the building arrangements. Therefore, our findings indicate that the required size of the Influence Region that is derived from uniform building arrays can be generally applied to non-idealized configurations.

5. Concluding remarks

The accuracy of natural ventilation analysis relies largely on how the Influence Region is chosen in neighborhood-scale CFD simulations. A sufficient amount of buildings near the point of interest (i.e., a cross-ventilated building) must be modeled explicitly in order to accurately predict the ventilation flow rate and ACH. Although the sheltering effect by surrounding buildings is recognized in the literature, the derivation of the required size of the Influence Region has not been analytically investigated. In this study, we present the early efforts to derive the required size of the Influence Region in the CFD domain in which the ACH at the cross-ventilated building no longer varies with increasing number of surrounding obstacles using a coupled indoor-outdoor CFD simulation. The impact of various urban parameters (e.g., wind condition, aspect ratio, building height relative to surroundings, downstream obstacles, and non-idealized surroundings) on ACH are evaluated against the number of surrounding layers. The conclusions from our analysis are briefly outlined. Only including the adjacent layer of surrounding obstacles is not adequate to correctly predict the ACH due to the artificial channeling effect between buildings. We found that three layers of surroundings (n = 3) must be modeled explicitly for regular street canyons (H/W = 1) under normal and oblique wind directions whereas the Influence Region can be reduced to two layers (n = 2) for wide canyons (H/W = 1/3) and high-rise configuration. 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 obstacles are modeled. A real urban configuration with non-uniform spacing and dimensions is also evaluated. Our analysis indicates that the irregularity does not noticeably change the required size of the Influence Region.

The derived conclusions are not intended to be exhaustive, and they may vary to certain degrees depending on the actual land-scape. Our aim here is to address an important, but often overlooked, issue in the setup of neighborhood-scale CFD simulations, and complement existing best practice guidelines for the design of natural ventilation in dense urban areas. Given the complexity of real urban landscapes, it is advisable to assess the sensitivity of any selected Influence Region following the steps presented here to avoid unintended modeling errors and unnecessary computing expense.

Acknowledgment

Z.T. and Y.C. are grateful for the postdoctoral fellowship from the Center for Green Buildings and Cities (CGBC) at Harvard University.

References
