

Field Modelling on Natural Ventilation for Houses With Different Opening Configurations

Yanfeng Li¹, Manpo Niu¹, Gaowan Zou², Huairong Huang¹

¹ College of Architecture and Civil Engineering, Beijing University of Technology, Beijing, 100022, China

² School of Architecture and Civil Engineering, Harbin Engineering University, Harbin, 150001, China

Abstract: A three-dimensional mathematical model is proposed to calculate the natural ventilation properties for an isolated room with different opening configurations. The Reynolds averaged Navier-Stokes (RANS) models is adopted and the standard k- ϵ turbulence mode is selected to study the wind-driven cross natural ventilation. Flow around a building, internal ventilation in building, pressure field are described and circulating flow phenomenon is captured. The calculated pressure coefficients on the mid-plane of building are compared with those of experimental data. The results show that the steady flow pressure coefficient has the same trend as the experimental results. It is also found that the plane of ventilated zone seems to have larger at exit for a larger exit and more air momentum transport would be carried by convection with the increase of exit area. The model can help in assigning comfortable locations for human inside the room.

Keywords: natural ventilation, building, fielding modelling, airflow

1. INTRODUCTION

Natural ventilation in buildings may create a comfortable and healthy indoor environment, and save energy compared to mechanical ventilation systems. Typically, the energy cost of a naturally ventilated building is 40% less than that of an air-conditioned building. Natural ventilation, therefore, contributes to a sustainable environment by reducing energy use in buildings. It has become a new trend in building design in the architectural community and has been used in many types of buildings, even in highly indoor climate controlled hospitals [1].

There are two main types of natural ventilation: cross and single-sided ventilation. The driving forces for natural ventilation are wind and buoyancy. Differences in wind pressure along the façade and differences between indoor and outdoor temperatures create a natural air exchange between indoor and outdoor air. The ventilation rate depends on the strength and direction of these forces and the resistance of the flow path. These physical processes are complex, and predicting ventilation rates is difficult. Hence, it is challenging to control natural ventilation in order to obtain the required indoor environment conditions.

There are three methods to study natural ventilation: empirical models, experimental measurements, and computational fluid dynamics (CFD) simulations or fielding modelling.

Many designers use empirical models to study the "worst scenario" during which the ventilation is only driven by buoyancy forces because of windless [2]. Although those models are simple and straightforward, they cannot account for the impact of building forms, surroundings, and interior spaces on ventilation performances of a building. Wind-tunnel and full scale

measurement are two commonly used experimental methods to provide detailed and reliable information about natural ventilation [3]. Wind-tunnel tests are often used to study natural ventilation driven by wind forces. However, to study buoyancy-driven ventilation, wind tunnels have difficulties generating high-Grashof number airflows analogous to a full-scale situation as required by a similarity theory [4]. Since the study of natural ventilation driven only by buoyancy forces requires windless conditions, the outdoor environment should be controllable during the experiment. Many projects have shown that the experimental measurement is an expensive method in terms of operation time and equipment costs.

CFD, which is also called fielding modeling, is an alternative approach to study natural ventilation in buildings. In this method, the solution region is divided into a large number of cells for solving key equations on mass, momentum, energy and species numerically. It is becoming popular due to its informative results and low labor and equipment costs. The substantial development of Reynolds averaged Navier-Stokes equation (RASNS) model over the years turned it into a powerful computational tool for predicting airflow and the performance of the RANS models has been remarkably good in many situations. The standard k- ϵ model is a two-equation eddy viscosity model which uses transport equations for two variables: k the turbulence energy, and ϵ the rate of viscous dissipation of turbulence energy. In view of enormous costs and time required for full scale or wind tunnel studies, computational fluid dynamics is employed in this present work.

Many studies have been made on the numerical modelling of natural ventilation [5,6]. However, most of studies were carried out in two-dimensional coordinate. The external airflow

surrounding the building and internal ventilation were calculated in separated phase. In present work an isolated building immersed in an atmospheric boundary layer is considered to test the basic natural ventilation rules affected by relative opening location and different upstream wind conditions. The effects of neighbouring buildings are not considered in order to emphasize the wind pressure on cross section natural ventilation. The RANS model and standard $k-\epsilon$ turbulence model is used to model the airflow in and out of building due to its simplicity and effectiveness. Time-averaged, steady three-dimensional simulation will be done for calculating and describing air speed, circulation zones which are important for the valuation of natural ventilation. The distributions of pressure in-room flow field at the mid plane of openings are also determined.

2. MATHETCIAL MODEL

In order to find a basic rule of cross natural ventilation it is assumed that flow around the isolated building in an atmospheric is steady, turbulent, Newtonian and incompressible for natural ventilation systems.

2.1 Governing equations

The prediction of air flow in a building isolated building immersed in an atmospheric boundary layer is based on the solution of general transport equation:

$$\text{div}[\rho \vec{V} \Phi - \Gamma_* \text{grad}(\Phi)] = S_* \quad (1)$$

where Φ denotes the dependent variable, ρ is the air density, $\vec{V}$ is velocity vector, Γ_* is the diffusion coefficient for variable Φ and S_* is source terms for variable Φ , Γ_* and S_* are given in Table 1 [7]. Values of the empirical constants C_1 , C_2 , C_μ , σ_k , σ_ϵ , σ , and σ_p are taken as those recommended by Launder and Spalding[8]. The combined laminar and turbulent stresses hen expressed by means of μ_{eff} in Table 1. μ is the dynamic viscosity, and μ_t is the local turbulent viscosity. Since wind pressure difference is the major force in the cross section natural ventilation system, the role of buoyancy is ignored. The generation term G in Table 1 is the shear production only.

2.2 Boundary condition

At the inlet, undisturbed stream wide velocity is assumed to obey the power law variation with height H . The wind speed of 1.4 m/s is assumed to have been measured at 10 m above ground level. The simulation used the following wind velocity profile [9]:

$$U_H = U_0 \left(\frac{H}{H_0} \right)^a \quad (2)$$

Where U_H is the wind velocity at height H , and U_{ref} is the reference wind velocity at height H_{ref} . In the present study, H_0 and U_0 are 10 m and 1.4 m/s respectively. The constant $a = 0.22$. At the inlet of

calculated region, the initial conditions for k and ϵ are:

$$k = 0.0036 U^2 \quad (3)$$

$$\epsilon = C_\mu k^{3/2} / l \quad (4)$$

l is the characteristic length and is taken to be 0.2 L , L is the building length.

Table 1: Transport equations for variable Φ in the flow field

Equation	Φ	Γ_*	S_*
Continuity	1	0	0
Momentum	u	μ_{eff}	$-\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[\mu_{eff} \frac{\partial u}{\partial x} \right] + \frac{\partial}{\partial y} \left[\mu_{eff} \frac{\partial v}{\partial x} \right] + \frac{\partial}{\partial z} \left[\mu_{eff} \frac{\partial w}{\partial x} \right]$
Momentum	v	μ_{eff}	$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[\mu_{eff} \frac{\partial u}{\partial y} \right] + \frac{\partial}{\partial y} \left[\mu_{eff} \frac{\partial v}{\partial y} \right] + \frac{\partial}{\partial z} \left[\mu_{eff} \frac{\partial w}{\partial y} \right]$
Momentum	w	μ_{eff}	$-\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left[\mu_{eff} \frac{\partial u}{\partial z} \right] + \frac{\partial}{\partial y} \left[\mu_{eff} \frac{\partial v}{\partial z} \right] + \frac{\partial}{\partial z} \left[\mu_{eff} \frac{\partial w}{\partial z} \right]$
Kinetic energy	k	$\frac{\mu_{eff}}{\sigma_k}$	$G - \rho \epsilon$
Dissipation rate	ϵ	$\frac{\mu_{eff}}{\sigma_\epsilon}$	$\frac{\epsilon}{k} (C_1 G - C_2 \rho \epsilon)$

$$\mu_{eff} = \mu + \mu_t = \mu + \rho C_\mu k^2 / \epsilon$$

$$G = \mu_t \left\{ 2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 \right\}$$

$$C_\mu = 0.09, \quad C_1 = 1.44, \quad C_2 = 1.92, \quad \sigma_k = 1.0,$$

$$\sigma_\epsilon = 1.3, \quad \sigma_p = 0.7$$

At the upper boundary of configuration, all derivatives of dependent variables are assumed to vanish. Correcting the outlet velocity components strictly imposes the overall mass conservation through the flow domain. Near the ground as well as the inner or outer building surface constant, stress layers are assumed to exist and the wall function is used to avoid the need for large number of nodal points usually required for a region with large velocity gradients.

The isothermal condition is considered for study the wind-driven force condition. Then no energy conservation equation is solved.

2.3 Solution procedure

The finite volume method is used to solve the time-averaged Navier-Stoke equations with a non-uniform network on a rectangular Cartesian grid. Smaller grid size is used near the walls and the ground. Standard wall functions are used for the enclosure walls. In order to satisfy overall conservation of mass, the velocity components are extrapolated from upstream nodal points and then adjusted to the desired air flow rate. At the supply outlet, a constant mass flux is applied. The diffusion terms and other gradients are discretized using the second-order central difference approximation. The first-order upwind scheme was employed for the convective terms. A time step of 1 second was employed for the computational calculations. At any time step, the solution was considered converged if

$$\frac{|\phi^{n+1}(i,j,k) - \phi^n(i,j,k)|}{\max |\phi^{n+1}(i,j,k)|} < 10^{-3} \quad (5)$$

where n is iteration loop counter

2.4 Analyzed physical model

Figure 1 shows the schematic diagram of flow domain under consideration. The dimensions of the isolated building are 5.0 m (length) $\times$ 5.0 m (width) $\times$ 5.0 m (height). In x-direction flow domain extends from 5 times building length upstream of building windward face to 10 times building lengths downstream of the building leeward face. The domain height is 4 times the building height. The width of flow domain on horizontal plane is 9 times the building length. Flow domain extends in y-direction are 4 times building width on building right and left surface respectively

Three building configurations are considered. In the first condition, the room is totally blocked in order to study the pressure difference around the block. In the second condition, there are two windows installed on upwind surface and leeward surfaces of the building respectively as shown in Fig.1. The size of each window is 2.0 $\times$ 2.0 m. In the third condition, there are also two windows installed on upwind surface and leeward surfaces of the building. However the size of the window on the windward surface is 2.0 $\times$ 2.0 m while the window on the opposite surface has the dimension of 2.0 m (width) $\times$ 3.0 m (height). The latter two conditions are selected in

order to study the external and internal flow of the building under natural convection by wind-driven force.

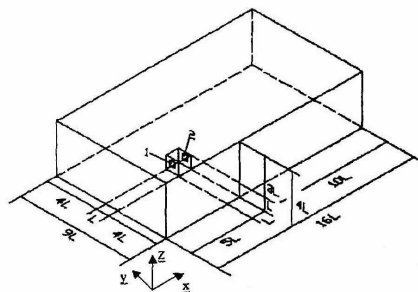


Figure 1: Schematic diagram of the solution region of natural ventilation

The computational domain is divided into 40 $\times$ 30 $\times$ 40 computing cells along the x, y, and z directions of a Cartesian coordinate system. As a compromise between cost and accuracy, all the computations presented here are performed on a 40 $\times$ 30 $\times$ 40 grid system. Results obtained finer grids do not show any noticeable changes in the predicted flow field.

In order to study the wind-driven pressure distribution, the pressure coefficient is defined as:

$$C_p = \frac{P - P_{\infty}}{0.5 \rho U_{\infty}^2} \quad (6)$$

The pressure, airflow velocity at top of flow domain are taken as the reference pressure P_{∞} , U_{∞} for the solution region.

3. RESULTS AND DISCUSSION

3.1 Validation of the natural ventilation model

Pressure coefficients are compared with the experimental values of Castro and Robins [9] in order to validate the proposed mathematical model. Table 2 lists the comparison between experimental data and computational results of pressure coefficient on mid-plane of building. Pressure on upper surface, the windward surface and leeward surface in first conditions are selected. The reference height is 10 m. However, the inlet velocity is considered to be uniform at different heights. Since their experiment was carried for a three-dimensional object, mid plane values are selected for comparison in order to avoid the end effects on surface values of pressure. It can be found the present results have the same trend as the experimental surface pressure values.

3.2 Flow around the building

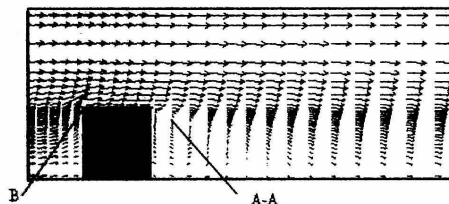
Figures (2a) to (2c) indicate the results of mean velocity vector for the flow around the building in the first and second conditions respectively. The model captures the re-circulating zone downstream of the

building, which extends about six building length downstream. The results show the circulation zone is near the lower corner of upwind building surface. The circulation region of internal flow is clearly shown for the second building configuration.

Table 2: Comparison between experimental data and computational results of pressure coefficient

Point No.	Δx	Δz	Cp Exp*	Cp Cal*
1	L/16	0	-0.9	-0.89
2	L/8	0	-0.95	-0.90
3	L/4	0	-0.75	-0.74
4	3L/8	0	-0.5	-0.49
5	L/2	0	-0.25	-0.35
6	5L/8	0	-0.2	-0.35
7	3L/4	0	-0.18	-0.34
8	7L/8	0	-0.15	-0.34
9	0	-L/16	0.75	0.80
10	0	-L/8	0.8	0.82
11	0	-L/4	0.95	0.95
12	0	-3L/8	0.75	0.75
13	0	-L/2	0.65	0.75
14	0	-5L/8	0.7	0.72
15	0	-3L/4	0.72	0.70
16	0	-7L/8	0.75	0.71
17	L	-L/16	-0.35	-0.40
18	L	-L/8	-0.32	-0.32
19	L	-L/4	-0.30	-0.31
20	L	-3L/8	-0.28	-0.30
21	L	-L/2	-0.28	-0.29
22	L	-5L/8	-0.28	-0.29
23	L	-3L/4	-0.25	-0.26
24	L	-7L/8	-0.24	-0.25

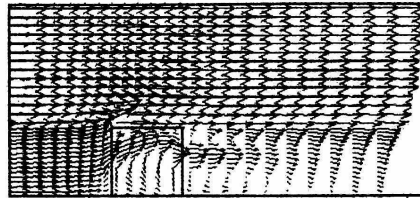
'Exp' mean the measurement results, 'Cal' mean the calculated results. The reference point is located at the top left corner of building on the mid-plane and its coordinate is zero.



(a) Airflow pattern in first building configuration



(b) Enlarge picture of airflow pattern at A-A

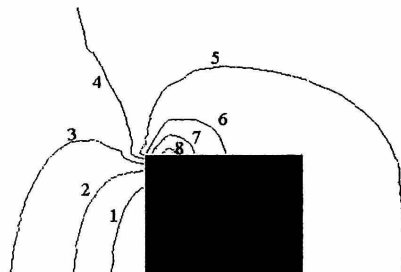


(c) Airflow pattern for second building configuration

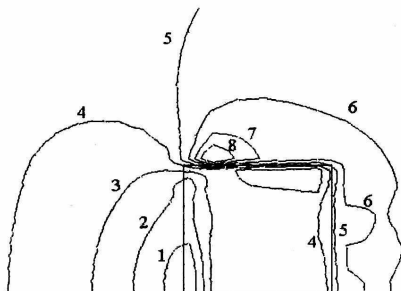
Figure2: Airflow pattern around the building on mid-plane of building

3.3 Pressure distribution around building

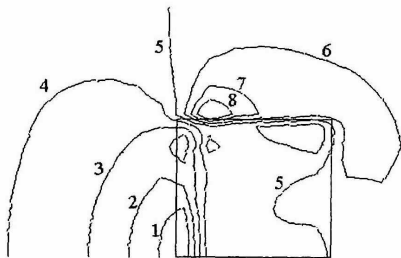
The pressure fields around the building on the mid-plane of building at $y = 22.5$ m in three building conditions are shown in Figs (3a) to (3c) respectively. The reference pressure is 1.1035×10^5 Pa. The relative pressure is shown and the unit is Pa. On the windward surface there is a higher pressure than those in other regions. As for the first condition, the pressure near the left corner on the upper surface is lowest. It can be also found pressure on the upper surface is lower than reference pressure and then pressure coefficient is negative.



(a) Pressure field in first condition (1—0.9, 2—0.58, 3—0.30, 4—0.09, 5— -0.24, 6— -0.40, 7— -0.72, 8— -1.37)



(b) Pressure field in second condition (1—0.91, 2—0.64, 3—0.37, 4—0.12, 5— -0.04, 6— -0.18, 7— -0.40, 8— -0.70)



(c) Pressure field in third condition (1—0.91, 2—0.78, 3—0.37, 4—0.09, 5—-0.04, 6—-0.18, 7—-0.32, 8—-0.73)

Figure 3: Pressure distribution at plane $y = 22.5$

By comparing pressure around the buildings in the second and third conditions it can be found that pressure on the windward surface are similar since they have the inlets of same size. For the building with a larger exit, it can be found the pressure difference between room and leeward surface is less than in the building with a smaller outlet. In addition, pressure difference between room air and air outside the leeward surface of building is larger in third condition. The plane of ventilated zone is larger. This would do good to the circulation of fresh air in the building.

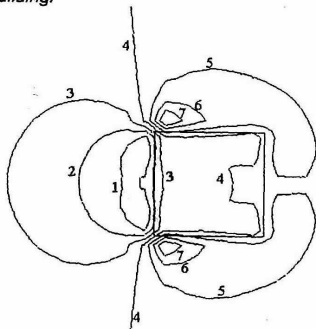


Figure 4: Pressure field at plane of $z = 2.5$ m (1—0.64, 2—-0.37, 3—-0.23, 4—-0.04, 5—-0.04, 6—-0.4, 7—-0.73)

In order to study the edge effects on surface values of pressure, pressure on horizontal plane at $z = 2.5$ m around the building are shown in Fig.4. The reference pressure is 1.1035×10^5 Pa. The relative pressure is shown and the unit is Pa. It can be seen that pressure distribution on this plane is symmetric. The highest pressure appear at locations near the upwind surface. Near the two left building corners, pressure on side surfaces are lowest. Since the present study models the isolated building and wind direction is directed at the building surface where the inlet is installed. If there is an orientation difference or the neighbouring effect should be

considered, three-dimensional model is necessary for calculating the edge effect in natural ventilation.

4. CONCLUSION

An isolated building immersed in atmosphere has been studied. The RANS model and the standard $k-\epsilon$ turbulence model are selected to study the wind-driven cross natural ventilation. Three building configurations are considered. The calculated pressure coefficients at the mid plane of building are compared with the experimental data. It can be found the steady flow pressure coefficient has the same trend as the experimental results. The model captures the circulating flow downstream of the building. The internal flow and external flow are calculated at same time. The circulated flow inside the building is also clearly shown. The larger exit is benefit to cross natural ventilation because more areas could be passed by the stream tube between inlet and exit openings.

ACKNOWLEDGEMENTS

This paper is supported by the Scientific Research Foundation for the Excellent Youth and Middle-aged Person in Beijing.

REFERENCES

- [1] Etheridge, D, Sandberg, M. 1996. Building Ventilation: Theory and Measurement. John Wiley and Sons, Chister, New York. U.S.A.
- [2] Allard, F. 1998. Natural Ventilation in Buildings: A Design Handbook, James & James, London UK.
- [3] Dascalaki, E, Santamouris, M, Argiriou A, Helmis, C, Asimakopoulos, D.N, Papadopoulos K, & Soilemes, A. 1996. On the Combination of Air Velocity and Flow Measurements in Single Sided Natural Ventilation Configurations. Energy and Buildings. Vol. 24. pp 155-165. Greece.
- [4] Katayama, T, Tsutsumi, M, & Ishii, A. 1992. Full-scale Measurements and Wind Tunnel Test on Cross Ventilation. Journal of Wind Engineering and Industrial Aerodynamics. Vol.44. pp 2553-2562. Japan.
- [5] Samir, S.A. 1999. Computational of Natural Ventilation, Journal of Wind Engineering and Industrial Aerodynamics. Vol.82. pp 49-68. Egypt.
- [6] Jiang, Y, & Chen, Q.Y. 2003. Buoyancy-driven Single-sided Natural Ventilation in Building with Large Opening. International Journal of Heat and Mass Transfer. Vol.46, pp 973-988. China
- [7] Tao, W.Q. 2001. Numerical Heat Transfer (second version). Xi'an Jiaotong University Press. Xi'an (in Chinese). China.
- [8] Launder, B.E, & Spalding, D.B. 1972. Lectures in Mathematical Modeling of Turbulence. London: Academic Press. U.K
- [9] Castro, I.P, & Ronins, A.G. 1977. The Flow around a Surface Mounted Cube in Uniform Flow and Turbulent Streams. Journal of Fluid Mechanics. Vol. 79. pp 307-355. U.K.