Summer 2018

Numerical Studies of Turbulence Effects in Cross-Flow Ventilation

Mina Mohammadmirzaei
San Jose State University

NUMERICAL STUDIES OF TURBULENCE EFFECTS IN CROSS-FLOW VENTILATION

A Thesis

Presented to

The Faculty of the Department of Mechanical Engineering

San José State University

In Partial Fulfillment

of the Requirements for the Degree

Master of Science

by

Mina Mohammadmirzaei

August 2018

The Designated Thesis Committee Approves the Thesis Titled

NUMERICAL STUDIES OF TURBULENCE EFFECTS IN CROSS-FLOW VENTILATION

by

Mina Mohammadmirzaei

APPROVED FOR THE DEPARTMENT OF MECHANICAL ENGINEERING

SAN JOSÉ STATE UNIVERSITY

August 2018

Kathryn Gosselin, Ph.D.	Department of Mechanical Engineering
Nicole Okamoto, Ph.D.	Department of Mechanical Engineering
Ernest Thurlow, Ph.D.	Department of Mechanical Engineering

ABSTRACT

NUMERICAL STUDIES OF TURBULENCE EFFECTS IN CROSS-FLOW VENTILATION

by Mina Mohammadmirzaei

Natural ventilation systems reduce a building's utilization of energy in terms of electricity consumption and fossil fuel usage. Many factors including building shape, window style and configuration, and wind turbulence impact the efficacy of natural ventilation. In order to investigate the effect of turbulence on natural ventilation, twelve different conditions of a cross-flow ventilated room were studied numerically using computational fluid dynamics. Wind tunnel studies of tracer gases in a scale model were used to validate the numerical results. A comparison of twelve investigated cases shows that air circulation inside a natural ventilated room is only slightly affected by turbulence, and that the impact is strongest at low velocities. However, the result is small compared to the impact of wind velocity, and the relative impact of turbulence approaches zero at high velocities. Therefore, using fans to increase flow velocity results in better air replacement compared to using retrofitting geometries.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF ABBREVIATIONS

CFD - Computational Fluid Dynamics
FFD - Fast Fluid Dynamics
HVAC - Heating, Ventilation, and Air Conditioning
LES - Large Eddy Simulation
MV - Mechanical Ventilation
NV - Natural Ventilation
RANS - Reynolds Averaged Navier-Stokes
SIMPLEC - Semi-Implicit Method for Pressure Linked Equations-Constant

I. Introduction

Since the second half of the 20th century, the majority of non-domestic buildings in most developed countries use mechanical ventilation (MV) due to increasing user expectations of thermal comfort and indoor air quality standards [1]. Heating, ventilation and air conditioning (HVAC) systems account for 40%-60% of total building energy consumption [2,3]. Since residential and commercial buildings are responsible for nearly 40% of the total energy consumption in the United States [4], there is a high demand to improve efficiency. One possible approach is to replace mechanical ventilation with natural ventilation (NV). Natural ventilation reduces the building's energy consumption in terms of electricity consumption and fossil fuel usage [5-9] and decreases environmental pollution [10], which can help mitigate global warming and climate change.

However, since the introduction of central heating and cooling, most buildings have been constructed to only use mechanical ventilation, and natural ventilation has been neglected [1]. There is therefore a need to identify how existing structures might perform with natural ventilation and to identify possible retrofitting strategies to improve airflow.

II. Literature Review

A. Natural Ventilation Fundamentals

Natural ventilation may be categorized into wind-driven natural ventilation and buoyancy-driven natural ventilation, where pressure differences generated by wind or buoyancy forces act on the opening(s) of the building, respectively. Wind-driven and buoyancy-driven natural ventilation may also occur simultaneously, where wind and stack effects could reinforce or oppose each other [5]. In experimental and numerical simulations of wind-driven natural ventilation, it is common to neglect the unpredictable changes of wind velocity and direction and model the wind as an airflow with a constant mean velocity and a fixed direction [11-13]. However, it is important to note that setting the proper boundary conditions and allowing for wind fluctuations lead to more accurate results [14].

Different research approaches have been utilized to analyze natural ventilation. Taking full-scale measurements of a naturally ventilated building is the most accurate approach among all; however, it is not very flexible, since the correlation between meteorological data measurements and the air velocity is specific to the case study, and the conditions cannot be controlled. Additionally, the results cannot be applied to buildings with different layouts. [15].

Scaled model wind tunnel studies are more flexible and easier to control compared to a full-scale research approach [16]. However, some limitations exist to this approach as there is a certain degree of variation and error in measurements and environmental conditions in the test chamber that cannot be absolutely controlled [17].

Computational fluid dynamics (CFD) is the most flexible approach for investigating natural ventilation. However, the results of this approach must be validated with scaled model wind tunnel experiments [18]. A review of the literature indicates that 3D steady Reynolds averaged Navier-Stokes (RANS) and large eddy simulation (LES) are the most suitable modeling approaches for cross-ventilation, which will be examined in this work.

Jiang *et al.* [19] have studied both wind-driven and buoyancy-driven natural ventilation with the help of three CFD models including steady and unsteady RANS, and LES. The results showed that LES is more accurate and informative than the other two models, but it requires a longer computation time. In this study, Jiang *et al.* also found that since the peak turbulence energy is at higher frequencies for wind-driven natural ventilation, the fluctuating flow field is more important in determining the ventilation rate in wind-driven natural ventilation compared to buoyancy-driven natural ventilation.

Moreover, Jiru *et al.* [20] have utilized both whole domain (coupled) and domain decomposition (decoupled) approaches in CFD modeling of wind-driven natural ventilation. In the coupled approach, the internal and external airflow are analyzed simultaneously in one domain, whereas in the decoupled approach the internal and external airflow are simulated separately. The analysis results showed that the whole domain approach is more accurate, but computationally more expensive than the decomposition approach. Efficient computing algorithms and grid generation techniques were suggested to relatively reduce the computations.

Jin *et al.* [21] have validated fast fluid dynamics (FFD) with different wind-driven and buoyancy-driven natural ventilated test cases at the early design stage. The results showed that for wind-driven natural ventilation, FFD accurately predicts the velocity distribution on the upstream side of a building. However, FFD is not as accurate as CFD in simulating airflow distribution inside and on the downstream side of a building. For buoyancy-driven natural ventilation, FFD accurately predicts thermal stratification and airflow pattern.

B. Impact of Building Geometry

Over the past decade, various techniques and parameters of natural ventilation including wind towers [22], building shape [23], and window style [24] and configuration [25] have been studied. Hughes *et al.* [22] have reviewed the importance of wind towers as an application of passive cooling systems to replace high energy-consuming mechanical ventilation systems. Wind towers provide natural ventilation based on the pressure difference surrounding the building. The primary driving force for the wind tower is the external driving wind. However, in the absence of wind, the stack or buoyancy forces resulting from temperature difference act as the primary driving force for the wind tunnel.

Abohela *et al.* [23] have studied various configurations to determine the effect of building height and roof shape on natural ventilation. The results demonstrated that as the height of a building increases, the turbulent intensity of the wind flow above the building roof increases as well. Moreover, all different roof shapes including flat, domed, gabled, pyramidal, vaulted and wedged have an accelerating effect on the wind velocity with the dome and the barrel vault having the potential to produce more energy than the other roof shapes.

Wang *et al.* [24] have investigated the impact of three different types of windows including awning, hopper, and casement on natural ventilation. The results showed that the impact of different types of windows on the ventilation rate varies greatly with wind directions due to the change of flow pattern introduced by the windows and turbulence effect. However, at a window opening angle of 45 degrees, the ventilation rate for hopper windows is always greater than awning and casement windows regardless of the wind direction.

Window placement strategies in natural ventilation could be divided into single-sided ventilation and cross-flow ventilation based on the number and the relative position of the windows [10]. Schulze *et al.* [26] have analyzed steady state ventilation in three different single-sided configurations subjected to various temperature and pressure differences boundary conditions. The configurations include partially opened tilted bottom-hung window, horizontally sliding window, and a window with two openings for inflow and outflow. The results demonstrated that indoor air quality can be easily maintained for all configurations in moderate climates. However, the summer cooling potential and thermal comfort strongly depend on the configuration. It is important to note that if proper control over the openings is not achieved, the building might overcool even in summer condition.

In cross-flow ventilation, where windows are placed on adjacent or opposite walls of a structure [27], the impact of various flow and geometry conditions on ventilation has been studied extensively [28]. For example, Bangalee *et al.* [10] have studied the effect of surrounding structures as well as the number and configuration of the windows on a cross-ventilated building. The results prove experimentally and numerically that maximum flow rate occurs when the windows are located in the middle rather than in the corner of opposite walls.

In another study Bangalee *et al.* [29] have analyzed three different window configurations for a single story, cross-ventilated building with the help of CFD. The results demonstrate that in the case of face to face cross ventilation through four openings, the uniform mixing of the external and the internal air is enhanced, as the windows are placed further from each other on the wall. Furthermore, in the case of diagonal cross ventilation with two openings, the closer the windows are located towards each other, the higher the ventilation rate and the shorter the ventilation time become. In this study Bangalee also proves that the mean pressure coefficient is subjected to the configuration and the number of windows, as well as the wind angle.

On the contrary, there is little existing research on the impact of turbulence on cross-flow ventilation, which depends strongly on the surrounding built environment, i.e. the presence of nearby structures, trees, or landscape. Meyer *et al.* [30], reviewing the effect of envelope flow turbulence on occupant thermal comfort and building energy consumption in natural ventilation, found that increased turbulent intensity may improve the occupant thermal comfort in warmer environments.

Chu *et al.* [31] have demonstrated that for two different adjacent and opposite cross-ventilation configurations, discharge coefficient, unlike inlet velocity and internal pressure, is independent of external turbulent intensity. However, discharge rate, which is normally used to predict the natural ventilation performance, gives an incomplete picture of ventilation efficiency. An efficient ventilation system must be capable of replacing all interior air with exterior air, meaning that an increased throughput alone does not ensure complete ventilation [10]. For a limited number of cross-ventilation cases, Bangalee *et al.* [10] demonstrates that air replacement may be negatively impacted by turbulence in cases even when the discharge rate is not.

In order to control flow turbulence, various active and passive flow control methods have been used in a variety of applications. Vortex generators are the most common form of passive flow control devices to delay transition and control separation. These consist of geometrical features that induce local vortices in order to control the shape and intensity of local turbulence behavior and prevent flow separation. These have been used in a variety of applications, including the work of Fransson [32], who applied circular roughness elements to a flat plate in order to delay the transition to turbulence.

Similarly, Chishty *et al.* [33] examined the effect of a dimple on the suction side of a turbine blade to prevent the boundary layer separation at low turbulence intensities.

When the flow stream lacks the sufficient energy to overcome the adverse pressure gradient, the viscous dissipation along the flow path, and the energy loss caused by the modification in momentum, the separation occurs [34]. The dimple acts as a vortex generator energizing the flow and changing the incoming laminar flow into turbulent flow prior to separation. Since turbulent flow is hardly affected by pressure gradient, the flow separation is suppressed or delayed. As a result, the normalized loss coefficient and fuel cost are reduced.

III. Objectives

The primary objective of this thesis is to quantify the effect of turbulence on the efficacy of cross-flow ventilation in depth. If turbulence positively affects the air circulation, it will be determined whether various common passive vortex generation geometries can effectively control turbulence and improve ventilation. Otherwise, the relative impact of velocity versus turbulence at various locations in the room will be quantified.

IV. Methodology

A. Model Configuration and Grid Generation

A true-to-size model of a 12 ft x 12 ft x 8 ft room with two 3 ft x 3 ft windows horizontally centered on the opposite walls was developed, surrounded by a 172 ft x 92 ft x 48 ft cuboid environment. The lateral extension of the domain and the extension in inlet flow direction were 5 times the room's height, and the extension of the domain in outlet flow direction was 15 times the room's height to allow the flow to redevelop behind the wake region [35]. A quadrilateral fine mesh of 2 in element size was applied to the building, and the surrounding domain was discretized more coarsely with a quadrilateral mesh of 3 ft element size. In order to take the boundary effects into account, a mesh inflation was applied on the room as well. Figs. 1 and 2 show the geometry in ANSYS Design Modeler and the proper mesh applied on this geometry.

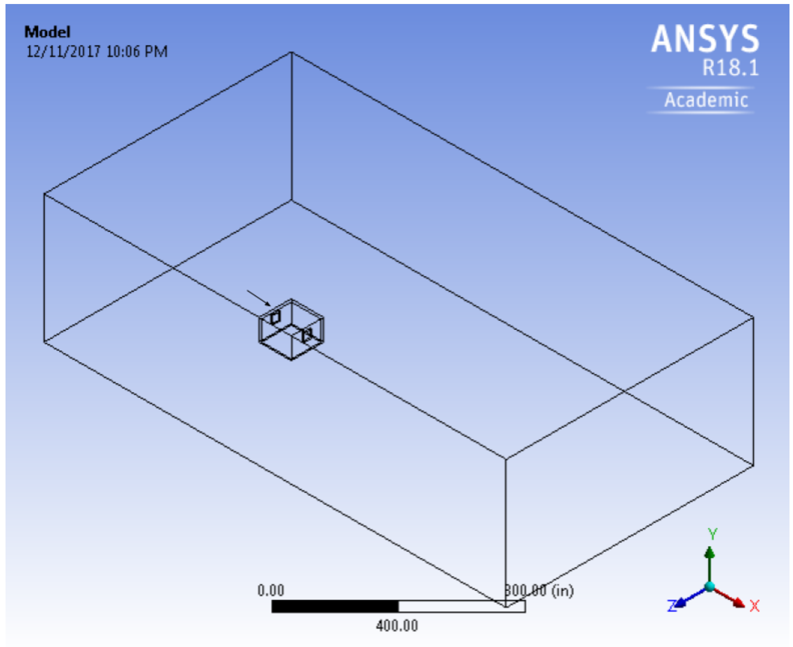


Fig. 1.   True-to-size room and its domain in ANSYS Design Modeler

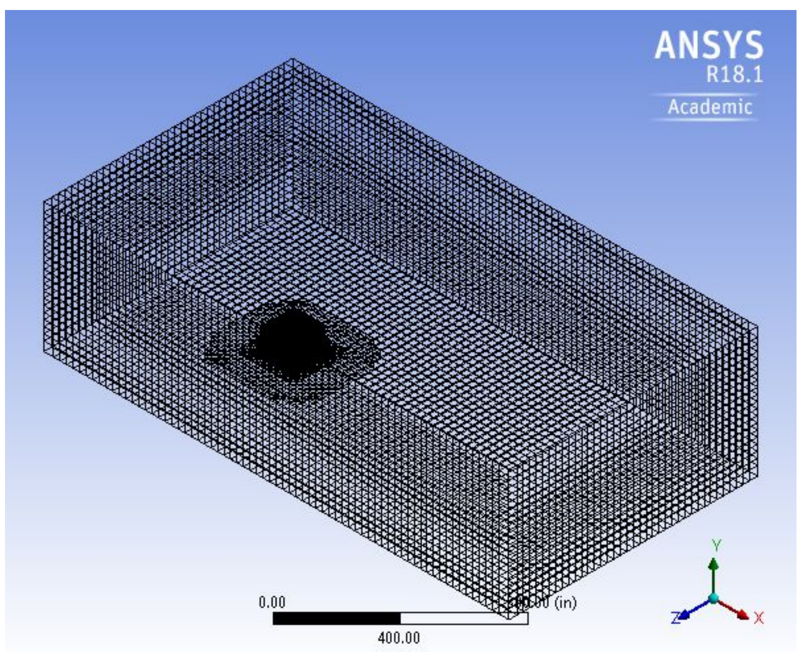


Fig. 2.   Discretization of the geometry

The number of cells reported by Fluent was 538,359, with a minimum volume of 1.923464e-05 m^3 and a maximum volume of 3.015788 m^3. The minimum orthogonal quality and the maximum aspect ratio were reported as 7.15909e-02 and 1.27617e+01 respectively.

B. Flow Characteristics and Computational Domain

The system was assumed to be incompressible, viscous, turbulent, non-buoyant, and three-dimensional. Using ANSYS Fluent solver, a k-epsilon realizable model with standard wall functions was used to simulate the airflow in the domain. Three uniform velocities of 2, 4, and 6 m/s each with four different turbulent intensities of 5, 10, 15, and 20% were established at the inlet boundary investigating both medium and high turbulence cases [36]. The outlet boundary was assumed an opening with zero relative pressure, and a symmetry boundary condition was applied on the top and the sides of the computational domain. A no-slip wall boundary condition was imposed on the remaining sides of the geometry.

C. Numerical Solution

The SIMPLEC scheme for pressure-velocity coupling with under-relaxation factors of 0.7 and 0.3 for pressure and momentum was utilized to achieve better convergence. A transient simulation with an inlet airflow of 310 K was performed for each set of boundary conditions initializing based on converged steady-state data at 300 K. Although steady-state simulations are sufficient to demonstrate velocity profiles and flow characteristics, the transient simulation allowed for a better analysis of how the air inside the room mixes with the entering fluid [10]. The minimum residual target of 10^{-6} for momentum and 10^{-4} for continuity, turbulent kinetic energy (k), and turbulent dissipation rate (ε) was achieved for all of the cases.

D. Grid Independence

To assure the grid independence, a finer mesh with 1 in and 2 ft element sizes was applied to the room and the domain, respectively. Figs. 3-6 show the velocity profiles plotted for four vertical lines at different locations inside the room for both mesh settings.

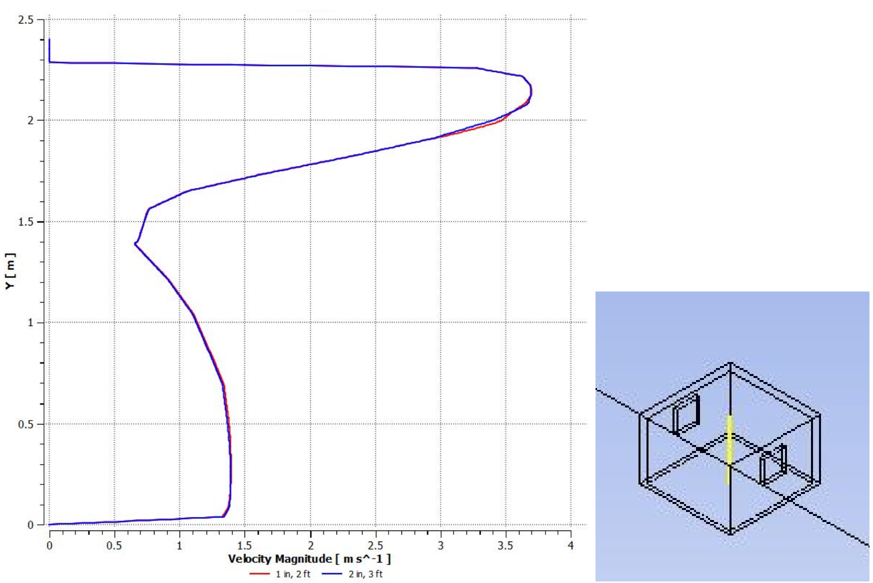


Fig. 3. Variation of velocity magnitude across a vertical line at the center of the room for original and refined meshes

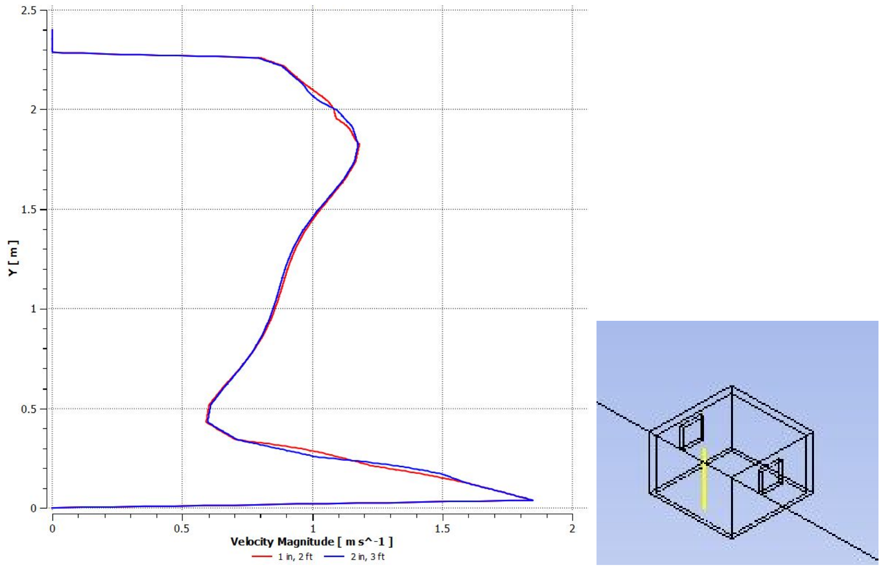


Fig. 4. Variation of velocity magnitude across a vertical line on the right side of the room for original and refined meshes

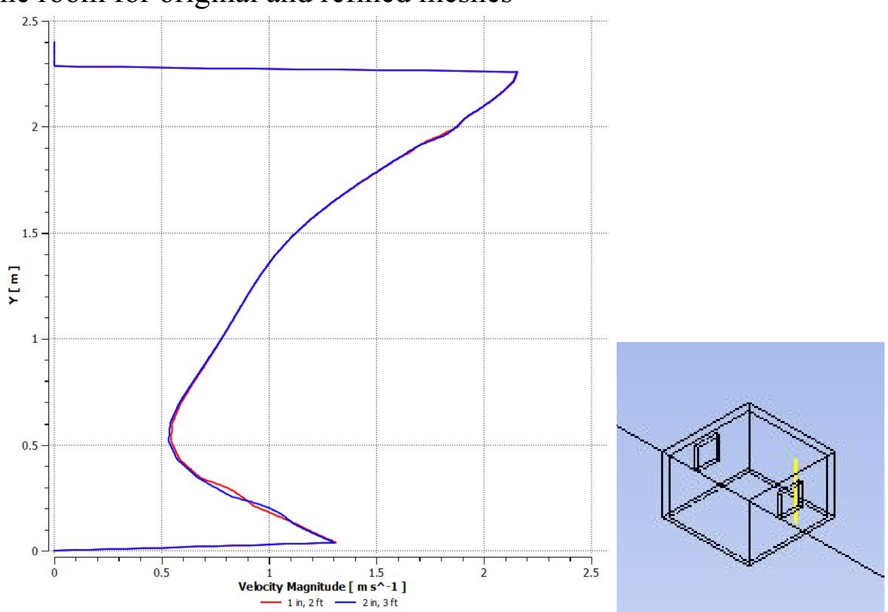


Fig. 5. Variation of velocity magnitude across a vertical line on the left side of the room for original and refined meshes

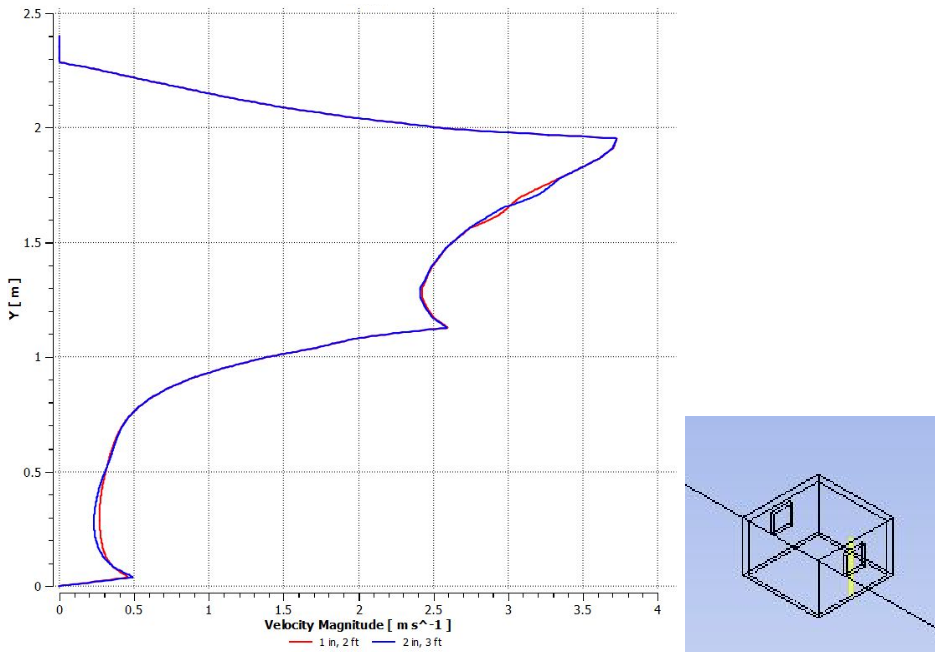


Fig. 6. Variation of velocity magnitude across a vertical line at the downstream of the room for original and refined meshes

The negligible difference between the results of the two models indicates that the initial mesh is sufficient, as the solution does not change as the mesh is refined.

The mesh was refined even further to 0.2, 0.1, and 0.08 in elements corresponding to y^+ values of 50 to 25 near the wall, but the velocity profile did not change. The enhanced wall treatment option resulted in the same velocity profile as well, indicating that the velocity resolution near the wall is sufficient. Given that the trends of the numerical results matched well with the experimental validation, which will be discussed subsequently, the sharp velocity profiles were deemed to not be concerning.

E. Solution Validation

In order to validate the computational simulation, the cross-flow ventilation was investigated for a 1:24 scaled optically transparent model made of plexiglass inside a closed-loop wind tunnel. The test section of the wind tunnel was 24 in x 24 in x 48 in with a maximum velocity of 15 m/s. The local velocities and turbulent intensities inside and outside of the room as well as the laser absorption of tracer gases for circulation time scales were measured. The experimental part of this research was performed by other students [37].

The scaled model wind tunnel simulation was not performed at the Reynolds number of the full-size model, since for scaled model Reynolds numbers above 10,000, the equality of the Reynolds number of scaled and full size models is not necessary for sharp-edged structures [38,39].

V. Results and Discussion

A. Experimental Data Validation

Fig. 7 demonstrates how the time constant changes with inlet velocity and turbulent intensity based on experimental data.

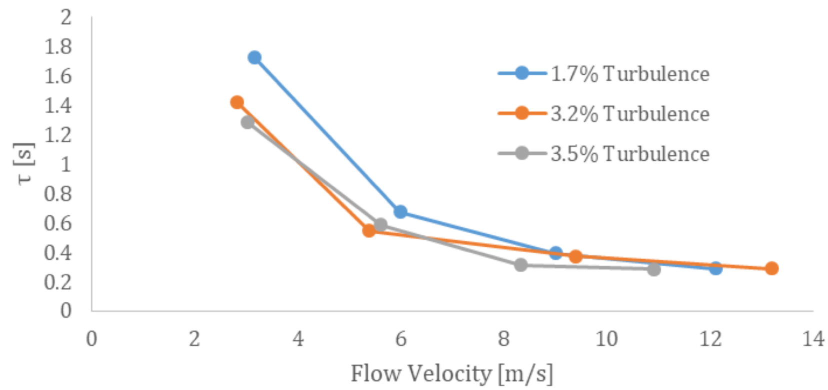


Fig. 7. Experimental results: effect of inlet velocity and turbulent intensity on the time constant adapted from "Experimental Studies of Cross-Flow Ventilation in the Presence of Turbulence" [36]

The experimental data show that turbulent intensity has a slight impact on the quality of air replacement especially at higher velocities; however, inlet velocity affects the air circulation more significantly.

B. Transient Studies of Air Replacement

For all transient cases, temperature at any given point inside the room follows an approximately first-order linear response, as shown in Fig. 8.

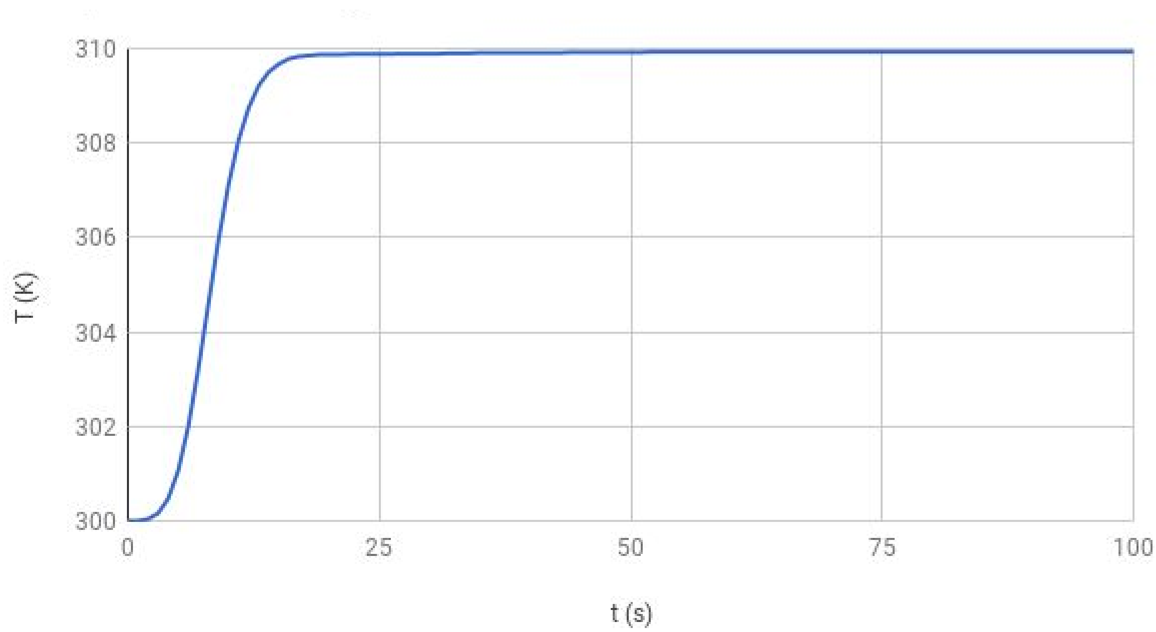


Fig. 8. Change of temperature with time at a given point inside the room

The time constant (τ) can be determined from the following equation;

$$T_{norm}(t) = \frac{T(t)-T_f}{T_i-T_f} = e^{-\frac{t}{\tau}} \quad (5.1)$$

where T_i = 300 K and T_f = 310 K.

A MATLAB least-square fit function was used to fit the equation above into the temperature data at each point inside the room with respect to time, and extract the time constant at any given point. The change in the room's temperature with respect to time determines the degree to which the air inside the room is replaced with the entering fluid. Therefore, a smaller time constant corresponds to better air circulation at that location.

For each of the 12 cases, the average time constant was calculated from 9 data points at three different x, y, and z locations inside the room. The three x, y, and z locations were selected to be in the middle and at ⅛ of each side's length further away from the corners. For each of the three given x locations, the time constant was reported as the average of 9 data points located on the yz plane at the given x location. The same calculations were performed for three y and z locations on the xz and xy planes located at given y and z locations respectively. Fig. 9 shows the location of the 9 data points on any xy, xz, or yz plane.

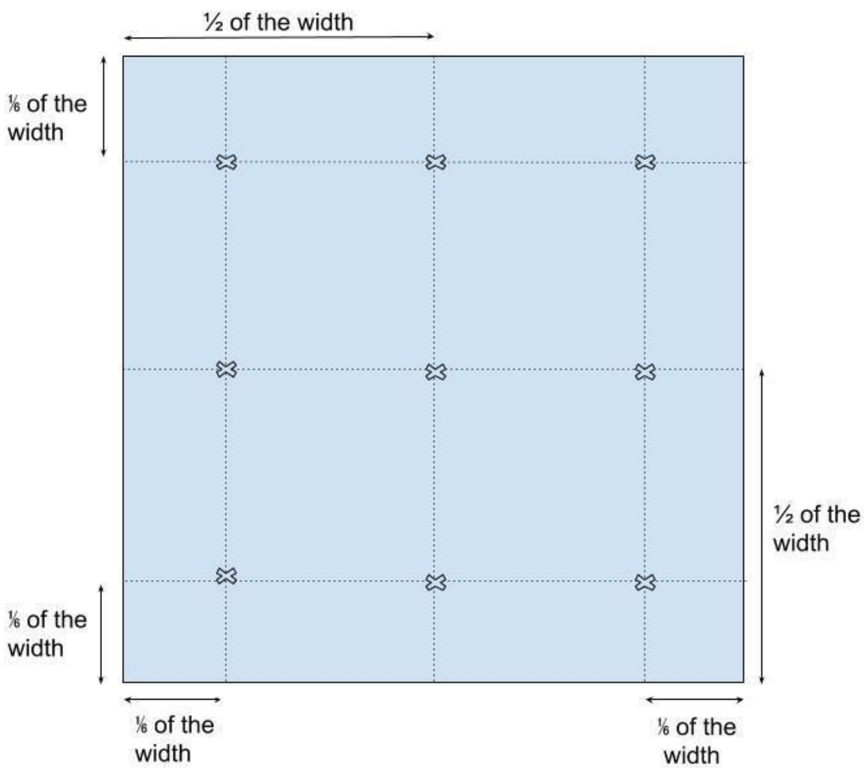


Fig. 9. Location of data points used to average the time constant at any given xy, xz, and yz plane

Figs. 10-12 demonstrate how the average time constant changes along the x, y, and z axes respectively. Fig. 10 demonstrates that at lower inlet velocities, the air replacement happens the best in the downstream side of the room, but this effect is not as visible at higher velocities. However, it is evident that increasing inlet velocity has a very significant positive impact on the time constant, resulting in better air circulation.

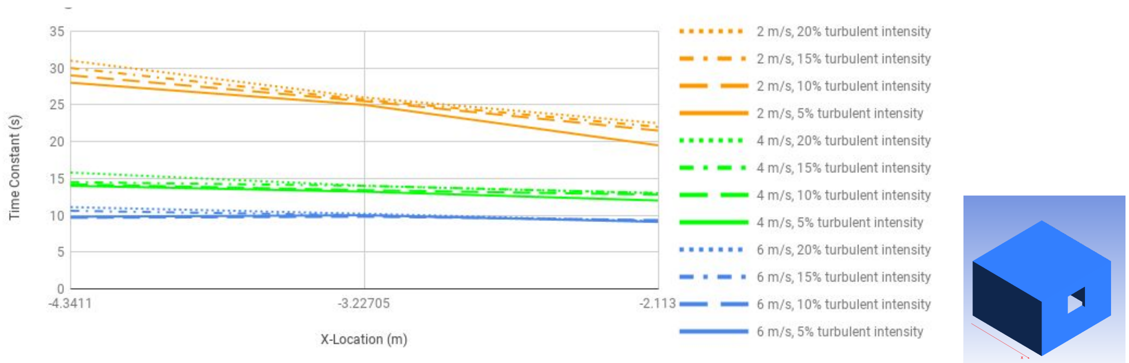


Fig. 10. Average effect of x-location on the time constant

Fig. 11 shows that at lower inlet velocities, the air replacement happens the best at towards the bottom of the room. However, this effect decreases significantly at higher velocities. Like the previous results, Fig. 5 also demonstrates that increasing inlet velocity has a very significant impact on the time constant, leading to a more complete air replacement.

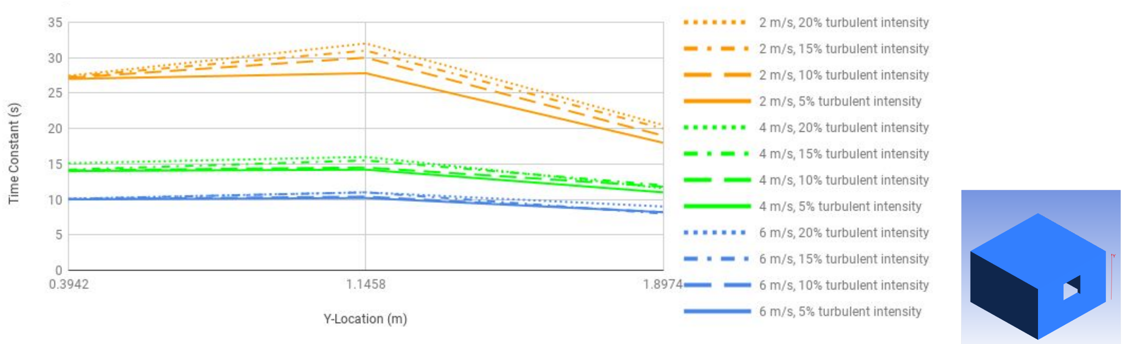


Fig. 11. Average effect of y-location on the time constant

Fig. 12 demonstrates that at lower inlet velocities, the air replacement happens the best in the middle of the room along the windows rather than on the sides. However, this effect disappears at higher inlet velocities, perhaps because mixing is more efficient at higher velocities. Similar to Figs. 10 and 11, Fig. 12 also shows that increasing inlet velocity decreases the time constant and results in better air replacement inside the building. Figs. 10-12 show that numerical results are in good agreement with the results obtained from experiments.

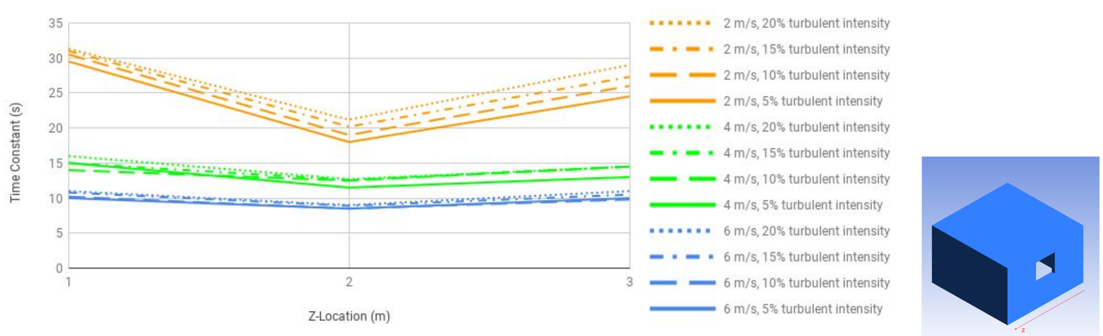
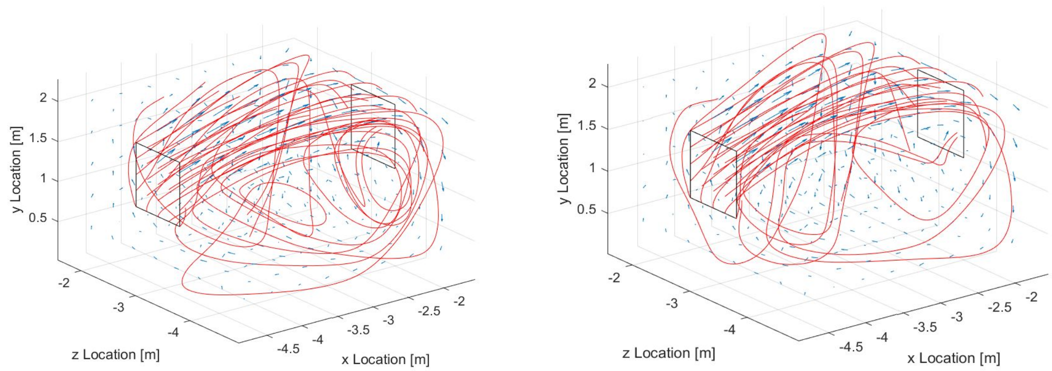


Fig. 12. Average effect of z-location on the time constant

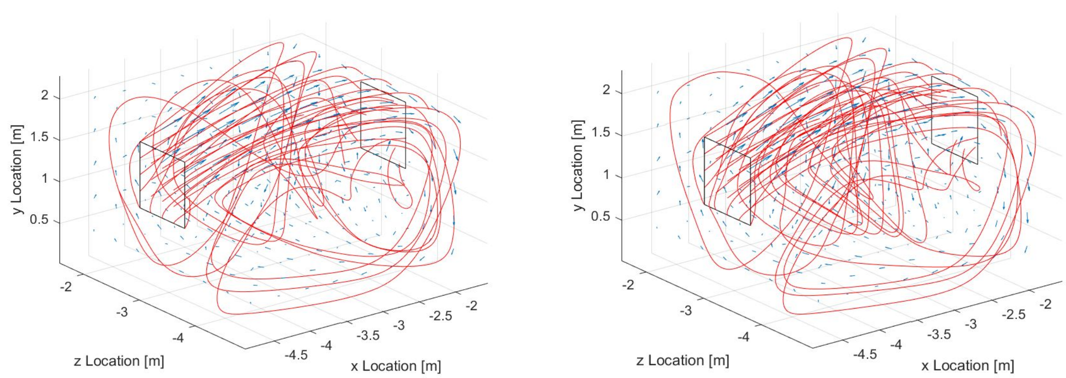
C. Streamline and Velocity Vector Plots

Figs. 13-15 show the velocity vectors inside the room for three different inlet velocities with the minimum and maximum turbulent intensities investigated.



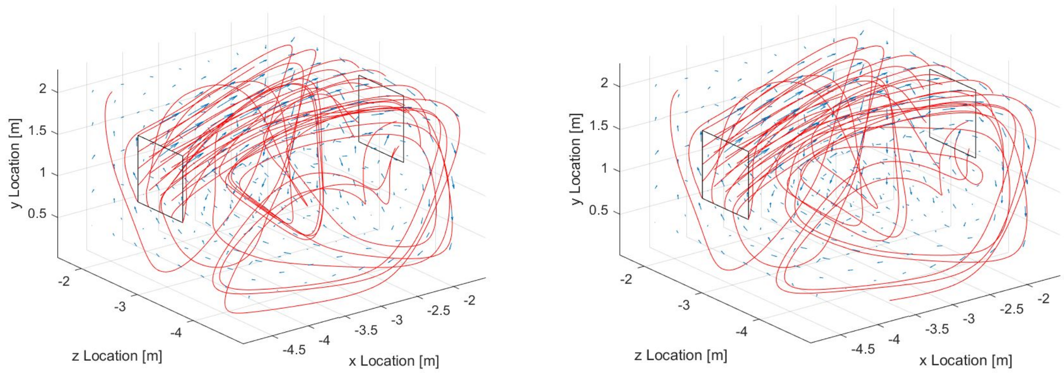
5% turbulent intensity 20% turbulent intensity

Fig. 13. Streamline and velocity vector plots for an inlet velocity of 2 m/s at 5% and 20% turbulent intensities



5% turbulent intensity 20% turbulent intensity

Fig. 14. Streamline and velocity vector plots for an inlet velocity of 4 m/s at 5% and 20% turbulent intensities



5% turbulent intensity

20% turbulent intensity

Fig. 15. Streamline and velocity vector plots for an inlet velocity of 6 m/s at 5% and 20% turbulent intensities

The streamline plots demonstrate better air circulation in the downstream and more towards the bottom and the middle of the room along the windows. At the inlet velocity of 2 m/s, the air circulation is negatively impacted by the turbulent intensity, meaning that with a 5% turbulent intensity, a better air circulation is visible. However, at the inlet velocities of 4 m/s and 6 m/s, turbulent intensity does not have a notable impact on the air circulation.

The velocity vectors represent the highest velocity magnitudes directly between the windows, while the lowest velocities happen near the walls. This is a good indication of the air turnover inside the room which proves that air circulation happens the best towards the middle of the room between the windows. Turbulent intensity does not seem to have a noteworthy effect on the locations where the velocity magnitude is at its highest or lowest.

The pattern of change of the time constant with the x, y, and z locations across the room is visible in the streamline plots as well. For example, in the y-direction especially at lower velocities a bulk flow moves around the perimeter rather than in the middle of the room rather than resulting in better air circulation at the top and bottom of the room.

VI. Conclusions

This study used a numerical approach to determine the effect of turbulence on the quality of cross-flow ventilation. The results of 12 simulations with different inlet velocities and turbulent intensities show that turbulent intensity has a very slight effect on air circulation, and this effect is more pronounced at lower velocities. In fact, velocity magnitude has a more significant effect on the quality of air circulation compared to turbulent intensity, and higher inlet velocities result in better air replacement inside the room. Therefore, to improve air replacement, using fans to increase flow velocity may be a more effective option than using retrofitting geometries.

References

[1] G. Carrilho da Graça and P. Linden, "Ten questions about natural ventilation of non-domestic buildings", *Building and Environment*, vol. 107, pp. 263-273, 2016.

[2] U.S. Energy Information Administration, Commercial Building Energy Consumption Survey, 2012, 2016.

[3] U.S. Energy Information Administration, Residential Energy Consumption Survey, 2009, 2013.

[4] U.S. Energy Information Administration, Monthly Energy Review, December 2015, 2015.

[5] C. Allocca, Q. Chen and L. Glicksman, "Design analysis of single-sided natural ventilation", *Energy and Buildings*, vol. 35, no. 8, pp. 785-795, 2003.

[6] E. Gratia and A. De Herde, "Guidelines for improving natural daytime ventilation in an office building with a double-skin facade", *Solar Energy*, vol. 81, no. 4, pp. 435-448, 2007.

[7] P. Karava, T. Stathopoulos and A. Athienitis, "Wind-induced natural ventilation analysis", *Solar Energy*, vol. 81, no. 1, pp. 20-30, 2007.

[8] M. Haase and A. Amato, "An investigation of the potential for natural ventilation and building orientation to achieve thermal comfort in warm and humid climates", *Solar Energy*, vol. 83, no. 3, pp. 389-399, 2009.

[9] K. Zhong, X. Yang, W. Feng and Y. Kang, "Pollutant dilution in displacement natural ventilation rooms with inner sources", *Building and Environment*, vol. 56, pp. 108-117, 2012.

[10] M. Bangalee, J. Miau, S. Lin and J. Yang, "Flow visualization, PIV measurement and CFD calculation for fluid-driven natural cross-ventilation in a scale model", *Energy and Buildings*, vol. 66, pp. 306-314, 2013.

[11] T. Larsen and P. Heiselberg, "Single-sided natural ventilation driven by wind pressure and temperature difference", *Energy and Buildings*, vol. 40, no. 6, pp. 1031-1040, 2008.

[12] Z. Bu, S. Kato and T. Takahashi, "Wind tunnel experiments on wind-induced natural ventilation rate in residential basements with areaway space", *Building and Environment*, vol. 45, no. 10, pp. 2263-2272, 2010.

[13] S. Kato, S. Murakami, T. Takahashi and T. Gyobu, "Chained analysis of wind tunnel test and CFD on cross ventilation of large-scale market building", *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 67-68, pp. 573-587, 1997.

[14] L. Ji, H. Tan, S. Kato, Z. Bu and T. Takahashi, "Wind tunnel investigation on influence of fluctuating wind direction on cross natural ventilation", *Building and Environment*, vol. 46, no. 12, pp. 2490-2499, 2011.

[15] S. Omrani, V. Garcia-Hansen, B. Capra and R. Drogemuller, "Effect of natural ventilation mode on thermal comfort and ventilation performance: Full-scale measurement", *Energy and Buildings*, vol. 156, pp. 1-16, 2017.

[16] H. Pabiou, J. Salort, C. Ménézo and F. Chillà, "Natural Cross-ventilation of Buildings, An Experimental Study", *Energy Procedia*, vol. 78, pp. 2911-2916, 2015.

[17] C. Walker, G. Tan and L. Glicksman, "Reduced-scale building model and numerical investigations to buoyancy-driven natural ventilation", *Energy and Buildings*, vol. 43, no. 9, pp. 2404-2413, 2011.

[18] T. van Hooff, B. Blocken and Y. Tominaga, "On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: Comparison of RANS, LES and experiments", *Building and Environment*, vol. 114, pp. 148-165, 2017.

[19] Y. Jiang, C. Allocca and Q. Chen, "Validation of CFD Simulations for Natural Ventilation", *International Journal of Ventilation*, vol. 2, no. 4, pp. 359-369, 2004.

[20] T. Jiru, T. and G. Bitsumalak, "Advances in applications of CFD to natural ventilation", *The Fifth International Symposium on Computational Wind Engineering,* vol. 2, no. 2, pp. 131-147, 2010.

[21] M. Jin, W. Zuo and Q. Chen, "Simulating Natural Ventilation in and Around Buildings by Fast Fluid Dynamics", *Numerical Heat Transfer, Part A: Applications*, vol. 64, no. 4, pp. 273-289, 2013.

[22] B. Hughes, J. Calautit and S. Ghani, "The development of commercial wind towers for natural ventilation: A review", *Applied Energy*, vol. 92, pp. 606-627, 2012.

[23] I. Abohela, N. Hamza and S. Dudek, "Effect of roof shape, wind direction, building height and urban configuration on the energy yield and positioning of roof mounted wind turbines", *Renewable Energy*, vol. 50, pp. 1106-1118, 2013.

[24] H. Wang and Q. Chen, "Modeling of the Impact of different Window Types on Single-sided Natural Ventilation", *Energy Procedia*, vol. 78, pp. 1549-1555, 2015.

[25] K. Visagavel and P. Srinivasan, "Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD", *Solar Energy*, vol. 83, no. 1, pp. 2-5, 2009.

[26] T. Schulze and U. Eicker, "Controlled natural ventilation for energy efficient buildings", *Energy and Buildings*, vol. 56, pp. 221-232, 2013.

[27] M. Ohba and I. Lun, "Overview of natural cross-ventilation studies and the latest simulation design tools used in building ventilation-related research", *Advances in Building Energy Research*, vol. 4, no. 1, pp. 127-166, 2010.

[28] P. Linden, "THE FLUID MECHANICS OF NATURAL VENTILATION", *Annual Review of Fluid Mechanics*, vol. 31, no. 1, pp. 201-238, 1999.

[29] M. Bangalee, J. Miau, S. Lin and M. Ferdows, "Effects of Lateral Window Position and Wind Direction on Wind-Driven Natural Cross Ventilation of a Building: A Computational Approach", *Journal of Computational Engineering*, vol. 2014, pp. 1-15, 2014.

[30] M. Ryan and G. Tan, "An Overview of Unsteady Analysis Techniques for Natural Wind Turbulence and its Effects on Natural Ventilation", *International Journal of Ventilation*, vol. 13, no. 1, pp. 65-76, 2014.

[31] C. Chu, Y. Chiu, Y. Chen, Y. Wang and C. Chou, "Turbulence effects on the discharge coefficient and mean flow rate of wind-driven cross-ventilation", *Building and Environment*, vol. 44, no. 10, pp. 2064-2072, 2009.

[32] J. Fransson, "Transition to Turbulence Delay Using a Passive Flow Control Strategy", *Procedia IUTAM*, vol. 14, pp. 385-393, 2015.

[33] M. Chishty, H. Hamdani and K. Parvez, “Effect of turbulent intensities and passive flow control on LP turbine”, *10th International Bhurban Conference on Applied Sciences & Technology (IBCAST)*, 2013.

[34] P. Martínez-Filgueira, U. Fernandez-Gamiz, E. Zulueta, I. Errasti and B. Fernandez-Gauna, "Parametric study of low-profile vortex generators", *International Journal of Hydrogen Energy*, vol. 42, no. 28, pp. 17700-17712, 2017.

[35] P. Nejat, J. Calautit, M. Majid, B. Hughes, I. Zeynali and F. Jomehzadeh, "Evaluation of a two-sided windcatcher integrated with wing wall (as a new design) and comparison with a conventional windcatcher", *Energy and Buildings*, vol. 126, pp. 287-300, 2016.

[36] L. Bardal and L. Sætran, "Influence of turbulence intensity on wind turbine power curves", *Energy Procedia*, vol. 137, pp. 553-558, 2017.

[37] A. Coto, G. Gosselin, R. Lee, and J. Zhu, "Experimental Studies of Cross-Flow Ventilation in the Presence of Turbulence", *Unpublished manuscript,* 2018.

[38] J. Cermak and N. Isyumov, "Wind tunnel studies of buildings and structure", *ASCE Publications*, pp. 9-15, 1999.

[39] J. Cermak, "Applications of Fluid Mechanics to Wind Engineering—A Freeman Scholar Lecture", *Journal of Fluids Engineering*, vol. 97, no. 1, p. 9, 1975.