CFD Simulations to Examine Natural Ventilation of a Work Area in a Public Building

An-Shik Yang, Chiang-Ho Cheng, Jen-Hao Wu, Yu-Hsuan Juan

Abstract—Natural ventilation has played an important role for many low energy-building designs. It has been also noticed as a essential subject to persistently bring the fresh cool air from the outside into a building. This study carried out the computational fluid dynamics (CFD)-based simulations to examine the natural ventilation development of a work area in a public building. The simulated results can be useful to better understand the indoor microclimate and the interaction of wind with buildings. Besides, this CFD simulation procedure can serve as an effective analysis tool to characterize the airing performance, and thereby optimize the building ventilation for strengthening the architects, planners and other decision makers on improving the natural ventilation design of public buildings.

Keywords—CFD simulations, Natural ventilation, Microclimate, Wind environment.

I. INTRODUCTION

WITH the global energy and environment concerns, natural ventilation, considered as old and traditional technology for enhancing building environment, has attracted great attentions. Natural ventilation has been regarded as an efficient built air delivery mode to cut the building energy consumption, especially in the transitional seasons (such as spring and autumn) as the indoor air temperature level approaches the surrounding environment temperature [1]. The two dominant drivers causing natural ventilation flows are buoyancy and wind. In nearly all instances these work all together, with the resulting flow being the combination of the two. Natural ventilation strategies based on wind involve the design of a building with openings on both the windward and leeward exposures of a building. In this case, a pressure differential is created across the building structure as a result of the velocity reducing on the aspect of the building facing the wind, and the velocity increasing as the airflow passes over the roof in addition to other sides of the building [2]. The performance of natural ventilation could be improved through the suitable layout of building orientation and airing openings respecting to the prevailing wind with the concentrated aspects including the air movement and comfort matters in residential areas [3]. The methods employed for investigating the natural ventilation behavior to evaluate the ventilation performance are categorized into three groups: field measurements, computer simulations and controlled experiments in general [4]. Field measurements can only acquire on the site data from selected buildings as well as restricted locations of the instruments for safety reason, causing likely uncertainties of the measurements and difficulties for further data analysis [5]. On the other hand, the experimental results from a controlled state such as the full-scale models in wind tunnel tests are more reliable than those from the field measurements, although management of these experiments are relatively time consuming and high cost [6]. As an alternative approach, the CFD analysis can save the cost and time. Consequently, increasing number of researchers use computer software tools to analyze the physical processes around buildings. For instance, Hussein [7] performed CFD calculations to simulate the wind corrosion phenomenon of the Egyptian monuments site with the countermeasures developed to protect the monuments. Gousseau [8] carried out the airflow computations to characterize the wind field near the buildings. To enhance the air quality and comfort level in the urban area, the predicted results were exercised to identify the potential regions for accumulation of air pollutants. These studies suggest that computer simulations can be a cost-effective and practical method to correctly predict the detailed outdoor and indoor airflow characteristics for appraising the thermal performance of different naturally ventilated designs of buildings [9]. In recent times, many advances have been developed in a variety of CFD analytical models to assess ventilation flow rates and thermal comfort for various natural ventilation configurations [10]-[16]. The main objective of the present study is to conduct CFD simulations using the software Fluent® for resolving the air flow field inside a demonstration building [17]. The predictions were utilized to appraise the natural ventilation effectiveness via the patio and corridors of the building. The computer results of the airflow properties could form a digitized database system and allow the architects to better understand the interaction between the building structures and the urban wind environment for improving their design qualities.

II. DESCRIPTION OF BUILDING MODEL

Using as the demonstration building for study purpose, Fig. 1 illustrates the pictures of the new administrative building of Guanyin Township. It is the first highest grade (i.e. the diamond grade) green building in the Taoyuan, Taiwan. This building is known by its sustainable ventilation design concept using the particular central patio and corridors for aeration enhancement. Because of its coastal nature, it is expected that the wind can largely influence the micro-climate around the building in this study. The building models were generated from the computer software packages consisting of SketchUp®, SolidWorks® and ANSYS Fluent® for the administrative building of Guanyin Township. Essentially, the building is alike the traditional courtyard houses having a central building with two wings

An-Shik Yang, Jen-Hao Wu and Yu-Hsuan Juan are with Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology, Taipei 10608, Taiwan (886-2-2771-2171 ext. 3523; fax: 886-2-2731-49191; e-mail: asyang@ntut.edu.tw).

Chiang-Ho Cheng is with the Department of Mechanical and Automation Engineering, University of Dayeh, Changhua 51591, Taiwan, (886-4-851-1888 ext. 2119; fax: 886-4-851-1224; e-mail: chcheng@mail.dyu.edu.tw).

linked perpendicularly to both ends. The associated geometric dimensions were 62.75 m long, 68.90 m wide and 23.2 m high, respectively. Designed by C.P. Hsueh Architect & Associates, the energy-saving design features were adopted by using the sun shield, solar energy and vent for increasing lighting and natural aeration in the basement. Fig. 2 shows the schematic of the central patio within the administrative building. The designer also united the central patio and corridors to accomplish the ventilation effect in the working area on the third floor.

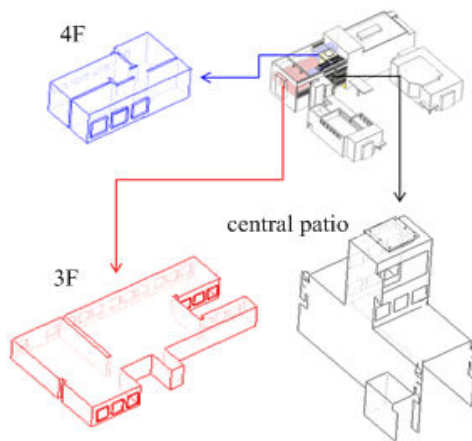Fig. 1 Pictures of the administrative building of Guanyin Township.

Fig. 2 Schematic of the central patio within the administrative building
.

III. COMPUTATIONAL ANALYSIS

The present study considers the terrestrial wind flowing over the administrative building of the Guanyin Township. In agreement with the google map database, the computer software SketchUp® was first employed to create the preliminary three- dimensional (3D) solid model of the geometric information including the terrain of the site and the building structures. Another modeling software SolidWorks® was used to refine the solid model for emulating the buildings details. The scrupulous shapes were specifically rebuilt via their factual dimensions of the full-size building and from the 3D contour maps and satellite images. The solid model was then readily transferred to the preprocessor of the CFD software ANSYS/Fluent® for forming a digital model using the grids generator module ANSYS/ ICEM® [18].

To conduct the computational analysis for evaluating the layout of the residential buildings to be constructed, the physical model takes into account the environmental wind flowing over the administrative buildings of Guanyin Township. The inlet boundary condition in CFD computations of the atmospheric boundary layer (ABL) flow was used to model the associated atmospheric processes [19]. Numerical computations by the CFD software ANSYS/Fluent® were performed to explore the wind field structure characterized by the interaction of wind flow with residence buildings. The theoretical approach was based on the steady-state 3D conservation equations of mass and momentum for the incompressible isothermal turbulent airflow over the calculation domain [20]. The governing equations are stated as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial(\rho u_i u_j)}{\partial x_j} = \rho g_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu_{eff})\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

In the aforementioned equations, u_i designates the velocity component in the i direction; whereas p, ρ, μ_{eff} and ρg_i represent the pressure, density, effective viscosity (defined as the sum of laminar viscosity μ and turbulent viscosity μ_t) and gravity force, respectively. Considered as the most popular, well-established and widely tested turbulence model, a standard k-ε two-equation turbulent model [21] was adopted for turbulence closure, as follows:

$$\frac{\partial \rho u_i k}{\partial x_i} = \rho P - \rho \varepsilon + \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] \tag{3}$$

$$\frac{\partial \rho u_i \varepsilon}{\partial x_i} = C_{\varepsilon 1}\frac{\rho P \varepsilon}{k} - C_{\varepsilon 2}\frac{\rho \varepsilon^2}{k} + \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] \tag{4}$$

$$P = \frac{\mu_t}{\rho}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_j}{\partial x_i} \tag{5}$$

The inflow boundary of the atmospheric boundary layer (ABL) flow was treated to model the related environmental processes [22], [23].

$$u^*_{ABL} = \frac{KU_z}{\ln\left(\frac{h+z_0}{z_0}\right)} \tag{6}$$

$$U_{(z)ABL} = \frac{1}{K} u^*_{ABL} \ln\left(\frac{z+z_0}{z_0}\right) \tag{7}$$

$$k = \frac{u^2_{*ABL}}{\sqrt{C_u}} \tag{8}$$

$$\varepsilon = \frac{u^2_{*ABL}}{K(z+z_0)} \tag{9}$$

The signs z0 and K symbolize the aerodynamic roughness and the von Karman's constant (≈ 0.4). The ABL friction velocity u*ABL is computed from a specified velocity Uh at a reference height h. The sign UABL, the mean inlet velocity at the height of z, can be obtained via (7) to generate a velocity profile within the ABL with the turbulence kinetic energy and dissipation rate computed via (8) and (9), respectively. In essence, roughness will increase the drag for the cross-flow over the surface, leading to the case that the logarithmic law for velocity profile, being the basis of the standard wall function approach, is no longer valid in the presence of roughness. In this study, the real obstruction effect on the wind flow was modeled in terms of equivalent roughness by the roughness wall functions to replace the obstacles applied to the bottom plane of domain.

According to the updated Davenport roughness classification, the aerodynamic roughness z0 was set to be 0.7 m for simulating the high-rise buildings area in this case. The constant static pressure boundary condition was applied at the outlet of the calculation domain. Computations using the ANSYS/Fluent® software were performed to examine the wind field structure characterized by the interaction of airflow with buildings. The computer analysis was based on the steady three-dimensional conservation equations of mass and momentum for the incompressible isothermal turbulent flow. The ambient pressure and temperature were 1atm and 24°C with the density and viscosity of 1.188 kg/m^3 and 1.844×10-5 N-s/m^2, respectively, for the baseline case in Guanyin [24]. From the recorded data of a local meteorological station, the east wind speed of 6.4 m/s in autumn time was used to calculate the neutral ABL velocity, turbulent kinetic energy and turbulence dissipation rate profiles, respectively, as the incoming flow conditions at the inlet of the simulation domain . In essence, roughness increases the drag for the cross flow over the surface. This study modeled the influence of actual obstacles on the crossing wind flow with the roughness wall function setup to simulate the obstacles applied to the bottom surface in the ANSYS/Fluent® software [25], [26]. The aerodynamic roughness was specified to be 0.25 m based on the updated Davenport roughness classification for the state of rough terrain [27].

The constant static pressure boundary condition was used at the outlet of the calculation region. When the top and lateral boundaries were placed far away from the boundary layer, the symmetry boundary conditions were applied by prescribing the zero normal component of velocity and nil normal derivatives for all flow variables at the boundaries. The above mathematical equations were discredited by using the finite control volume approach. An iterative semi-implicit method for pressure-linked equations consistent (SIMPLEC) numerical method was also utilized for velocity-pressure coupling [28]. The mathematical equations were discretized by the finite control volume method. A steady-state flowfield was completed with convergence of the normalized residual errors of flow variables to 10^{-6} and the mass balance check under 0.5% for fully resolving the wind flowfield of indoor and outdoor environments.

IV. RESULTS AND DISCUSSION

Numerical simulations were made with the CFD software ANSYS/Fluent® to investigate the wind field for evaluating ventilation effectiveness of the new administrative building of Guanyin Township. Fig. 3 illustrates the overall mesh system and a locally enlarged view of the numerical grids of the new Guanyin administrative building. From the recorded data of a local meteorological station, a southeast wind with the mean speed of 6.4 m/s in autumn was exploited to calculate the ABL velocity along the x direction in autumn. For the mesh system modeling a representative public building in urbanized area, the averaged cell size was around 0.78 m with the least spacing of 0.027 m to resolve steep variations of flow properties associated with the interaction of airflow with buildings. In this research, numerical computations were performed on the total number grids of 59843268. In this work, we showed the CFD simulation results of outdoor and indoor airflow characteristics to appraise the quality of ventilation.
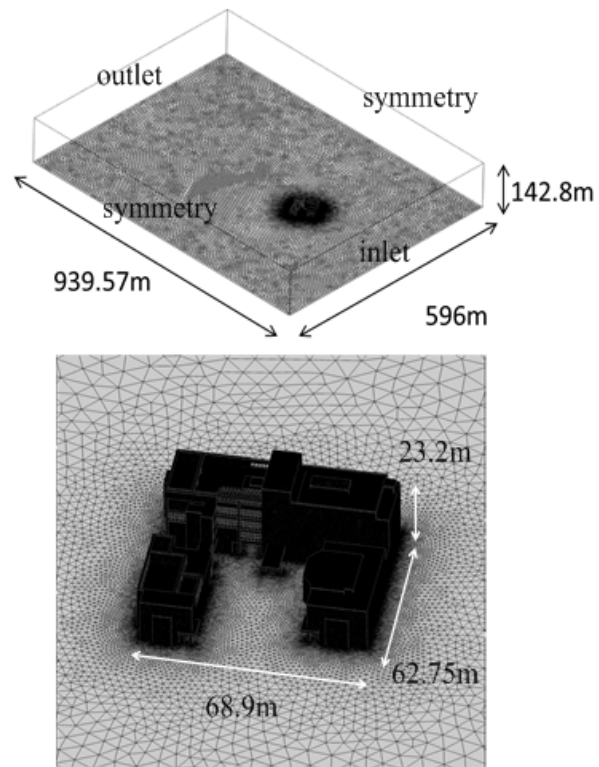

Fig. 3 Overall mesh system and a locally enlarged view of numerical grids of the new Guanyin administrative building

The wind environment adjoining the building was explored by simulating the wind blowing over the administrative building from the east with a speed of 6.4 m/s. To examine the airflow process, Fig. 4 shows the predicted 3D streamlines and velocity magnitude contours in the (a) front view and (b) back view. As the incoming wind from the right (in the east) to the left (in the west), the airflow velocity was reduced owing to the obstruction of the buildings in the windward side. The predictions obviously indicated the appearance of a large recirculating vortex (with comparatively low flow velocities under 0.5 m/s) residing in a mid-open courtyard of three constructions. This swirling flow might draw in and accumulate the contaminants in this area.
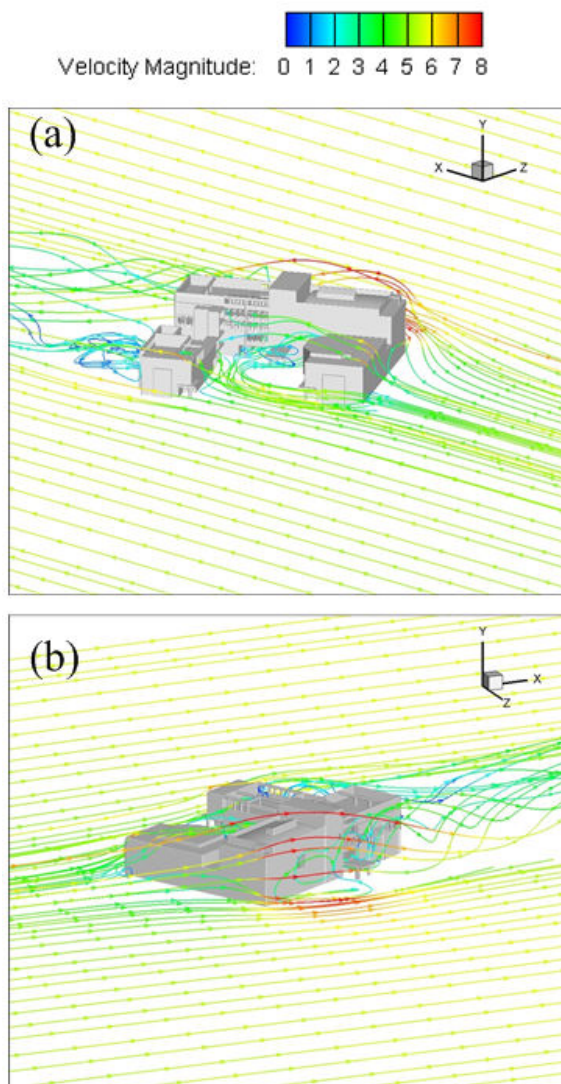

Fig. 4 Predicted 3D streamlines in the (a) front view and (b) back view

Resulting from the adjacent airflow around the buildings, a wake region was evidently observed with the recirculating flow immediately behind the solid structures in the leeward side. From the back view, the predicted 3D streamlines and velocity

magnitude indicated that the ventilated airflow, exhausted from the exit openings in the top and bottom, intermingled into the ambient wind in the back region of the main building. It should be also noted that a part of air stream flowed into the central patio from the gap in the top of the building. To explore the airing process of external airflow into the central patio of the main building, Fig. 5 demonstrates the CFD simulated results of 3D streamlines and velocity magnitude contours for the vertical cross-sectional plane of central patio. In compliance with the architect's design concept, the predicted results plainly revealed that the air flowed into the central patio from the hallway and half-open terrace to form an upward flow at the velocity magnitude varied from 1.5 to 5.0 m/s along the centerline across the central patio. Afterward, the wind moved out of the central patio mainly from the side narrow opening around the hood on the rooftop of the building with a part of the airflow running toward the office areas in the backside of the 3rd and 4th floors. Therefore, it can be deduced that the application of the central patio design to the buildings tends to strengthen the air change flow rate, and consequently enhance the natural ventilation outcome.
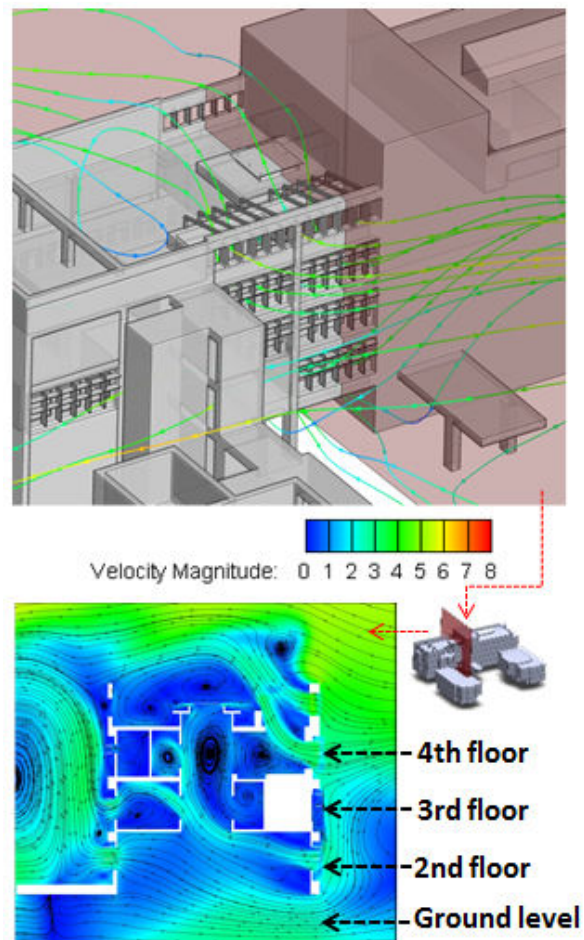

Fig. 5 Predicted 3D streamlines and velocity magnitude contours in the vertical cross-sectional plane of the central patio
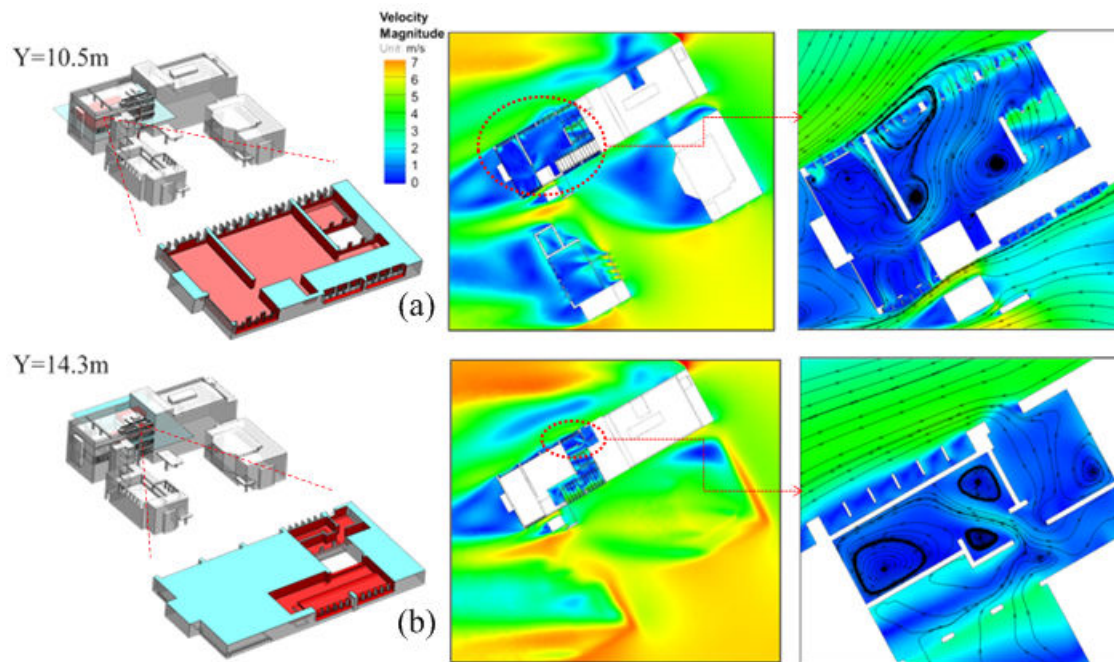
Fig. 6 Predicted velocity magnitude contours and streamlines in horizontal planes at heights of 10.5and 14.3 m

To further investigate the airflow pattern for realization of an effective ventilating mechanism, Fig. 6 shows the CFD simulated results of velocity magnitude and streamlines in the horizontal planes on the 3rd and 4th floors at the heights of (a) 10.5 and (b) 14.3 m, respectively. As presented in Fig. 6 (a), the wind streamed into the office space base with the speeds in the range of 2 to 3 m/s from the central patio with the black arrow denoting the wind direction. The inflow from central patio and windward side windows streams into the middle of the working area with high speeds. This also resulted in lower air velocities at the left and right of the work space, although the airflow can transport the indoor air out and improve air circulation with the outdoor environment.

In contrast, in Fig. 6 (b), it can be visualized that low wind speed areas appeared in most locations due to the blockage of small compartments and walls. Therefore, people tend to feel a little discomfort if they stay those areas longtime. Consequently, adequate natural ventilation is important to improve indoor air quality in the planning and design of public buildings.

V. CONCLUSIONS

In this investigation, we have described a computational framework to investigate the original design concept of natural airing for the administrative building of Guanyin Township with a main focus on probing the building design and its interaction with the indoor and outdoor wind environments. The associated procedure can be also implemented to formulate a CFD-based performance simulator for assessing natural airing effectiveness. The predictions are constructive to better understand the natural ventilation process for managing the indoor microclimate of public buildings.

REFERENCES

[1] Y. Wanga, F. Y. Zhaoc, J. Kuckelkorna, D. Liud, J. Liue, J. L. Zhangf, "Classroom energy efficiency and air environment with displacement natural ventilation in a passive public school building", Energy and Buildings, vol. 70, 2014, p.258-270.

[2] L. Haselbach, "The Engineering Guide to Leed-New Construction: Sustainable Construction for Engineers", New York, McGraw-Hill, 2008.

[3] L. Bellia, F. D. Falco, F. Minichiello, "Effects of solar shading devices on energy requirements of standalone office buildings for Italian climates", Applied Thermal Engineering, Vol. 54, 2013, pp. 190-201.

[4] American Society of Heating, Refrigerating and Air-Conditioning Engineers, "ASHRAE standard: ventilation for acceptable indoor air quality", Atlanta, GA: American Society of Heating, Refrigerating and Air-Conditioning Engineers.

[5] B. Blockena, W. D. Janssena, T. van Hoof, "CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus", Environmental Modelling & Software, vol. 30, 2012, pp.15-34.

[6] M. Dupont, C. Celestine, T. Feuillard, "Natural ventilation in a traditional house on a West Indies Island (Guadeloupe):: Field testing on site and in a wind tunnel", Renewable Energy, vol. 4, 1994, pp.275-281.

[7] A. S. Hussein and H. El-Shishiny, "Influences of wind flow over heritage sites: A case study of the wind environment over the Giza Plateau in Egypt", Environmental Modelling & Software ,vol. 24, 2009, pp. 389-410.

[8] P. Gousseau, B. Blocken, T. Stathopoulos and G. J. F. van Heijst, "CFD simulation of near-field pollutant dispersion on a high-resolution grid: A case study by LES and RANS for a building group in downtown Montreal", Atmospheric Environment, vol. 45, 2011, pp. 428-438.

[9] B. Blocken and J. Persoon, "Pedestrian wind comfort around a large football stadium in an urban environment: CFD simulation alidation and application of the new Dutch wind nuisance standard", Journal of Wind Engineering and Industrial Aerodynamics, vol. 97, 2009, pp.255-270.

[10] G. Gan, S. B. Riffat, "A numerical study of solar chimney for natural ventilation of buildings with heat recovery", Applied Thermal Engineering, vol. 18, 1998, pp. 1171-1187.

[11] S. H. Ho, L. Rosario, and M. M. Rahman, "Comparison of underfloor and overhead air distribution systems in an office environment", Building and Environment, vol. 46, 2011, pp.1415-1427.

[12] T. Catalina, J. Virgone, and F. Kuznik, "Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling", Building and Environment, vol. 44, 2009, pp.1740-1750.

[13] S. Hussain, P. H. Oosthuizen, "Numerical study of buoyancy-driven natural ventilation in a simple three-storey atrium building", International Journal of Sustainable Built Environment, vol. 1, 2012, pp.141-157.

[14] W. Pasut, M. D. Carli, "Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin façade", Applied Thermal Engineering, vol. 37, 2012, pp. 267-274.

[15] M. J. Suárez, C. Sanjuanb, A. J. Gutiérrez a, J. Pistonoa , E. Blanco, "Energy evaluation of an horizontal open joint ventilated façade", Applied Thermal Engineering, vol. 37, 2012, pp. 302-313.

[16] M. Aghakhani and G. Eslami, "Thermal Comfort Assessment of Underfloor vs. Overhead Air Distribution System", Journal of Applied Sciences, vol. 12, 2012, pp. 473-479.

[17] J.C. Teixeira, R Lomba1, S.F.C.F. Teixeira, and P. Lobarinhas, "Application of CFD Tools to Optimize Natural Building Ventilation Design", Computational Science and Its Applications ICCSA 2012, Part III, LNCS 7335, pp. 202–216.

[18] N. H. Wong, S. Heryanto, ''The study of active stack effect to enhance natural ventilation using wind tunnel and computational fluid dynamics (CFD) simulations,'' Energy and Buildings, vol.36 (7), 2004, pp.668–678.

[19] C. Ghiaus, F. Allard, Natural ventilation in the urban environment: assessment and design, Earth scan, London, 2005.

[20] P. J. Richards, Computational modeling of wind flows around low rise buildings using PHOENIX. Report for the ARFC Institute of Engineering Research Wrest Park, Silsoe Research Institute, Bedfordshire, 1989.

[21] P. J. Richards, R. P. Hoxey, , "Appropriate boundary conditions for computational wind engineering models using the k-E turbulence model", Journal of Wind Engineering and Industrial Aerodynamics, vol. 46 & 47, 1993, pp.145-153.

[22] R. M. Harrison, ''Understanding Our Environment: An Introduction to Environmental Chemistry and Pollution'', Royal Society of Chemistry, Cambridge, 1992.

[23] B. Blocken, T. Stathopoulos, J. Carmeliet, ''CFD simulation of the atmospheric boundary layer: wall function problems,'' Atmospheric Environment, vol. 41, 2007, pp 238-252.

[24] World Meteorological Organization, Weather Information for Noumea, 2010, http://worldweather.wmo.int/1 40/c00296.htm.

[25] Y. Ji, M. J. Cook, and V. Hanby, ''CFD modelling of natural displacement ventilation inan enclosure connected to an atrium,'' Building and Environment, vol. 42, 2007, pp. 1158–1172.

[26] O. S. Asfour, and M. B. Gadi, ''A comparison between CFD and Network models for predicting wind-driven ventilation in buildings,'' Building and Environment, vol. 42(12), 2007, pp. 4079–4085.

[27] J.Wieringa, ''Updating the Davenport roughness classification,'' Journal of Wind Engineering and Industrial Aerodynamics, vol. 41–44, 1992, pp. 357–368.

[28] D.S. Jang, R. Jetli, S. Acharya, ''Comparison of the PISO, SIMPLER, and SIMPLEC algorithms for the treatment of the pressure-velocity coupling in steady flow problems'', Numerical Heat Transfer, vol. 10, 1986 pp.209-228.

An-Shik Yang received his Bachelor of Science (1982) and Master of Science (1984) degrees from the National Tsing Hua University in Taiwan, and Ph.D. (1993) degree from the Pennsylvania State University in USA. He joined the Dept. of Energy and Refrigerating Air-Conditioning Engineering at National Taipei University of Technology (NTUT) as an associate professor in fall 2007, and was promoted to full professor in spring 2010. Dr. Yang is an Associate Fellow of American Institute of Aeronautics and Astronautics (AIAA). His research interest is in the areas of environmental fluid mechanics, multiphase fluid dynamics, and heat transfer.

Chiang-Ho Cheng joins the Department of Mechanical and Automation Engineering at Dayeh University as an associate professor on February 1999, and was promoted to full professor in fall 2012. He earned his Ph.D. degree in Institute of Applied Mechanics from National Taiwan University in 1996. His research interests include the MEMS system, the design and analysis of piezoelectric droplet inkjets and micropumps.

Jen-Hao Wu received his bachelor degree in department of mechanical and computer-aided engineering from the Feng Chia University in 2012 and is currently pursuing the master degree in department of energy and refrigerating air-conditioning engineering from the National Taipei University of Technology, in Taipei, Taiwan R.O.C. His main research interests include the fluid dynamics, heat transfer and numerical simulation analysis.

Yu-Hsuan Juan received her bachelor degree in department of aerospace and systems engineering from the Feng Chia University in 2011 and got her master degree in department of energy and refrigerating air-conditioning engineering from National Taipei University of Technology in 2013. She is currently working as research assistant on the National Taipei University of Technology. Her main research interests include the fluid dynamics of Environmental, heat transfer and numerical simulation analysis of microclimate.