Proceedings of the ASME 2013 International Mechanical Engineering Congress and Exposition
IMECE2013
November 15-21, 2013, San Diego, California, USA

IMECE2013-65843

COMPUTATIONAL STUDY OF SINGLE-SIDED VENTILATION THROUGH A 3-DIMENSIONAL ROOM WITH ROUNDED EDGES

A. Idris
Department of Mechanical Engineering
Universiti Kuala Lumpur
Malaysian Spanish Institute
Kulim Hi-Tech Park, 09000 Kulim, Kedah
Malaysia

B.P. Huynh
Faculty of Mechanical Engineering & IT
University of Technology, Sydney
Sydney, NSW, Australia

ABSTRACT

A commercial Computational Fluid Dynamics (CFD) software package is used to investigate numerically a 3-dimensional rectangular-box room with rounded edges. The room has all its window openings located on one wall only. The standard *K–ε* turbulence model is used. Air's flow rate and flow pattern are considered in terms of wind speed and the openings' characteristics, such as their number, location, size and shape. Especially, comparison with ventilation rate corresponding to when the room edges are sharp is made; and thereby the effects of the edges being rounded are examined.

INTRODUCTION

Natural ventilation is created by pressure differences between the inside and outside of building induced by wind and air temperature differences, and it has been widely recognized as one way towards achieving low-energy building and clean environment. Air exchange is necessary to remove stale air and odours, dilute the product of combustion and respiration, remove harmful chemicals, and combat excess heat. Effective ventilation in a building will affects the occupant health and productivity.

In achieving optimum efficiency of natural ventilation, the building design should start from the climatic conditions and orography of the construction to ensure the building permeability to the outside airflow to absorb heat from indoors to reduce temperatures. In the United State, large and thick building shapes, fire codes, security requirements and privacy concerns often prevent design of cross ventilations eventhough the single-sided ventilation has fewer adaptive comfort hours than cross ventilation and much less ventilation volume [1].

In temperate climates, depending on local variations, design and occupational patterns, roughly 50 percent of total energy use for a typical building is used for indoor air conditioning, in term of cooling and heating[2]. According to Linden, the energy used in building corresponds to a significant proportion of total energy consumption, particularly in countries where the proportion of air-conditioned building is large. In the US, about 30% of total energy consumption is used in non-domestic buildings, and of that fraction about 30% is used in heating and cooling [3]. A study conducted in Sydney showed a reduction of 25 – 33% in naturally ventilated mixed mode buildings and high occupant comfort satisfaction score[4]. In OECD countries, the building sectors' share of total energy consumption remains between 25% and 40% [5].

Most of the buildings in urban area are ventilated with single-sided ventilation only. Single-sided ventilation occurs when the building communicates with the outdoor environment through one or more openings located on the same exterior wall. Effective ventilation is important for buildings: houses, shelters, mobile homes, warehouses, greenhouses, etc. There are two major natural ventilation types: cross and single-sided ventilation. In both cases, air is driven in/out of the building due to pressure differences across the openings, which result from the combined action of wind and buoyancy driven forces. The physical processes are very complex and the interpretation of their role in ventilation effectiveness is a difficult task. Mathematical interpolation of the physical phenomena related to this type of ventilation has so far been handled by empirical and simulation models. Computational Fluid dynamic (CFD) has proven to be a powerful tool for the prediction of air flow and related variables distribution, thus helping the design and evaluation of HVAC and fire emergency systems under a wide range of scenarios[6].

NOMENCLATURE

U_j (j=1-3) – component of average velocity vector
g-the gravity acceleration
μ_t- turbulent viscosity
C_μ- empirical constant
K-turbulent kinetic energy
ε- dissipation rate
κ- Von Karman's constant
U_{ave}- the average flow velocity
T_i –turbulence intensity
L- length scale

METHODOLOGY

a. Problem description

When dealing with wind driven natural ventilation, two different ventilation patterns can be identified, namely cross ventilation and single-sided ventilation. The first one is characterized by the presence of two or more openings on the opposite sides of the building, and it can provide ventilation rates larger than those obtained by single-sided ventilation where the openings are located on a single facade. However, there are situations wherein single-sided ventilation is the only option. This occurs, for example, with buildings which have only one side exposed to the wind. Thus single-sided ventilation is investigated in the present study. Five different cases are considered:

- Case 1: Leeward ventilation with an opening on the walls of sharps edges (a) and rounded edges (b).
- Case 2: Left-side ventilation with double opening on the walls of sharps edges (a) and rounded edges (b).
- Case 3: Top ventilation with an opening on the walls of sharps edges (a) and rounded edges (b).
- Case 4: Windward ventilation with an opening on the walls of sharps edges (a) and rounded edges (b).
- Case 5: Cross ventilation of sharps edges (a) and rounded edges (b).

In this work, the intention is focused on computational study on single-sided wind driven ventilation. In order to perform the simulations, a building-like model, whose dimensions are 5 m x 5 m x 5 m. Ten different building models are used: the difference among five models is the location of the openings, which have the same total area of 4 m^2. Figure 1 shows a schematic view of the building model with different openings used in this work. Furthermore, a logarithmic wind profile upwind of the building is employed. The computational domain is sufficiently large to avoid disturbance of air flow around the building as shown in Figure 2.

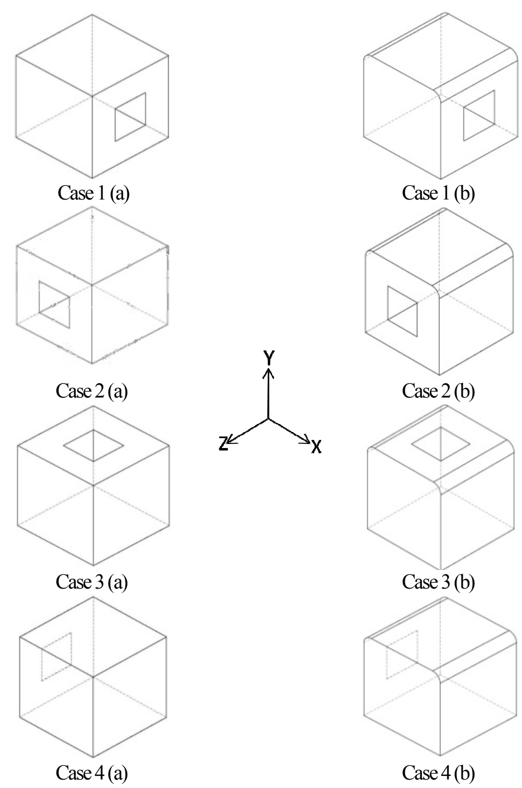
b. Computational method

The commercial software package CFD-ACE from the ESI group is used for the computation. It is based on the numerical solution of the governing equations which are the continuity and Navier-Stokes equations.

The package employs the Finite Volume Method to solve iteratively the couple system of governing equations for the three mean velocity components and pressure, plus K and ε, until satisfactory convergence has occurred, or steady state has been reached.

A convergence criterion of reduction of residuals in the solved variables by four orders of magnitude is adopted. The selection of a structured grid, instead of a more convenient unstructured one was due to more accurate results that may be obtained, at least for the case of the flow around a bluff body, using a structure grid as found in reference. Several attempts was taken to test different mesh distribution started with 583, 200, with 14% increment, until it's reaching 926,100 of total numbers of cell. These trials were made to all the cases and it shows a good results. A 685,900 numbers of cells were retained. Upwind spatial differential schemes have been used. Grid convergence tests have been performed to ascertain the adequacy of grid pattern use. Consequently, a specific technique found in literature review were applied concerning the computational domain [7]. Thus, in the case of vertical wind direction the computational domain was constructed had a height of 4H, width of 9H and length of 13H (H=5m)

All fluid properties are assumed to be constant and corresponding to those of air at 300 K and standard pressure at sea level (101.3 kPa). In common notation, the following values of molecular properties are used: $\rho = 1.1614\ kg/m^3$; $\mu = 1.846\times10^{-5}\ N\text{-}s/m^2$; $\nu = 1.589\times10^{-5}\ m^2/s$.



Case 5 (a) Case 5 (b)

Fig. 1. Description of building models with single-sided ventilation. Mean wind is along the positive x-direction; y is the upward-pointing vertical direction.

The mathematical model applies numerical techniques to solve the Navier-Stokes continuity and energy equations for 3D, turbulent fluid flow. In common notation, governing equations and turbulence-model constants are as follows [8]

$$\frac{\partial U_j}{\partial x_j}=0 \tag{1}$$

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial p}{x_i}+\frac{\partial}{\partial x_j}\left[v\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\overline{u_i u_j}\right] \tag{2}$$

$$-\overline{u_i\, u_j}=v_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\frac{2}{3}K\delta_{ij}$$

$$\frac{\partial K}{\partial t}+U_j\frac{\partial K}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(v+\frac{v_t}{\sigma_K}\right)\frac{\partial K}{\partial x_j}\right]+v_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}-\varepsilon \tag{3}$$

$$K=\frac{1}{2}\overline{u_i\, u_j}\quad;\ \varepsilon=v\left(\frac{\partial u_i}{\partial x_j}\right)\left(\frac{\partial u_i}{\partial x_j}\right)$$

$$\frac{\partial \varepsilon}{\partial t}+U_j\frac{\partial \varepsilon}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(v+\frac{v_t}{\delta_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_1\frac{\varepsilon}{K}v_t\left[\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}\right]-C_2\frac{\varepsilon^2}{K} \tag{4}$$

where subscript t refers to turbulence; $\mu_t={}^{\rho C_\mu K^2}/_{\varepsilon}$; $v_t={}^{\mu_t}/_{\rho}$; $C_\mu=0.09$; $C_1=1.44$; $C_2=1.92$; $\sigma_K=1.0$; $\sigma_\varepsilon=1.0$

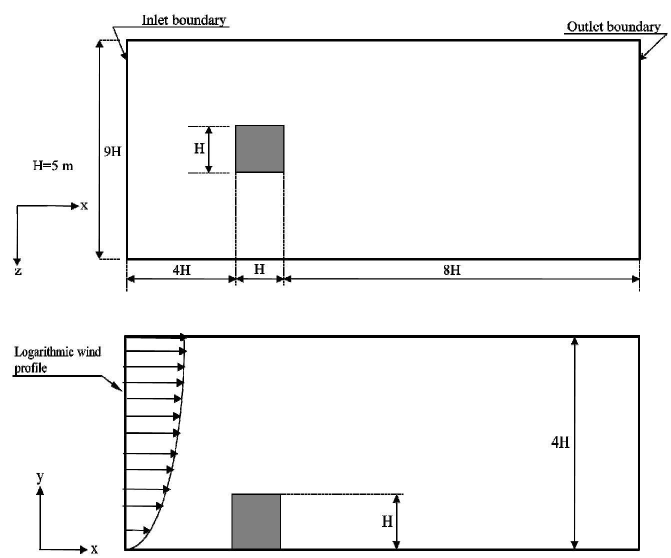


Fig. 2. Dimensions of computational domain

Near the ground, the wind is strongly braked by obstacles and surface roughness. High above the ground in the undisturbed air layers of the geostrophic wind (at approx. 5 km above ground) the wind is no longer influenced by the surface. Between these two extremes, wind speed changes with height. This phenomenon is called *vertical wind shear*. In flat terrain and with a neutrally stratisfied atmosphere, the logarithmic wind profile is a good estimation for the vertical wind shear [8]:

$$U_h=U_{hz_{ref}}\frac{ln\left[\frac{h}{h_o}\right]}{\ln\left[\frac{h_{ref}}{h_o}\right]} \tag{5}$$

where h is the height from the ground, U_h wind speed at h, $U_{hz_{ref}}$ the reference wind speed measured at reference height h_{ref}, h_o the roughness height. In this work, the following values are employed: $U_{hz_{ref}}$ = 5 m/s, h_{ref}= 10 m and h_o = 0.15 m [8]. The velocity components along Y and Z directions are equal to zero.

In order to determine the inlet turbulence values, turbulence intensity is assumed. For internal flows the turbulence intensity can be around 1-5%. For this simulation, turbulence intensity is assumed to be 5%. Therefore, the following relations are used for K and ε at inlet to the computational flow domain:

$$K=\frac{3}{2}(U_{ave}T_i)^2 \tag{6}$$

$$\varepsilon=\frac{C_\mu^{3/4}K^{3/2}}{\kappa L} \tag{7}$$

where U_{ave} is the average flow velocity, T_i is the turbulence intensity, L is the reasonable length scale and κ = 0.41 is the Von Karman's constant.

According to the literature, the outlet of the computational domain [7], a constant pressure is assumed, while the gradients of all the dependent variables are assumed to vanish. Cross-ventilation (Case 5) was simulated in this investigation purposely to verify that the grid number of geometrical model used is trusted.

RESULT AND DISCUSSION

The main purpose of the present study is to investigate the flow patterns and air flow rate of sharp edges buildings and rounded edges buildings using RANS scheme when modeling wind driven natural ventilation in buildings. In Fig. 3, three vertical lines in the middle section of the building A, B, and C are identified. Locations A and C have been chosen in order to describe the flow pattern close to the openings. The velocity components have been determined, along streamwise and vertical direction, respectively. The third velocity component in Z axis has been neglected as it is supposed to be zero, due to the symmetry of the domain.

The plots in Figs. 4 – 8 show the vertical distribution of X-direction velocity U for the 10 ventilation patterns evaluated. Numerical results have been obtained through K-ε model. It is shows in Table 1.

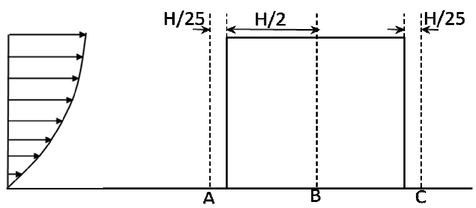


Fig. 3. Location of air velocity is measured of simulation

Fig. 9 illustrates the flow pattern inside the building for the five configuration considered in this analysis. For the Case 1, 2 4, and 5 it is found that the sharp edges and rounded edges has almost same flow pattern except for the Case 3.

Among the configurations of single-sided ventilation, the air flow rate of rounded edges is higher than sharp edges. The cross-ventilation (Case 5) shows a very high air flow rate compare to the single-sided but in this case the sharp edges has a higher flow rate than the rounded edges. It is believe the rounded edges not very much blocking the air flow through the top of building then it gives the good effect to the single-sided ventilation.

Table 1: Numerical values of the ventilation rate (m^3/s) of sharp edges buildings

Cases	Air flow rate (m^3/s)	Cases	Air flow rate (m^3/s)
Case 1 (a)	2.01×10^{-9}	Case 1 (b)	2.29×10^{-5}
Case 2 (a)	4.01×10^{-8}	Case 2 (b)	5.43×10^{-4}
Case 3 (a)	3.07×10^{-9}	Case 3 (b)	7.66×10^{-4}
Case 4 (a)	6.69×10^{-9}	Case 4 (b)	7.39×10^{-5}
Case 5 (a)	7.2×10^{0}	Case 5 (b)	7.01×10^{0}

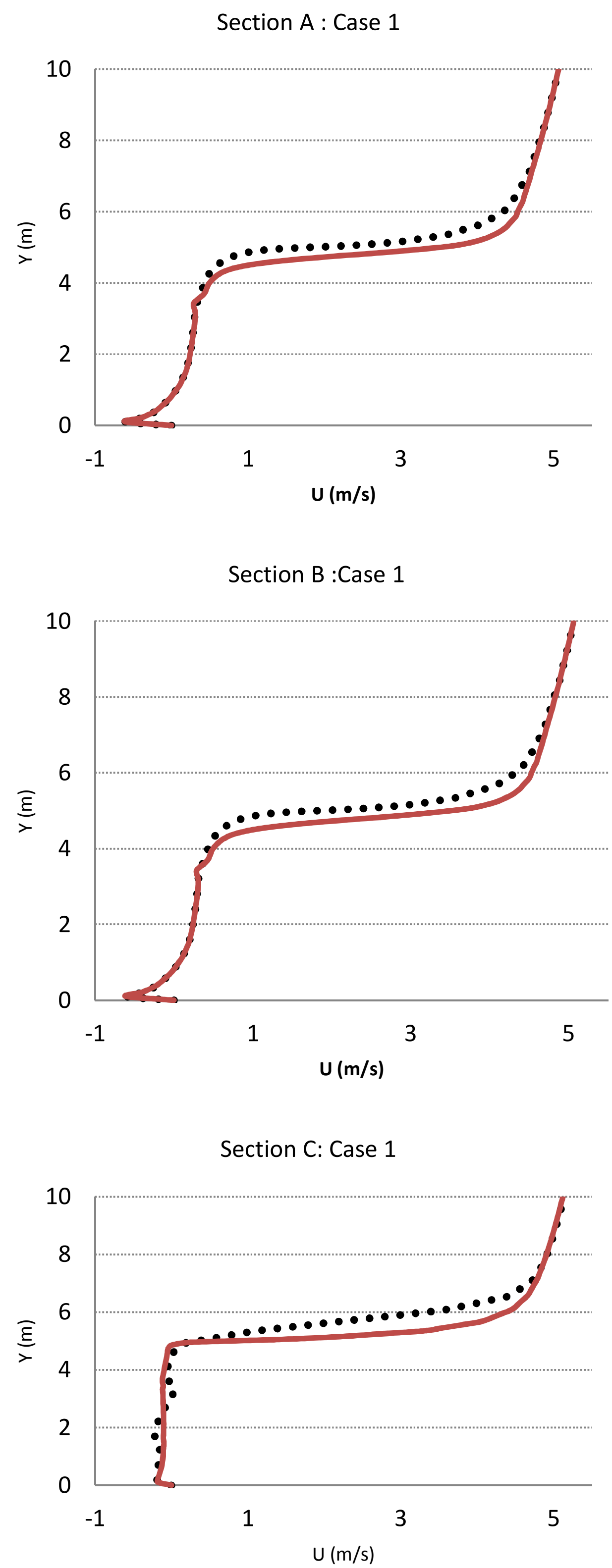


Fig. 4. Velocity distribution for single-sided, leeward ventilation (Case 1). Dots line: Sharp edges model; Solid line: Rounded edge model

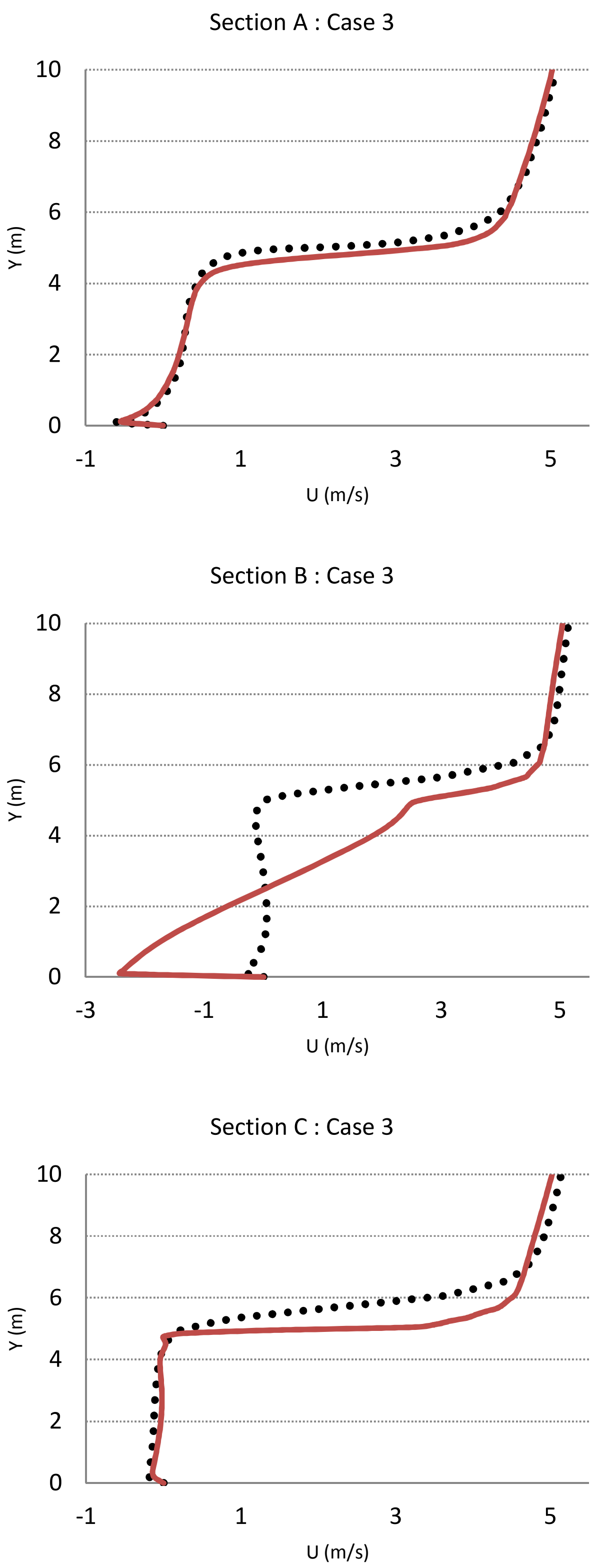


Figure 6. Velocity distribution for single-sided, top ventilation (Case 3). Dots line: Sharp edges model; Solid line: Rounded edge model

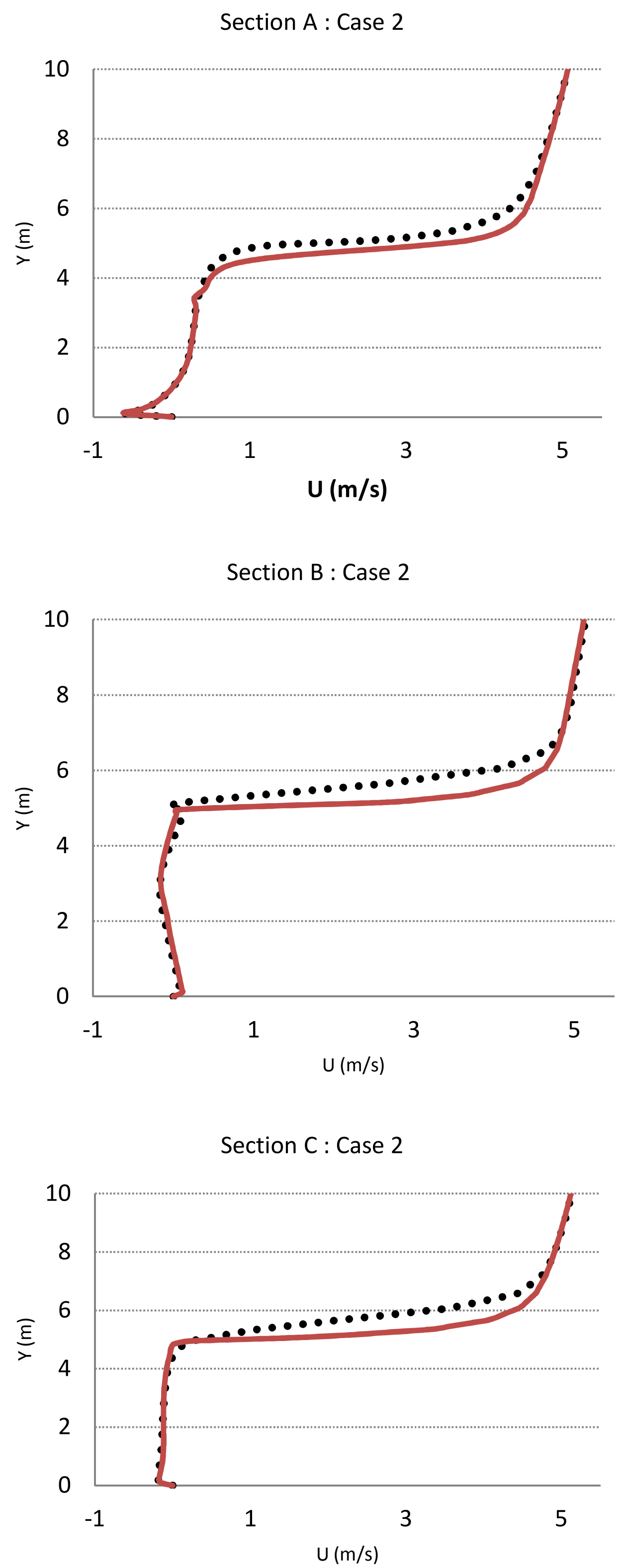


Fig. 5. Velocity distribution for single-sided, left-side ventilation (Case 2). Dots line: Sharp edges model; Solid line: Rounded edge model

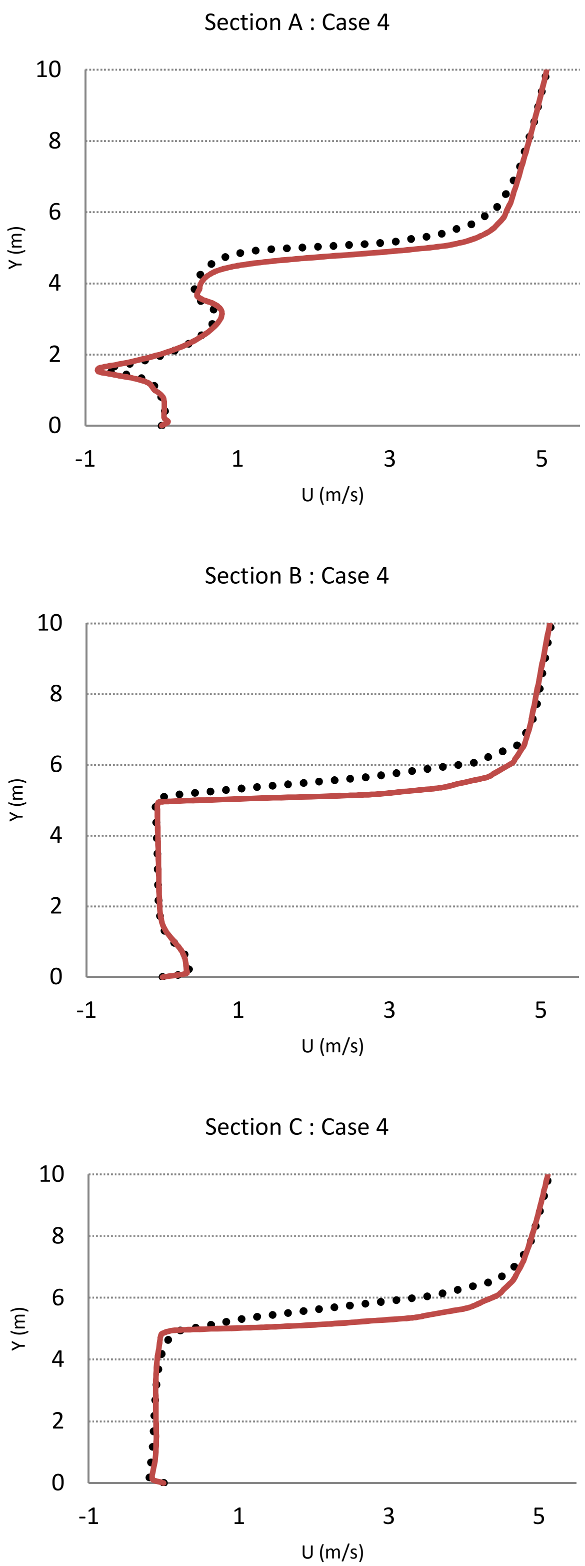


Figure.7. Velocity distribution for single-sided, windward ventilation (Case 4). Dots line: Sharp edges model; Solid line: Rounded edge model

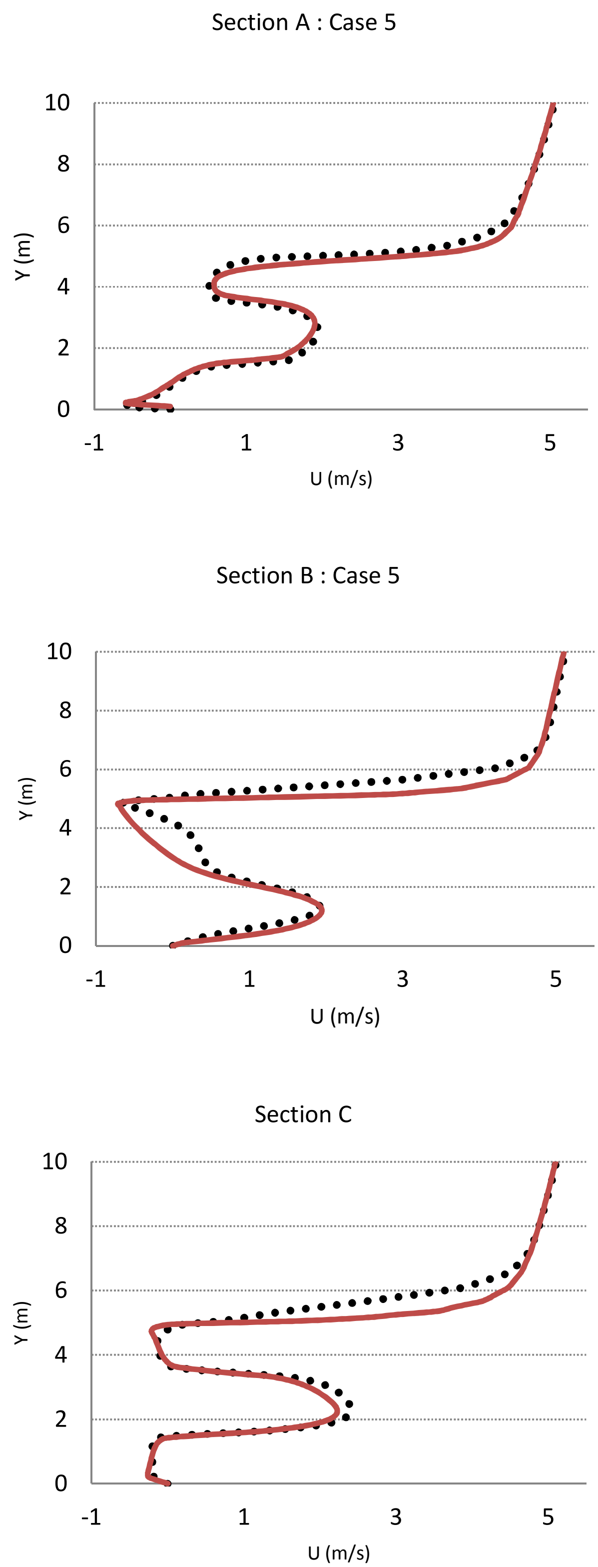
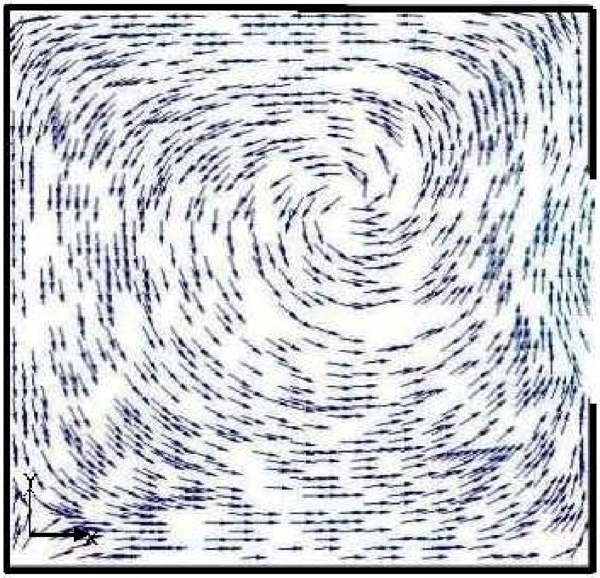
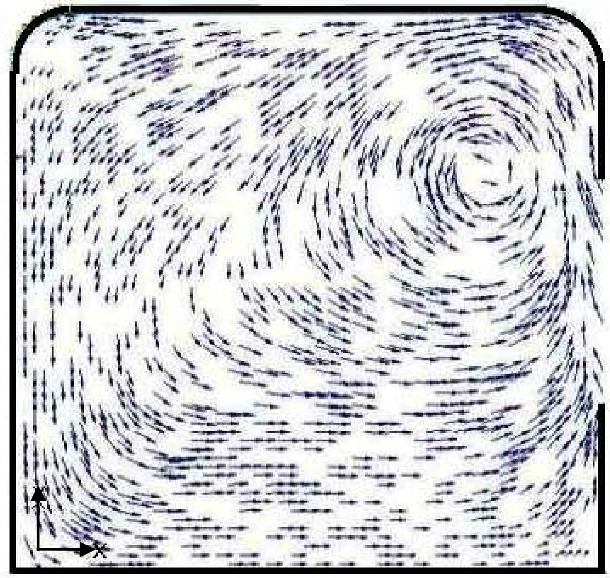


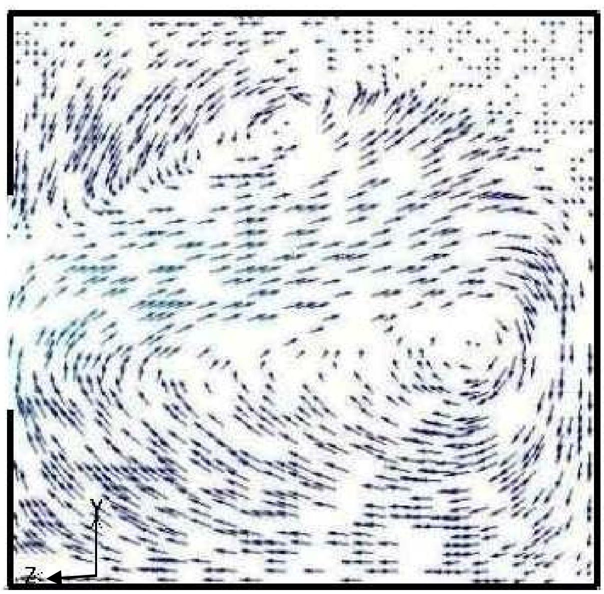
Figure 8. Velocity distribution for single-sided, cross ventilation (Case 5). Dots line: Sharp edges model; Solid line: Rounded edge model



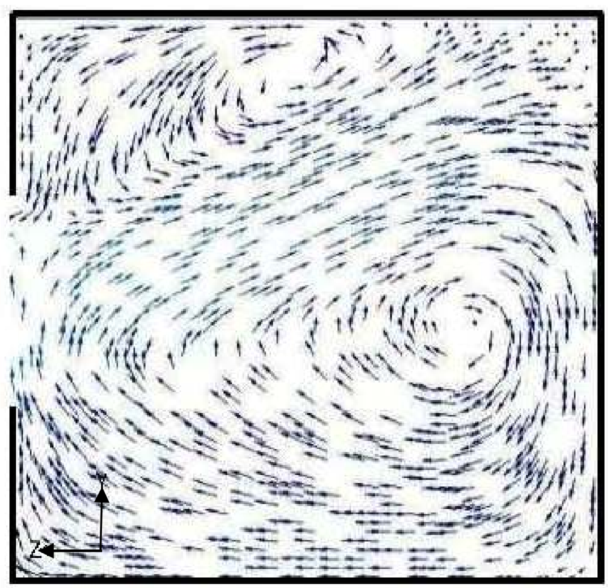
Case 1 (a)



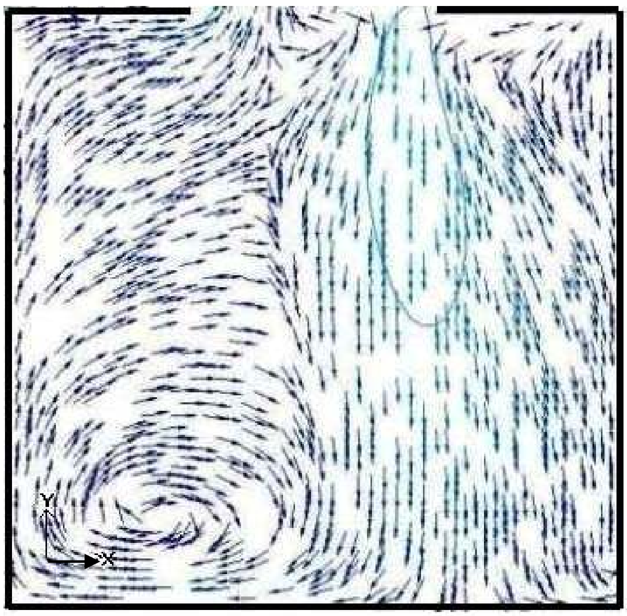
Case 1 (b)



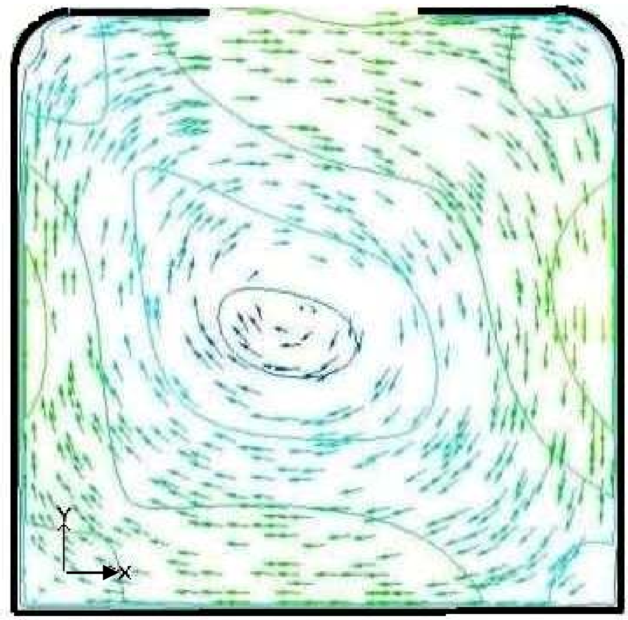
Case 2 (a)



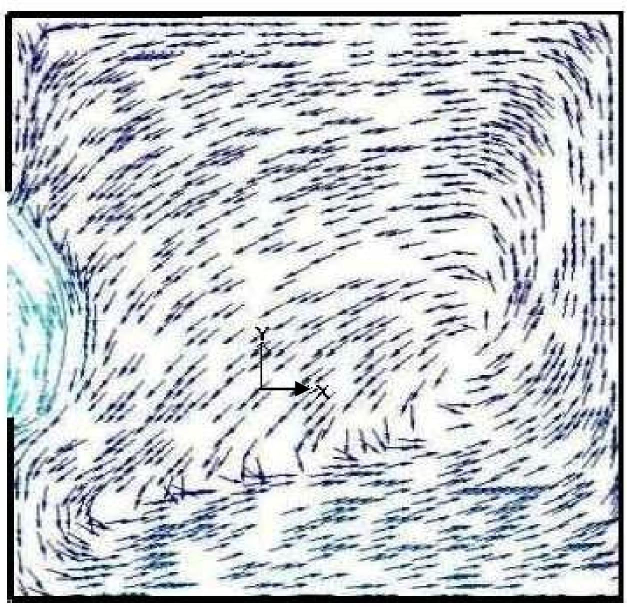
Case 2 (b)



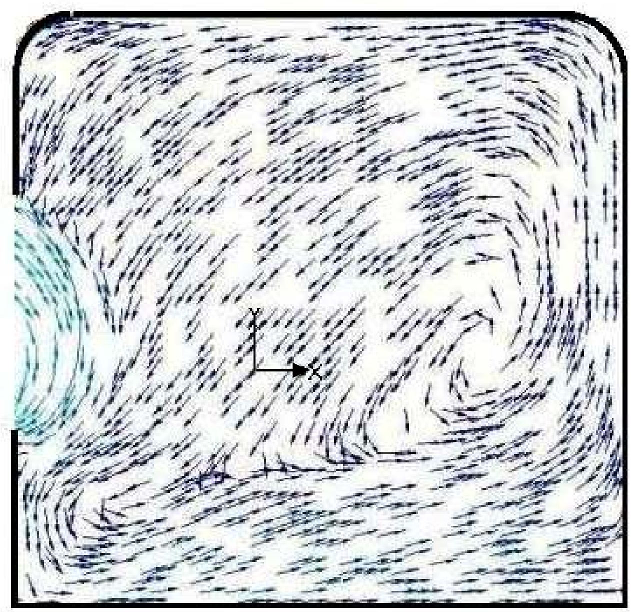
Case 3 (a)



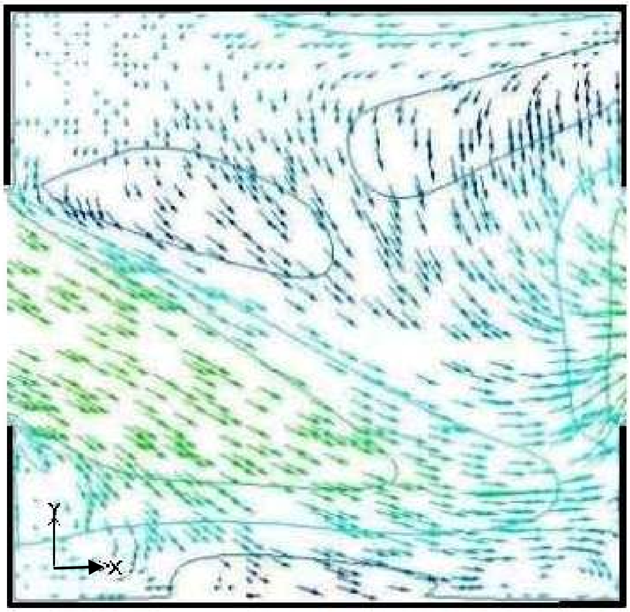
Case 3 (b)



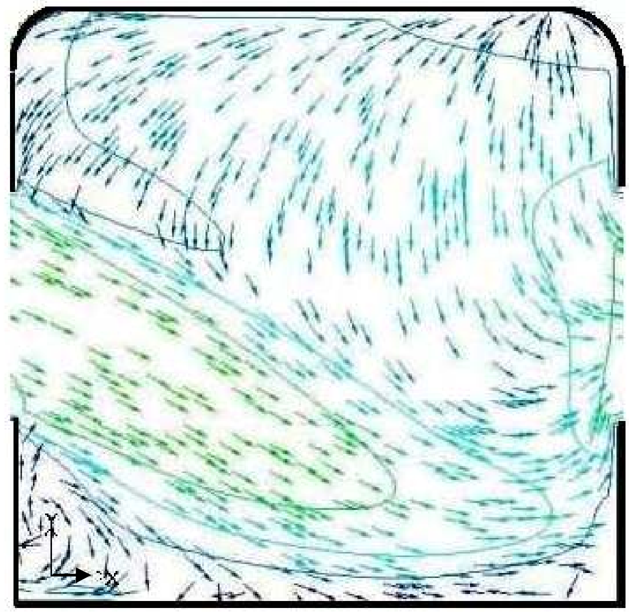
Case 4 (a)



Case 4 (b)



Case 5 (a)



Case 5 (b)

CONCULUSION

In the present study, Reynolds averaged Navier–Stokes approach has been applied to wind driven natural ventilation in a cubic building; ten different configurations were considered (Five configurations of sharp edges and another five configurations of rounded edges). The simulations were performed with the standard turbulence *K–ε* models. As a result, the velocity distribution inside and around the building was determined, and the ventilation rate.

The numerical results show that the rounded edges building will provide more ventilated air compared to sharp edges building. The cross ventilation been simulated for the verification of the right grid numbers used. Further computational investigation will be performed using RNG and LES. However, a further experimental investigation is needed to validate the simulation results.

REFERENCES

[1] Y. Wei, Z. Guo-qiang, W. Xiao *et al.*, "Potential model for single-sided naturally ventilated buildings in China," *Solar Energy,* vol. 84, no. 9, pp. 1595-1600.

[2] R. M. Pulselli, E. Simoncini, and N. Marchettini, "Energy and emergy based cost-benefit evaluation of building envelopes relative to geographical location and climate," *Building and Environment,* vol. 44, no. 5, pp. 920-928, 2009.

[3] P.F.Linden, "The Fluid Mechanics of Natural Ventilation," *Annual Rev. Fluid Mechanic,* vol. 31, pp. 201-238, 1999.

[4] C. Atkins, "All in the Mind. Essential Matters 2007 ", 2007.

[5] J. Morrissey, T. Moore, and R. E. Horne, "Affordable passive solar design in a temperate climate: An experiment in residential building orientation," *Renewable Energy,* vol. 36, no. 2, pp. 568-577.

[6] J. Domingo, R. Barbero, A. Iranzo, "Analysis and optimization of ventilation systems for an underground transport interchange building under regular and emergency scenarios," *Tunnelling and Underground Space Technology,* vol. 26, no. 1, pp. 179-188.

[7] G. Evola, and V. Popov, "Computational analysis of wind driven natural ventilation in buildings," *Energy and Buildings,* vol. 38, no. 5, pp. 491-501, 2006.

[8] H.Versteeg, and W.Malalasekra, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method Approach*, New York: John Wiley & Sons, 1995.