Natural ventilation of an isolated generic building with a windward window and different windexchangers

Citation for published version (APA):

Castillo, J. A., Huelsz, G., van Hooff, T., & Blocken, B. (2019). Natural ventilation of an isolated generic building with a windward window and different windexchangers: CFD validation, sensitivity study and performance analysis. *Building Simulation*, 12(3), 475-488. <https://doi.org/10.1007/s12273-018-0502-4>

Document license:
TAVERNE

DOI:
[10.1007/s12273-018-0502-4](https://doi.org/10.1007/s12273-018-0502-4)

Document status and date:
Published: 01/06/2019

Document Version:
Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

www.tue.nl/taverne

Take down policy

If you believe that this document breaches copyright please contact us at:

openaccess@tue.nl

providing details and we will investigate your claim.

Natural ventilation of an isolated generic building with a windward window and different windexchangers: CFD validation, sensitivity study and performance analysis

J. Antonio Castillo¹ (✉), Guadalupe Huelsz², Twan van Hooff^{3,4}, Bert Blocken^{3,4}

1. Centro de Investigaciones en Arquitectura, Urbanismo y Paisaje, Facultad de Arquitectura, Universidad Nacional Autónoma de México, Av. Universidad 3000, 04510, Coyoacán, Ciudad de México, Mexico

2. Instituto de Energías Renovables, Universidad Nacional Autónoma de México, 62580, Temixco, Morelos, Mexico

3. Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, P.O. box 513, 5600 MB Eindhoven, the Netherlands

4. Building Physics Section, Department of Civil Engineering, KU Leuven, Kasteelpark Arenberg 40, bus 2447, 3001 Leuven, Belgium

Abstract

Windexchangers are relatively small structures located on the building rooftop to promote natural ventilation. This paper presents a Computational Fluid Dynamics (CFD) validation study, sensitivity analysis and performance comparison of three windexchanger (WE) configurations applied to a generic isolated building with a windward window. The study is limited to wind-driven (isothermal) ventilation, for wind perpendicular to the windward facade. The CFD simulations are based on the 3D-steady Reynolds-Averaged Navier–Stokes (RANS) equations. The validation study is performed with experimental results from a previously published water channel test. The sensitivity analysis focuses on the domain size, grid resolution and turbulence model. The performance evaluation of the three WE configurations is based on the mean velocity and mean static pressure coefficients in the vertical centerplane, the ventilation volume flow rate and the volume percentage of the living zone with air speed ratio equal to or above 0.10. The WE configuration with four openings and one duct shows the highest ventilation flow rate (0.232 m³/s) and the highest volume percentage (21%). This study shows that the assessment and selection of WE configurations should not only be based on volume flow rate or ACH but should consider the living zone air speed ratio as well, specifically concerning the flow distribution in the living zone.

Keywords

windexchanger,
windcatcher,
natural ventilation,
CFD,
validation

Article History

Received: 7 August 2018

Revised: 11 November 2018

Accepted: 23 November 2018

© Tsinghua University Press and
Springer-Verlag GmbH Germany,
part of Springer Nature 2019

1 Introduction

A windexchanger (WE) is a relatively small structure located on the building rooftop to promote natural ventilation (Cruz-Salas et al. 2014). This structure is more commonly referred into the literature as windcatcher, but the term windexchanger accounts for the fact that this structure can act either as an injector or as an extractor of air depending on the orientation when the building has another opening in one of the facades (Su et al. 2008; Liu et al. 2011; Cruz-Salas et al. 2014; Castillo et al. 2017) or can act simultaneously as an injector and extractor when it has no other opening (Elmualim and Awbi 2002; Elmualim 2006a; Li and Mak

2007; Su et al. 2008). A WE can be a suitable alternative to increase natural ventilation when the construction area or surrounding buildings impose limitations to use cross-ventilation by two windows, one at the windward and the other at the leeward facade (Karava et al. 2011; Etheridge 2012; Bangalee et al. 2013; Shetabivash 2015; Carrilho da Graça et al. 2015; Perén et al. 2015a).

In hot and specially in hot-humid climates, natural ventilation is an important thermal comfort strategy, that can yield important energy savings (Oropeza-Perez and Østergaard 2014). A Mexican official recommendation includes this strategy for those climates (CONAVI 2010). In this case, the airflow distribution inside the living zone

of the building becomes relevant since the increase of the airflow speed can increase the sweat evaporation rate. More than 70% of the geographical area of Mexico has a hot climate, and most of the dwellings that are being constructed in Mexico have one or two floors, with only one opening per room. In general, the construction area imposes limitations in increasing the ventilation by two windows in opposite walls of a room, thus the use of a windexchanger on the roof of these dwellings has been proposed to increase natural ventilation (Cruz-Salas et al. 2014, 2018; Castillo et al. 2017). Building geometries and wind conditions have to be investigated in order to provide guidelines for the design of WEs that promote an airflow distribution that is as uniform as possible within the living zone of a building and a sufficiently high airflow rate.

WEs exhibit a great variety of geometric configurations, and their performance is a function of the building and WE geometry (Montazeri and Azizian 2008; Montazeri et al. 2010), the existence and location of others openings, e.g. windows, in the building and the wind conditions, i.e. speed and direction (Montazeri et al. 2010; Cruz-Salas et al. 2014; Castillo et al. 2017) and the internal arrangements in the building, such as window louvers, furniture, etc. Experiments are an important methodology for WE performance assessment, but using this method can be a slow and tiresome process when parametric analyses need to be performed. In contrast, Computational Fluid Dynamics (CFD) simulations are generally considered to be more suitable for such analyses. CFD allows full control over the boundary conditions and easily and efficiently allows parametric studies to be performed to evaluate alternative design configurations and provides so-called whole flow-field data, i.e. information about the computational variables in all points of the computational domain (Murakami 1997; Stathopoulos 1997; Baker 2007; Solari 2007; Tominaga and Stathopoulos 2013; Meroney and Derickson 2014; Blocken 2014, 2015). Because of these reasons and because of the increase in computational resources, in the past decades, CFD has become a common methodology for natural ventilation studies (Chen 2009; Blocken 2018).

An often studied WE geometry is that with a square cross-section and four subducts (Elmualim and Awbi 2002; Elmualim 2006a,b; Li and Mak 2007; Hughes and Ghani 2009, 2010; Liu et al. 2011; Cruz-Salas et al. 2014; Khodakarami and Aboseba 2015; Castillo et al. 2017), and most of the existing commercial WE have this geometry (Hughes et al. 2012). A similar WE with circular cross-section and four subducts has also been studied (Elmualim and Awbi 2002; Su et al. 2008; Montazeri 2011). In addition, square or rectangular cross-section WEs with one duct (one-sided) (Montazeri and Azizian 2008; Montazeri et al. 2010; Esfeh et al. 2012; Cruz-Salas et al. 2014; Calautit et al. 2015; Castillo

et al. 2017) and two subducts (two-sided) (Montazeri et al. 2010; Cruz-Salas et al. 2014; Castillo et al. 2017) have been analyzed, as well as circular cross-section WEs with two, three, six and twelve subducts (Montazeri 2011). When comparing a square or rectangular cross-section WE, in the same conditions with a circular cross-section WE, having the same subducts number, it was found that the square or the rectangular one provides a higher airflow rate than that with a circular cross-section (Elmualim and Awbi 2002; Montazeri 2011).

To the best of our knowledge, in all CFD simulations to evaluate WEs, the WE was placed on the roof of an isolated, generic building consisting of a single room. In these studies, the WE was placed in the center of the roof plane, except in one study (Khodakarami and Aboseba 2015), where the WE was placed in one edge of the room roof. These CFD simulations can be classified into four types depending on the building/room and windows (Fig. 1): (1) building/room not considered (the connection of WE subducts to the building/room is considered as a pressure outlet or inlet boundary condition) (Li and Mak 2007; Su et al. 2008), (2) room considered with no window (Elmualim and Awbi 2002; Elmualim 2006a,b; Hughes and Ghani 2009, 2010), (3) room considered with windows as inlet or outlet pressure boundary condition (Montazeri et al. 2010; Montazeri 2011; Calautit et al. 2015), and (4) building and room immersed in a larger simulation domain representing the exterior surroundings (Su et al. 2008; Liu et al. 2011; Khodakarami and Aboseba 2015). In only one of these CFD studies, the flow distribution inside the room was analyzed (Liu et al. 2011).

Fig. 1 CFD simulation types for WE: (a) building/room is not considered, (b) room is considered with no window, (c) room is considered with window and (d) building and room are immersed in a large simulation domain

Almost all these previous CFD studies were accompanied by a validation study with experimental data. Most of them used as comparison parameters: (a) flow rate (Elmualim 2006a,b; Li and Mak 2007; Su et al. 2008; Montazeri et al. 2010; Montazeri 2011; Liu et al. 2011), (b) mean pressure coefficients (Elmualim and Awbi 2002; Elmualim 2006a,b; Montazeri et al. 2010; Liu et al. 2011), (c) mean velocity at some points inside the WE (Hughes and Ghani 2009, 2010; Liu et al. 2011), and (d) mean velocity at some points inside the room (Calautit et al. 2015).

In summary, previous CFD research on WEs mainly focused on the increase of the ventilation flow rate. The same holds for previous experimental research on WEs. Concerning CFD and experimental studies combined, only four studies on WEs address the distribution of the airflow inside the room, which is important when natural ventilation is used for thermal comfort. These studies consider only one direction of the wind incident to an isolated room. Three of these are experimental studies with a limited number of cases, for a room with a windward window (Cruz-Salas et al. 2014), for a room with a leeward window (Castillo et al. 2017) and for both window orientations, analyzing the effect of WE duct cross-section area and geometry (Cruz-Salas et al. 2018). The other one is a CFD study where the airflow inside the room is analyzed for one WE configuration, considering four cases: no windows and no thermal buoyancy; no windows and thermal buoyancy; windward window and thermal buoyancy; and leeward window and thermal buoyancy (Liu et al. 2011). Nevertheless, in that study, the validation is made by comparing measurements made at the WE-Room interconnection and inside the WE, but not inside the room. Thus, to the best of our knowledge, no study so far has focused on the CFD validation of the airflow distribution produced in a room by a WE and a window using measurements in the interior of the room. The goal of this paper therefore is to realize such a CFD validation study using velocity measurements obtained in the interior of the room. In addition, the goal is to perform a sensitivity and performance analysis of three WE configurations.

In this paper, CFD simulations of natural ventilation flow in a generic isolated building with a square cross-section WE and a window in the windward facade are presented. The focus is on wind perpendicular to the windward building facade. The simulations are validated with open water channel experiments by Cruz-Salas et al. (2014). The validation target parameters are the mean velocity inside the room along two lines of interest and the percentage area in the vertical center plane in the living zone with air speed equal to or above 0.10. Additionally, the impact of some of the computational parameters is investigated, including the height of the computational domain, the boundary condition for the (experimental) air-water interface, the resolution of the

computational grid and the turbulence model. The comparison of the performance of the three WEs is performed in terms of mean velocity and mean pressure coefficient in the vertical centerplane and in terms of the ventilation volume flow rate and the volume percentage of the living volume with air speed ratio equal to or above 0.10.

The paper is organized as follows. Section 2 briefly describes the open water channel experiments by Cruz-Salas et al. (2014). Section 3 presents the computational settings and parameters for the reference case. Section 4 contains the validation of the CFD simulations for three windexchangers. Section 5 presents the sensitivity analysis for some of the computational parameters of the reference case. The results of the performance analysis of the three windexchangers evaluation are presented in Section 6 and the main discussions and conclusions are presented in Section 7.

2 Open water channel experiments

Stereo particle image velocimetry (SPIV) measurements of wind-driven (isothermal) natural ventilation for a generic isolated building, consisting of a single room, with a window in the windward facade and different WEs with square cross-section, were conducted by Cruz-Salas et al. (2014) in the test section of an open water channel located at the Renewable Energy Institute of the National Autonomous University of Mexico, Morelos, Mexico. The open water channel (OWC) is 6 m long and has a test section of 1.0 m $\times$ 0.315 m $\times$ 0.41 m. The room models were scaled 1:25. In the following, dimensions are given for the reduced-scale model and in parenthesis for full scale. The models were made of transparent acrylic, with thicknesses of 6 mm (0.1500 m) for walls and room roof, 9 mm (0.2250 m) for the floor, 3 mm (0.0750 m) for the WE roof, and 1.5 mm (0.0375 m) for the WE interior partitions. The room had interior dimensions $W \times D \times H = 12$ (3.0) $\times$ 12 (3.0) $\times$ 10.8 (2.7) cm³ (m³) (Fig. 2). The room had a square window, with 5.2 cm (1.300 m) edge length and centered in the plane of the windward wall, resulting in a wall porosity (opening area divided by wall area) of 17% (Etheridge 2012). Its base was at a height of 3.6 cm (0.900 m) from the floor. All WEs had a height of 5.5 cm (1.375 m) measured from the roof, and had an interior square cross section of 2.6 cm (0.650 m) edge length. The WEs were designed with a roof eave of 2.0 cm (0.500 m) as solar and rain protection.

In Cruz-Salas et al. (2014), six WE configurations and three orientations to the incident flow were analyzed experimentally. In this paper, we focus on three of those WE configurations for wind flow perpendicular to the windward facade: (A) the WE with four openings and four subducts, (B) the WE with four openings and one duct, and (C) the WE with two openings and two subducts (Fig. 3).

Fig. 2 Model of the room with one of the windexchangers. (a) Top and front view, units in centimeter; (b) isometric view with the measurement plane and flow direction, where $h = 0.123$ m and $w = 0.132$ m are the external height and width of the room model, respectively

Fig. 3 Windexchanger configurations: (A) with four openings, four partitions, and four subducts, (B) with four openings and one duct, (C) with two openings, two partitions, and two subducts. Top view in the upper part, the arrows indicate the wind incidence angle tested, and isometric view in the lower part

The SPIV measurements were performed in the vertical central plane, as shown in Fig. 2(b). An atmospheric boundary layer (open-terrain roughness profile) was reproduced by placing vertical plates at the entrance of the water channel. The mean velocity U and turbulence intensity I profiles, shown in Fig. 4, were measured in the test section at the

Fig. 4 Vertical profiles of incident mean velocity U and longitudinal turbulence intensity I measured in the water channel (Cruz-Salas et al. 2014). The blue line is the logarithmic fit to the mean velocity data represented by black dots. The turbulent intensity data are represented by the red circles. The dashed line indicates the height of the scaled room ($h = 12.3$ cm)

room position but without the room present, these are called the incident profiles (Blocken et al. 2008). The power law exponent is $\alpha = 0.29$ (Bañuelos-Ruedas et al. 2010) and the aerodynamic roughness length y_0 is 0.06 cm (0.015 m) (Wieringa 1992). Blocken et al. (2008) recommend targeting the reproduction of the incident profiles rather than the approach profiles for a reliable validation study. A reference mean water speed $U_{ref} = 0.089$ m/s and a reference turbulence intensity of 20% were measured at the reference height y_{ref} taken as the external height of the building (WE excluded) $h = 12.3$ cm (3.075 m). The building Reynolds number $Re = U_{ref}h/\nu = 1.4 \times 10^4$, where ν is the kinematic viscosity of the fluid, was used to conserve the dynamic similarity in the experiments ($U_{ref} = 0.089$ m/s with water at $h = 12.3$ cm) with a single building at full scale ($U_{ref} = 0.062$ m/s with air at $h = 3.075$ m).

3 CFD simulations: settings and parameters

In this section, the computational settings and parameters for the three studied WE configurations are presented. The simulations are performed with the commercial CFD code ANSYS Fluent 15 (ANSYS 2013). They are performed at reduced scale with water as a fluid medium.

3.1 Computational domain and grid

The computational domain and grid are constructed to reproduce the open water channel test section. The dimensions of the computational domain are partly based on the CFD simulation guidelines by Franke et al. (2007) and Tominaga et al. (2008). However, the upstream length is reduced to 3 times of h to restrict the development of unintended streamwise gradients (Blocken et al. 2007b, 2008; Blocken 2015). The width and height domain are adjusted to represent the OWC width and the water depth, respectively. It is noted that this represents a strong deviation

from the guidelines by Franke et al. (2007) and Tominaga et al. (2008), but it should be noted that this deviation is required for the purpose of the validation study (given the narrow width of OWC) and that the actual performance evaluation later in this paper will strictly adhere to the guidelines (Franke et al. 2007; Tominaga et al. 2008; Blocken 2015). The resulting domain dimensions are $W_{ts} \times D_{ts} \times H_{ts} = 31.5 \text{ (7.875)} \times 234.6 \text{ (58.65)} \times 41.0 \text{ cm}^3 \text{ (10.25 m}^3\text{)}$ (Fig. 5(a)). As recommended by van Hooff and Blocken (2010a), the computational grid is block-structured and it is created using the surface-grid extrusion technique. The advantage of this technique is that it allows the full control of the size and shape of every grid cell. It has been successfully used in previous studies of natural ventilation (van Hooff and Blocken 2010a,b; Blocken et al. 2011; Ramponi and Blocken 2012; Perén et al. 2015a). Thus, the grids used in this study do not contain any pyramidal or tetrahedral cells (Fig. 5(b)). The grid selected from the sensitivity analysis is grid B with 1,837,108 cells, shown in Fig. 12(b). The grid-sensitivity analysis will be reported later in this paper.

3.2 Boundary conditions

The inlet boundary conditions are set in accordance to the incident profiles measured in the test section of the water channel. The mean velocity inlet boundary condition is given by the logarithmic law:

$$U(y) = \frac{u_{ABL}^*}{\kappa} \ln\left(\frac{y + y_0}{y_0}\right) \quad (1)$$

where $y_0 = 0.06 \text{ cm}$ is taken from Cruz-Salas et al. (2014), $\kappa = 0.42$ is the von Karman constant and $u_{ABL}^* = 0.007 \text{ m/s}$ is the

atmospheric boundary layer (ABL) friction velocity that is determined based on the values of U_{ref} and y_{ref} from the experiment. The inlet turbulent kinetic energy (TKE) k profile is calculated from the standard deviations of the velocity in x -direction σ_u , y -direction σ_v , and z -direction σ_w reported by Cruz-Salas et al. (2014) and following Eq. (2). The resulting turbulent kinetic energy profile is shown in Fig. 6.

$$k(y) = \frac{\sigma_u^2(y) + \sigma_v^2(y) + \sigma_w^2(y)}{2} \quad (2)$$

The turbulence dissipation rate (TDR) and specific dissipation rate (SDR) profiles, ε and ω , respectively, are calculated as follows:

$$\varepsilon(y) = \frac{u_{ABL}^{*3}}{\kappa(y + y_0)} \quad (3)$$

$$\omega(y) = \frac{\varepsilon(y)}{C_\mu k(y)} \quad (4)$$

where $C_\mu = 0.09$ (Tominaga et al. 2008). Since measurements were only performed in the vertical center plane, the following procedure is carried out to obtain the target 3D numerical incident profiles of mean velocity, TKE, TDR and SDR, to be imposed as inlet conditions at the inlet face of the computational domain. Initially, in an empty OWC domain (without the building model), a 2D velocity profile is imposed as inlet boundary condition. This profile is modified iteratively until the incident velocity profile in the vertical center plane matches the experimental one.

The standard wall functions (Launder and Spalding 1974) are used for the ground and room surfaces using the sand-grain roughness modification (Cebeci and Bradshaw 1977). For the ground surface, the value of the sand-grain roughness height is considered as $k_s = 0.002 \text{ m}$ due to the height of the first vertical control volume (Blocken et al. 2007b). The roughness constant C_s is calculated using the expression derived by Blocken et al. (2007b) for this wall function and roughness modification in ANSYS Fluent:

$$C_s = \frac{9.793 y_0}{k_s} \quad (5)$$

The room and WE surfaces are set as smooth no-slip walls applying $k_s = 0 \text{ m}$ and $C_s = 0.5$. The lateral boundaries are specified as smooth no-slip walls. At the outlet boundary, zero static gauge pressure is imposed and the top of the domain is modeled as a free-slip condition, i.e. zero normal velocity and zero normal gradients (Gallardo et al. 2013). Unintended streamwise gradients, i.e. differences between the inlet and the incident profiles are assessed from the corresponding vertical profiles in the computational domain

Fig. 5 Perspective views of: (a) computational domain and room scale model; (b) computational grid for reference case with 1,837,108 cells

Fig. 6 Profiles of the mean velocity U , the turbulent kinetic energy k and the specific dissipation rate ω at the inlet (solid line) and in the empty domain at the position where the building will be placed (incident; dashed line). The subscripts “IN” and “BP” refer to “inlet” and “building position”, respectively. The height of the building model ($h = 0.123$ m) is indicated in the figure

without the model following the recommendations by Blocken et al. (2007a,b). Figure 6 compares the inlet and the incident profiles. The average differences between the inlet and the incident profiles of U and ω are below 3%. However, the k profiles have a maximum difference of 40% close to the floor. This large difference in k profiles has been reported in other building simulations of natural ventilation (Blocken et al. 2007b; Ramponi and Blocken 2012; Blocken 2015).

3.3 Solver settings

For the reference case, the simulations are performed by combining the 3D steady RANS equations with the shear-stress transport (SST) k - ω model (Menter 1994). Second order discretization schemes are used for the convective and viscous terms of the governing equations and for the equations of the turbulence model. Pressure interpolation is second order, and the SIMPLEC algorithm is used for pressure-velocity coupling (Ramponi and Blocken 2012). Convergence is assumed to be obtained when all scaled residuals leveled off and are below 10^{-5} for x , y and z momentum, 10^{-4} for k and ω and 10^{-3} for continuity. The scaled residuals as a function of the number of iterations show oscillatory convergence, as shown in Fig. 7. Ramponi and Blocken (2012) explain that these small oscillations in these scaled residuals, that represent an average over all cells in the computational domain, could reflect stronger local oscillations in the flow field. Figure 8 presents the presents the x -component (a) and the y -component (b) of the mean velocity normalized with U_{ref} , U/U_{ref} and V/U_{ref} , respectively, for seven representative points in the computational domain. As can be observed, in this particular study, there are no

Fig. 7 Scaled residuals monitored over 10,000 iterations for the reference case

oscillations in these velocity components at these seven positions, and therefore averaging over iterations is deemed unnecessary and is not performed.

4 Validation of the CFD simulations

The CFD simulation results of the three WE configurations are compared with the SPIV measurements obtained by Cruz-Salas et al. (2014).

Figure 9 compares the velocity vector fields in the vertical centerplane. The CFD simulations reproduce the main vortices in the flow, such as the standing vortex upstream of the windward wall, the recirculation vortex on the roof upstream of the WE base, and the large recirculation in the rear of the room. The vortex in the room rear is larger in the CFD simulation than in the corresponding experiment, for each of the three configurations. In literature, it is reported that steady RANS CFD is deficient in reproducing the wind flow downstream to the building (van Hooff and Blocken 2010; Murakami 1993).

The percentage area with air speed ratio equal to or above 0.10, P , is defined as the percentage area of the living zone in the vertical center plane (up to a height of $L_z = 1.80$ m in full scale) that has an air speed ratio larger or equal to 0.10 (taken from Cruz-Salas et al. (2014)). The velocity vectors in this part of the area are shown with thicker red arrows. The differences in percentage area with air speed ratio above 0.10 between SPIV and CFD simulations of P are 8%, 14% and 5% for A, B and C WE configurations, respectively.

In Fig. 10, U/U_{ref} and V/U_{ref} along a horizontal central line L_h and a vertical central line L_v , respectively, are presented. The comparison of these variables, between numerical and experimental results, has an average error below 7% which indicates good match. The experimental results around the window opening are not considered because SPIV measurements cannot provide accurate information here

Fig. 8 Components of the normalized mean velocity at seven monitoring points over 1000 iterations: (a) x -component U/U_{ref} in three selected points along the window centerline at upstream ($x/D = -0.5$) of the inlet opening (Pt. 1), in the center of the window ($x/D = 0$) (Pt. 2) and in the center of the room ($x/D = 0.5$) (Pt. 3), and in two points along the WE centerline upstream ($x/D = 0.25$) of the WE opening (Pt. 4) and in the center of the WE duct ($x/D = 0$) (Pt. 5); (b) y -component V/U_{ref} in one representative point at the center of the room (Pt. 3) and in two points at the center of the interconnection room-WE ($x/D = 0.45, 0.55$) (Pt. 6-7)

due to reflections and shading effects. This problem has also been encountered in other PIV measurements (Karava et al. 2011).

Fig. 9 Experimental (SPIV) and numerical (CFD) velocity vector fields for three WE configurations. WE configuration (left column); experimental vector field (Cruz-Salas et al. 2014) (central column) and numerical vector field (right column). The percentage area with air speed ratio equal to or above 0.09 (taken from Cruz-Salas et al. (2014)) is shown using thicker-red arrows. The horizontal-blue line at 0.072 m above the floor indicates the upper bound of the living zone (1.80 m in full scale)

Fig. 10 Comparison of experimental (SPIV) and numerical (CFD) results for reference case in terms of components of the normalized velocity. WE configuration (left column), x -component U/U_{ref} along a horizontal centerline L_h (center column) and y -component V/U_{ref} along a vertical centerline L_v (right column)

5 Sensitivity analysis

A systematic sensitivity analysis is carried out for the WE A configuration. A given set of parameters is used as a reference case and the impact of the variation of one parameter at a time on the simulation results is evaluated. The evaluated parameters are: the height of the computational domain (Section 5.1), the resolution of the computational grid

(Section 5.2) and the turbulence model (Section 5.3). Table 1 shows the computational parameters for the sensitivity analysis. The reference case is indicated in bold.

5.1 Impact of height of computational domain

It is clear that the OWC width is too small compared to the size of the building model. In this subsection, it is analyzed whether changing the height and top of the top boundary condition for the (experimental) air-water interface has an impact on the natural ventilation CFD results. The position and type of the top boundary condition for the (experimental) air-water interface are varied by means of changing the height of the computational domain and imposing either free-slip or symmetry conditions. The computational domain defines the numerical blockage ratio BR , i.e. the ratio between the frontal area of the model facade and the cross-section of the computational domain (Barlow et al. 1999). For the reference case, the high value of $BR = 14\%$ is imposed by the required similarity of the numerical domain to that in the experiments, thus, the computational height domain $d = H_{ts}$ represents the water height measured in the experiments. In CFD, the water-air interface is simulated with a free-slip boundary condition. Three additional values of BR are considered: 7%, 4% and 2% with $d = 6 \times h$, $d = 10 \times h$ and $d = 15 \times h$, respectively. For these three BR values, the water-air interface is simulated with a symmetry boundary condition suggested by Franke et al. (2007) and Tominaga et al. (2008) (Fig. 11(a)). The upstream and downstream length of the domain remain fixed at $3 \times h$ and $15 \times h$, respectively (Fig. 5(a)). Figures 11(b) and 11(c) show that changing the top boundary condition has no impact on these results. Although the BR of 14% is clearly too large, it is retained for the validation and the sensitivity analysis—but not for the WE performance evaluation—because it represents the actual condition of the OWC experiments.

Table 1 Computational parameters for the sensitivity analysis. The reference case is indicated in bold

	Height domain (Section 5.1)	Grid resolution (Section 5.2)	Turbulence models (Section 5.3)
Ref. case	$d = H_{ts}$	1,837,108 cells (Grid B)	SST $k-\omega$ (Menter 1994)
	$d = 6 \times h$	823,192 cells (Grid A)	Sk- ϵ (Jones and Launder 1972)
	$d = 10 \times h$	3,263,478 cells (Grid C)	Rk- ϵ (Shih et al. 1995)
	$d = 15 \times h$	6,785,516 cells (Grid D)	RNG $k-\epsilon$ (Yakhot et al. 1992) Sk- ω (Yakhot et al. 1992) RSM (Launder et al. 1975)

Fig. 11 Sensitivity analysis: impact of (a) four different computational domain heights on (b) the dimensionless streamwise velocity component U/U_{ref} along L_h and (c) the dimensionless vertical velocity component V/U_{ref} along L_v (SST $k-\omega$ model, 1,837,108 cells)

5.2 Impact of computational grid resolution

Grid sensitivity analysis is performed to obtain a suitable grid limiting discretization errors and computational cost. The Grid A with 823,192 cells is made following the CFD simulation guidelines by Franke et al. (2007); Tominaga et al. (2008). Three successively refined grids (Figs. 12(a) to (d)); Grid B (reference case), Grid C, and Grid D, with 1,837,108 cells, 3,263,748 cells, and 6,785,516 cells, respectively, are generated. The grids are obtained by refining Grid A with a factor $\sqrt[3]{2}$ in each Cartesian coordinate direction. For the four grids, Figs. 13(a) and (b) show the comparison of the normalized mean velocity of the x -component U/U_{ref} and of the y -component V/U_{ref} along the horizontal central line L_h and the vertical central line L_v , respectively. The results show that Grid B provides nearly grid-independent results. In addition, the grid-convergence index (GCI) by Roache (1994) is calculated. In Figs. 13(c) and (d), the GCI results for U/U_{ref} and V/U_{ref} , respectively, show that grid sensitivity is most pronounced for the indoor area. The GCI is calculated by comparing the Grids B and C, given the small differences in the results of Grid C with D. The analysis confirms that the Grid B, that is used for the reference case, is an appropriate grid.

5.3 Impact of turbulence model

The CFD simulations are performed with the 3D steady RANS simulations with a turbulence model for closure. Six

Fig. 12 Perspective views of the room scale model grids for the grid-sensitivity analysis: (a) Grid A with 823,192 cells; (b) Grid B with 1,837,108 cells (reference grid); (c) Grid C with 3,263,478 cells; (d) Grid D with 6,785,516 cells

Fig. 13 Grid-sensitivity analysis for (a) U/U_{ref} along L_h and (b) V/U_{ref} along L_v . Grid Convergence Index error band (Roache 1994) for (c) U/U_{ref} along L_h and (d) V/U_{ref} along L_v

turbulence models are tested: (1) the shear-stress transport $k-\omega$ (SST $k-\omega$) (Menter 1994), (2) the standard $k-\epsilon$ (Sk- ϵ) (Jones and Launder 1972), (3) the realizable $k-\epsilon$ (Rk- ϵ) (Shih et al. 1995), (4) the renormalization group $k-\epsilon$ (RNG $k-\epsilon$) (Yakhot et al. 1992), (5) the standard $k-\omega$ (Sk- ω) (Yakhot et al. 1992) and (6) the Reynolds Stress Model (RSM) (Launder et al. 1975). In Figs. 14(a) and (b), the impact of the six turbulence models on U/U_{ref} and on V/U_{ref} is presented.

Fig. 14 Turbulence model sensitivity analysis: SST $k-\omega$, Sk- ϵ , k- ϵ , RNG $k-\epsilon$, Sk- ω , and RSM compared with SPIV experiments. For (a) U/U_{ref} along L_h and (b) V/U_{ref} along L_v

Comparing with the experimental results, the SST $k-\omega$ model (selected) and the RNG $k-\epsilon$ model have an averaged error (i.e. averaged along the line) below 6%, while the rest of the models yield a larger value. Despite the fact that both models, SST $k-\omega$ and RNG $k-\epsilon$, are suitable to be used for the reference case, the SST $k-\omega$ model is selected. This decision is made based on the fact that accurate results are obtained using this turbulence model in both the present study and in similar previous CFD studies of natural ventilation, which used air as a fluid and were validated with experiments in atmospheric boundary layer wind tunnels (Ramponi and Blocken 2012; Perén et al. 2015a,b,c).

6 Windexchanger performance evaluation

This paper presents a first step in the performance evaluation of different wind exchangers. It is therefore limited to the situation with wind direction perpendicular to the windward building facade. All computational parameters and settings are the same as in the reference case in Section 3 are employed, except for the size of the computational domain. Note that the computational domain for the validation study (Section 4) reproduced the width and depth of the OWC. To reduce the influence of the size walls, to reduce the blockage ratio and hence to obtain a more reliable results, a new domain is created by following the guidelines by Franke et al. (2007); Tominaga et al. (2008), where the lateral and the top distances from the exterior surface of the room to the lateral and top domain boundaries, respectively, are five times the characteristic length ($5 \times h$), see Fig. 15. For these extended boundaries the symmetry boundary condition is employed. Additionally, the domain is rescaled to full scale and air is used as working fluid. The inlet vertical profiles of U , TKE, TDR, and SDR are as in Eqs. (1, 2, 3, 4, 5) with $y_0 = 0.03$ m.

The characteristics of the three WE are summarized in Table 2. Figure 16 illustrates the computational grids in full scale of the room with the three WE configurations.

To evaluate the natural ventilation promoted by the three WE configurations, the pressure coefficient C_p and the

Fig. 15 Computational domain in full scale for WE performance analysis: (a) perspective view of the building in computational domain; (b) view of the computational grid on building and ground surfaces

Table 2 Geometrical windexchanger (WE) characteristics: number of openings N_o , number of subducts N_{SD} , ratio of WE total opening area to window opening area A_{WE}/A_w and ratio of WE total opening area to WE duct cross-sectional area A_{WE}/A_d

WE	N_o	N_{SD}	A_{WE}/A_w	A_{WE}/A_d
A	4	4	1.0	4
B	4	1	1.0	4
C	2	2	0.5	2

Fig. 16 Computational grid of the building with the three WE configurations: (A) with four openings, four partitions, and four subducts, (B) with four openings and one duct, (C) with two openings, two partitions, and two subducts

normalized velocity magnitude U_m/U_{ref} in the vertical central plane are compared, as well as the ventilation volume flow rate Q and the volume percentage with air speed ratio equal to or above 0.10 P_v are determined. P_v is calculated in the living zone, which is defined as the volume in the room below 1.80 m.

The pressure coefficient C_p is obtained by:

$$C_p = \frac{P_s - P_{ref}}{\frac{1}{2} \rho U_{ref}^2} \quad (6)$$

where P_s is the local static pressure, P_{ref} the reference static pressure and $\rho = 1.1839 \text{ kg/m}^3$ the air density at 25 °C. Figure 17 presents contour plots of U_m/U_{ref} and C_p in the vertical centerplane. It can be observed that the velocity magnitude distribution inside the room, for the three WE configurations, has three areas: (1) the incoming downward oriented jet, (2) the low-velocity zones in the center of the room and in the upper left corner and (3) the remaining area

in the right upper corner. The low-velocity (2) is smaller for the A and B configurations, while the C configuration produces the biggest area. This can be attributed to the value of the outlet and inlet area ratio A_o/A_i for the each WE configuration. For the three configurations, the window behaves as flow inlet. The WE openings can act as inlet or outlet depending on the configuration. A single WE side opening area represents 0.25 of the window area (A_w) (Table 2). For WE configuration A, the windward WE opening acts as inlet while the leeward and laterals WE openings act as outlets. This gives $A_o/A_i = 0.60$. As configuration B does not have subducts, the windward WE opening helps to increase the under pressure inside WE, but does not constitute an inlet flow to the room. Hence this area is neglected in the calculation of A_o/A_i . The laterals WE openings act as outlets, thus A_o/A_i for B is 0.75. For C, the windward WE opening acts as inlet and the leeward WE opening behaves as outlet, thus $A_o/A_i = 0.20$.

These observations are confirmed by the C_p difference between the windward facade of the room and the WE leeward opening, shown in Fig. 17.

The WE configuration B has the greatest value of $Q = 0.232 \text{ m}^3/\text{s}$, while A and C have $0.226 \text{ m}^3/\text{s}$ and $0.181 \text{ m}^3/\text{s}$, respectively. The values of air changes per hour ACH are 33, 34 and 27 for WE configurations A, B, C, respectively. It seems that in terms of flow resistance and resulting volume flow rate, the WE with single duct in configuration B acts very similar to the four-ducted WE in configuration A, with the windward opening acting as “inlet” and the three other openings as “outlet”.

To evaluate P_v , the living volume is discretized in 1,000 cells. Figure 18 shows P_v for each WE configuration, where the WE configuration B with $P_v = 21\%$ has the greatest value, followed by A and C with 13% and 5%, respectively.

The ventilation volume flow rate and the volume percentage with significant air speed ratio are normalized with the corresponding WE configuration A value, $Q^* = Q / Q_A$ and $P_v^* = P_v / P_{v-A}$. In Table 3, the values of Q^* and P_v^* are presented. The WE configuration B improves Q^* and P_v^*

Fig. 17 Contours of 3D velocity magnitude (first column), mean pressure coefficient (middle column) and velocity vector field (right column) in the vertical centerplane for the three WE configurations

Fig. 18 Comparison of the volume percentage with air speed ratio equal to or above 0.10, P_v , in the living zone. The dashed-blue line indicates the upper limit of the living zone (up to a height of $L_z = 1.80$ m)

by 3% and 62%, respectively. While, the WE configuration C decreases 20% for Q^* and 62% for P_v^* . The WE configurations A and B have similar value for Q^* , however, for P_v^* there is an important difference. This is attributed to the details of the velocity vector field in the room as influenced by the WE configuration. This implies that the WE configuration in terms of openings and subducts can

Table 3 Windexchanger (WE) performance comparison. Normalized volume flow rate Q^* with respect to the volume flow rate of WE configuration A, $Q^* = Q/Q_A$ and the normalized volume percentage with air speed ratio equal to or above 0.10 P_v , with respect to the volume percentage with air speed ratio of WE configuration A, $P_v^* = P_v/P_{v,A}$

WE	Q^*	P_v^*
A	100%	100%
B	103%	162%
C	80%	38%

have an important fact on the air velocity in the living zone and that assessment and selection of WE configurations should not only be based on volume flow rate or ACH.

7 Discussion and conclusions

A review of the literature showed that previous research on wind exchangers (WEs) mainly focused on the ventilation flow rate and options to increase it. Only four studies on WEs address the distribution of the airflow inside the room, which is important when natural ventilation is used for thermal comfort. Three of these studies are experimental studies with a limited number of cases (Cruz-Salas et al. 2014, 2018; Castillo et al. 2017) and while the fourth one is

a CFD study where the airflow inside the room is analyzed for four cases of one WE configuration but the validation is made by comparing measurements inside the WE, and at WE room interconnection (Liu et al. 2011). Thus, to the best of our knowledge, no study so far has focused on the CFD validation of the airflow distribution produced in a room by a WE and a window using measurements in the interior of the room. The goal of this paper therefore was to provide a first attempt of CFD validation using velocity measurements obtained in the interior of the room and compare the performance of three WE configurations. In addition, the goal of this paper was to provide a sensitivity analysis and a performance evaluation of three WE configurations for natural ventilation.

The simulations are performed with a window at the windward facade and at normal wind incidence angle of an isolated room with geometry $W \times D \times H = 3.0 \text{ m} \times 3.0 \text{ m} \times 2.7 \text{ m}$, in full scale. The three studied WE configurations are: A with four openings and four subducts, B with four openings and one duct, and C with two openings and two subducts. The three WEs are located at the roof center and both the building and the WE roof are flat. The study was subjected to a number of limitations:

- Only a single building geometry was considered, with fixed window opening and with fixed main dimensions of the WEs.
- Only wind perpendicular to the windward facade containing the window opening was considered.
- The study only considered wind-driven ventilation, not buoyancy, and as such all simulations were performed with under isothermal conditions.
- The indoor single zone was considered to be completely empty, the impact of furniture was not considered.
- The differences between the results on the four different grids are very small. This suggests that future studies can use grids based on the coarsest grid analyzed in the present study, and that it might even be possible to use even coarser grids without substantial loss in numerical accuracy.

In spite of these limitations, this study has provided the first CFD validation study for natural ventilation by different WEs based on internal flow variables. This study has also provided a detailed sensitivity analysis providing guidelines for future studies. Finally, the present study has provided a first performance evaluation of three different WE configurations. Although future research should address the limitations of the present study, based on the present study, the following conclusions are provided:

- The validation study showed that the SST $k-\omega$ and the RNG $k-\epsilon$ turbulence models provided the best agreement with the experimental results.

- The water-air interface in the water channel experiments could be properly simulated with a free-slip boundary condition imposed on a horizontal plane at the height of the interface.
- The low-velocity area inside the single-zone room was smaller when the building has the WE configurations A or B than with configuration C.
- The ventilation flow rate was highest for WE configuration B, i.e. 3% higher than in configuration A and 23% higher than in configuration C.
- The volume percentage with air speed ratio equal to or above 0.10 was used to evaluate the air speed distribution in the living zone. This volume percentage is highest for configuration B (21%) followed by configuration A (13%) and configuration C (5%).
- The better performance of configurations A and B in terms of ventilation flow rate or ACH was attributed to the similar overall air flow resistance in these two configurations, which was lower than in configuration C.
- The clearly better performance of configuration B in terms of magnitude of indoor air velocity ratio was attributed to the specific details of the velocity vector field in the room as influenced by the WE configuration.
- This implies that the WE configuration in terms of openings and subducts can have an important fact on the air velocity in the living zone and that assessment and selection of WE configurations should not only be based on volume flow rate or ACH.

Acknowledgements

The authors thank the Eindhoven University of Technology for the software, hardware and office facilities during an academic stay of J.A. Castillo. This work has been partially supported by the PAPIIT-UNAM IN103816 project. J.A. Castillo acknowledges the postdoctoral fellowship grant by the DGAPA-UNAM, the computational facilities provided by the SENER-CONACYT project 260155. J.A. Castillo also is grateful with Dr. Adriana Lira and Dr. Gerardo Oliva for their academic support during the postdoctoral stay. Twan van Hooff is currently a postdoctoral fellow of the Research Foundation Flanders (FWO) and acknowledges its financial support (project FWO 12R9718N). The authors acknowledge the partnership with ANSYS CFD.

References

- ANSYS (2013). Fluent 15 user's guide. Lebanon: Fluent Inc.
- Baker CJ (2007). Wind engineering—Past, present and future. *Journal of Wind Engineering and Industrial Aerodynamics*, 95: 843–870.
- Bangalee MZI, Miao JJ, Lin SY, Yang JH (2013). Flow visualization, PIV measurement and CFD calculation for fluid-driven natural

- cross-ventilation in a scale model. *Energy and Buildings*, 66: 306–314.
- Bañuelos-Ruedas F, Angeles-Camacho C, Rios-Marcuello S (2010). Analysis and validation of the methodology used in the extrapolation of wind speed data at different heights. *Renewable and Sustainable Energy Reviews*, 14: 2383–2391.
- Barlow JB, Rae WH, Pope A (1999). *Low-Speed Wind Tunnel Testing*, 3rd edn. New York: John Wiley & Sons.
- Blocken B, Carmeliet J, Stathopoulos T (2007a). CFD evaluation of wind speed conditions in passages between parallel buildings: Effect of wall-function roughness modifications for the atmospheric boundary layer flow. *Journal of Wind Engineering and Industrial Aerodynamics*, 95: 941–962.
- Blocken B, Stathopoulos T, Carmeliet J (2007b). CFD simulation of the atmospheric boundary layer: Wall function problems. *Atmospheric Environment*, 41: 238–252.
- Blocken B, Stathopoulos T, Carmeliet J (2008). Wind environmental conditions in passages between two long narrow perpendicular buildings. *Journal of Aerospace Engineering*, 21: 280–287.
- Blocken B, van Hooff T, Aanen L, Bronsema B (2011). Computational analysis of the performance of a venturi-shaped roof for natural ventilation: Venturi-effect versus wind-blocking effect. *Computers & Fluids*, 48: 202–213.
- Blocken B (2014). 50 years of Computational Wind Engineering: Past, present and future. *Journal of Wind Engineering and Industrial Aerodynamics*, 129: 69–102.
- Blocken B (2015). Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. *Building and Environment*, 91: 219–245.
- Blocken B (2018). LES over RANS in building simulation for outdoor and indoor applications: A foregone conclusion? *Building Simulation*, 11: 821–870.
- Calautit JK, Hughes BR, Shahzad SS (2015). CFD and wind tunnel study of the performance of a uni-directional wind catcher with heat transfer devices. *Renewable Energy*, 83: 85–99.
- Carrilho da Graça G, Daish NC, Linden PF (2015). A two-zone model for natural cross-ventilation. *Building and Environment*, 89: 72–85.
- Castillo JA, Cruz-Salas MV, Huelsz G (2017). Natural ventilation by windexchangers in a building with a window in prevailing winds: Design guidelines. *International Journal of Ventilation*, 16: 1–14.
- Cebeci T, Bradshaw P (1977). *Momentum Transfer in Boundary Layers*. Washington, D.C.: Hemisphere Publishing Corporation.
- Chen Q (2009). Ventilation performance prediction for buildings: A method overview and recent applications. *Building and Environment*, 44: 848–858.
- CONAVI (2010). Comisión Nacional de Vivienda, Código de Edificación de vivienda.
- Cruz-Salas MV, Castillo JA, Huelsz G (2014). Experimental study on natural ventilation of a room with a windward window and different windexchangers. *Energy and Buildings*, 84: 458–465.
- Cruz-Salas MV, Castillo JA, Huelsz G (2018). Effect of windexchanger duct cross-section area and geometry on the room airflow distribution. *Journal of Wind Engineering and Industrial Aerodynamics*, 179: 514–523.
- Elmualim AA (2006a). Effect of damper and heat source on wind catcher natural ventilation performance. *Energy and Buildings*, 38: 939–948.
- Elmualim AA (2006b). Dynamic modelling of a wind catcher/tower turret for natural ventilation. *Building Services Engineering Research and Technology*, 27: 165–182.
- Elmualim AA, Awbi HB (2002). Wind tunnel and CFD investigation of the performance of “windcatcher” ventilation systems. *International Journal of Ventilation*, 1: 53–64.
- Esfeh MK, Dehghan AA, Manshadi MD, Mohagheghian S (2012). Visualized flow structure around and inside of one-sided wind-catchers. *Energy and Buildings*, 55: 545–552.
- Etheridge D (2012). *Natural Ventilation of Buildings—Theory, Measurement and Design*. Chichester, UK: John Wiley & Sons.
- Franke J, Hellsten A, Schlünzen H, Carissimo B (2007). *Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment*. COST office.
- Gallardo JP, Pettersen B, Andersson HI (2013). Effects of free-slip boundary conditions on the flow around a curved circular cylinder. *Computers & Fluids*, 86: 389–394.
- Hughes BR, Ghani SAAA (2009). A numerical investigation into the effect of windvent dampers on operating conditions. *Building and Environment*, 44: 237–248.
- Hughes BR, Ghani SAAA (2010). A numerical investigation into the effect of windvent louvre external angle on passive stack ventilation performance. *Building and Environment*, 45: 1025–1036.
- Hughes BR, Calautit JK, Ghani SA (2012). The development of commercial wind towers for natural ventilation: A review. *Applied Energy*, 92: 606–627.
- Jones WP, Launder BE (1972). The prediction of laminarization with a two-equation model of turbulence. *International Journal of Heat and Mass Transfer*, 15: 301–314.
- Karava P, Stathopoulos T, Athienitis AK (2011). Airflow assessment in cross-ventilated buildings with operable facade elements. *Building and Environment*, 46: 266–279.
- Khodakarami J, Aboseba MR (2015). Impact of openings’ number and outdoor flow direction on the indoor vertical flow velocity in wind catchers. *International Journal of Renewable Energy Research*, 5: 325–333.
- Launder BE, Spalding DB (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, 3: 269–289.
- Launder BE, Reece GJ, Rodi W (1975). Progress in the development of a Reynolds-stress turbulence closure. *Journal of Fluid Mechanics*, 68: 537–566.
- Li L, Mak CM (2007). The assessment of the performance of a windcatcher system using computational fluid dynamics. *Building and Environment*, 42: 1135–1141.
- Liu S, Mak CM, Niu J (2011). Numerical evaluation of louver configuration and ventilation strategies for the windcatcher system. *Building and Environment*, 46: 1600–1616.
- Menter FR (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32: 1598–1605.
- Meroney R, Derickson R (2014). Virtual reality in wind engineering: The windy world within the computer. *Indian Journal Wind Engineering—Indian Society for Wind Engineering*, 11(2): 11–26.

- Montazeri H (2011). Experimental and numerical study on natural ventilation performance of various multi-opening wind catchers. *Building and Environment*, 46: 370–378.
- Montazeri H, Azizian R (2008). Experimental study on natural ventilation performance of one-sided wind catcher. *Building and Environment*, 43: 2193–2202.
- Montazeri H, Montazeri F, Azizian R, Mostafavi S (2010). Two-sided wind catcher performance evaluation using experimental, numerical and analytical modeling. *Renewable Energy*, 35: 1424–1435.
- Murakami S (1993). Comparison of various turbulence models applied to a bluff body. *Journal of Wind Engineering and Industrial Aerodynamics*, 46–47: 21–36.
- Murakami S (1997). Current status and future trends in computational wind engineering. *Journal of Wind Engineering and Industrial Aerodynamics*, 67–68: 3–34.
- Oropeza-Perez I, Østergaard PA (2014). Energy saving potential of utilizing natural ventilation under warm conditions—A case study of Mexico. *Applied Energy*, 130: 20–32.
- Perén JI, van Hooff T, Leite BCC, Blocken B (2015a). CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: Impact of roof angle and opening location. *Building and Environment*, 85: 263–276.
- Perén JI, van Hooff T, Leite BCC, Blocken B (2015b). Impact of eaves on cross-ventilation of a generic isolated leeward sawtooth roof building: Windward eaves, leeward eaves and eaves inclination. *Building and Environment*, 92: 578–590.
- Perén JI, van Hooff T, Ramponi R, Blocken B, Leite BCC (2015c). Impact of roof geometry of an isolated leeward sawtooth roof building on cross-ventilation: Straight, concave, hybrid or convex? *Journal of Wind Engineering and Industrial Aerodynamics*, 145: 102–114.
- Ramponi R, Blocken B (2012). CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters. *Building and Environment*, 53: 34–48.
- Roache PJ (1994). Perspective: A method for uniform reporting of grid refinement studies. *Journal of Fluids Engineering*, 116: 405–413.
- Shetabivash H (2015). Investigation of opening position and shape on the natural cross ventilation. *Energy and Buildings*, 93: 1–15.
- Shih T-H, Liou WW, Shabbir A, Yang Z, Zhu J (1995). A new k - ϵ eddy viscosity model for high Reynolds number turbulent flows. *Computers & Fluids*, 24: 227–238.
- Solari G (2007). The international association for wind engineering (IAWE): Progress and prospects. *Journal of Wind Engineering and Industrial Aerodynamics*, 95: 813–842.
- Stathopoulos T (1997). Computational wind engineering: Past achievements and future challenges. *Journal of Wind Engineering and Industrial Aerodynamics*, 67–68: 509–532.
- Su Y, Riffat SB, Lin Y, Khan N (2008). Experimental and CFD study of ventilation flow rate of a Monodraught™ windcatcher. *Energy and Buildings*, 40: 1110–1116.
- Tominaga Y, Mochida A, Yoshie R, Kataoka H, Nozu T, Yoshikawa M, Shirasawa T (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 96: 1749–1761.
- Tominaga Y, Stathopoulos T (2013). CFD simulation of near-field pollutant dispersion in the urban environment: A review of current modeling techniques. *Atmospheric Environment*, 79: 716–730.
- van Hooff T, Blocken B (2010a). Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam Arena Stadium. *Environmental Modelling & Software*, 25: 51–65.
- van Hooff T, Blocken B (2010b). On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. *Computers & Fluids*, 39: 1146–1155.
- Wieringa J (1992). Updating the Davenport roughness classification. *Journal of Wind Engineering and Industrial Aerodynamics*, 41: 357–368.
- Yakhot V, Orszag SA, Thangam S, Gatski TB, Speziale CG (1992). Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A: Fluid Dynamics*, 4: 1510–1520.