Wind Field Analysis for a High-rise Residential Building Layout in Danhai, Taiwan

An-Shik Yang, Chih-Yung Wen, Yu-Chou Wu, Yu-Hsuan Juan, and Ying-Ming Su

Abstract—**The present study shows the ground work results from the computational fluid dynamics (CFD) analysis for the residential new town Danhai in the northern Taiwan. As a new approach in urban and community planning and design subjects, we performed the simulations to obtain numerical predictions of flow characteristics around the buildings for exploring the wind field of the Danhai new town. In the analysis, the dimensions of calculation domain were 3 km long, 2 km wide and 0.6 km high. We considered the incompressible isothermal turbulent flow over the high-rise residential buildings to probe the interaction of airflow with buildings as well as to better understand the effect of the contiguous design on the urban ventilation outcome. A modified model was also proposed through removal of a single building in the windward of community to enlarge the wind pathway. The simulated results evidently indicated that the improved design significantly enhanced ventilation with a comfortable wind environment achieved in the central open space for residents.**

Keywords— **high-rise residential building, Layout of neighborhood, Danhai, Computational fluid dynamics**

I. Introduction

The studies of urban environment have become more and more important in the past decades because of the rapid development of urbanization in industrial countries. Many computational fluid dynamics (CFD) studies were performed to investigate the wind field characteristics within a building complex [1] and around high-rise buildings at a pedestrian level [2-3]. Prevailing winds in urban areas can be substantially modified by the increasing number of closely placed high-rise buildings, leading to a great change of the natural ventilation behavior [4]. In addition, air inside a building needs to be replenished recurrently. Without a supply of new air, sources of irritation and allergy in the air, such as dust or hair, can adversely influence human health. Hence, it has been noticed as an important issue to bring the fresh cool air from outside into a building and drive the stale warm air out via an outlet. Kotani et al. [5] and Chen [6] have conducted extensive reviews on natural ventilation and cross ventilation in planning and designing the high-rise buildings. Ayata and Yildiz [7] indicated that the performance of natural ventilation could be markedly improved by means of the proper arrangement of building orientation with respect to the prevailing wind. Chow [8] demonstrated that the openings of buildings and incident wind could be employed for improving the natural ventilation in high-rise apartments under different ventilation conditions. Blocken and Carmeliet [9] analyzed the pedestrian wind environment around buildings, and showed that the high wind speed introduced by high-rise buildings can result in uncomfortable or even dangerous conditions at the pedestrian level [10]. The terrain roughness effect can also play an important role in the outcome of wind comfort studies.

Orientations of the building and ventilation openings have to be designed with respect to the climate and topography conditions on a wind environment, while the CFD predictions should be carefully taken into account in the layout of high-rise buildings with the key focused aspects including the pedestrian wind environment, natural ventilation, air movement and safety/comfort issues in a living space. The researches were performed using the CFD tools to study the flow phenomenon in the buildings having an atrium [11, 12]. The high-rise residential buildings with atriums have large space for residents' activities and show different flow processes in nature, as compared to the common buildings. It is crucial to remove the pollution and heat at the top of an atrium by wind and drive air movement through an atrium well of buildings. Mouriki et al. [13] and Hussain and Oosthuizen [14] reported the comparisons of CFD predictions and experimental measurements for the atrium-type buildings. Taking various court dimensions into consideration, Kotani [15] surveyed the questionnaire for a light well of high-rise apartment buildings to study the evaluation of occupants on the environment. Holland [16] explored the effects of the external wind speed and direction on the air change rate of a two-story building in a naturally ventilated atrium space.

In recent times, it has become increasingly important to perform the pre-evaluation and design simulations in the planning and development of new towns. This CFD-based analysis, from the fluidic view, investigates the interaction of wind field with structures of Danhai New Town, located in northern Taiwan, for improving microclimate around the building cluster. To explore the airflow characteristics, we examined the distributions of streamwise, crosswise velocity components and the pressure over the plane at a height of 1.5 m around the buildings. The simulated results can be used to assess the sufficiency of natural ventilation for the central open space of community. In order to generate the suitable environments of natural ventilation for residents, an adequate

Manuscript received January 4, 2013; revised March 6, 2013. This work was supported in part by the National Science Council, Taiwan, ROC under Grant NSC101-2627-E-027-001-MY3.

An-Shik Yang and Yu-Hsuan Juan are with the Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology, Taipei, 106, Taiwan (phone: (886) 2-2771-2171 ext. 3523; fax: (886) 2- 2731-4919; e-mail: asyang@ntut.edu.tw).

Chih-Yung Wen is with the Department of Mechanical Engineering, The Hong Kong Polytechnic University, Kowloon, Hong Kong. (e-mail: chihyung.wen@polyu.edu.hk).

Ying-Ming Su and Yu-Chou Wu are with the Department of Architecture, National Taipei University of Technology, Taipei 106, Taiwan. (e-mail: ymsu@ntut.edu.tw).

transverse opening must be left with least obstacles along the wind pathway to allow the airflow passing into the desired central open space of the neighborhood. A modified layout of neighborhood was proposed to remove a single building in the windward for enhancement of urban ventilation. Furthermore, this study can help the planners and strategy makers to better understand the interaction between buildings and neighborhood for realizing an optimal urban environment.

II. Description of Danhai New Town planning

The construction of Danhai New Town was launched by the Construction and Planning Agency (CPA) of the Ministry of Interior Affairs of Taiwan in 1992. The objective at that time was to alleviate the tension in the growth of Taipei Metropolis, preserve land, solve the problem of insufficient housing in two neighboring metropolises, and establish a high standard in the city development process [17]. Danhai New Town is located 16 km northeast of the central Taipei City. The plan of the new town is to implement a grid system of the road network for distribution of buildings and facilities. The main basic concepts are to make best use of local and peripheral special resources, plan a detailed framework in which pedestrians and vehicles have dedicated lanes, construct a world-class fundamental infrastructure with a computerized monitoring system, and produce a high-quality living space and good city landscape. According to the official source, the total area to be developed is 1756 ha on which it is designed for accommodating 240,000 people to live and various supporting facilities. Danhai New Town is situated between hills and waters to take advantage of its natural features, and is very attractive in terms of mountain and sea travel for Taiwan's coastal recreation.

CPA has conducted orientation of the Danhai New Town planning with evaluation and adjustment of development strategies completed since 2002. The future Danhai New Town will concentrate on developing high-quality living, coastal business and recreation, and medical-care concerns. Hence, Taiwan's government and people have great concern in the urban and building development of Danhai in view of the fact that it practices as the blueprint for the modern new towns to Taiwan in this new century. Since the new town location is close to the coastline, it is expected that the wind can substantially influence the micro-climate near high-rise residential buildings. Considering different stages of development in Danhai, Fig. 1 illustrates (A) the original planning concept of Danhai New Town in 1992, (B) the top view of the central area of Danhai in 2009, (C) the current development of Danhai in 2012, (D) a typical high-rise residential building in Danhai, and (E) the zoning map and the location of simulated high-rise buildings in Danhai. In this research, we conducted the CFD analysis for the above representative high-rise building clusters and lower level facilities in accordance with the urban design guideline.

III. Computational analysis

To conduct the computational analysis for evaluating the layout of the residential buildings to be constructed, the physical model considers the environmental wind flowing over the high-rise buildings in Danhai New Town. The inlet boundary condition in CFD computations of the atmospheric

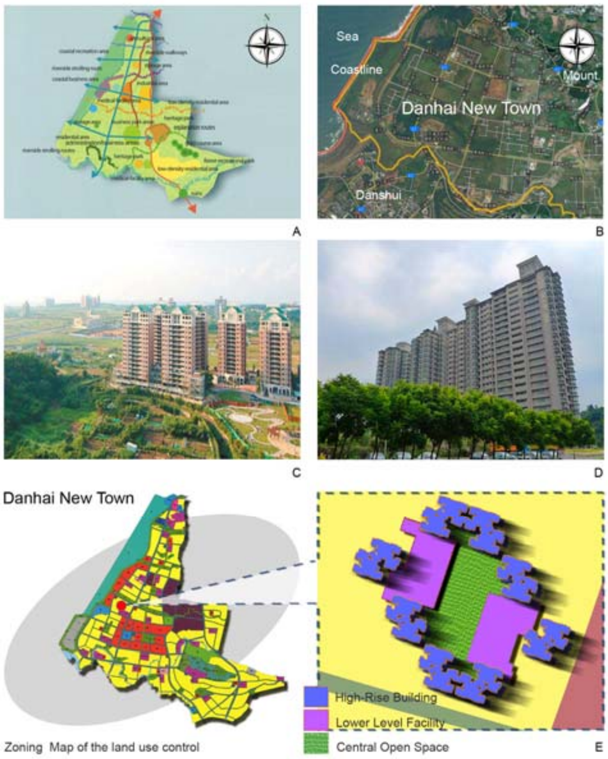

Fig. 1 (A) Original planning concept of Danhai New Town in 1992, (B) Top view of central area of Danhai in 2009, (C) Current development of Danhai in 2012, (D) A typical high-rise residential building in Danhai, and (E) Zoning map and the location of simulated high-rise buildings in Danhai. (Sources adapted from CPA, 2006, Google Earth, 2009, and New Taipei City government, 2012.)

boundary layer (ABL) flow was used to model the associated atmospheric processes [18]. 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 three-dimensional conservation equations of mass and momentum for the incompressible isothermal turbulent airflow over the calculation domain [19]. The governing equations are stated as follows:

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

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

In the above 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 gravitational force, respectively. Considered as the most popular, well-established and widely tested turbulence model, a standard k-ε two-equation turbulent model [20] was adopted for turbulence closure, as follows:

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

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

The production term is given as

ISBN: 978-988-19252-8-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

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

Here μ_t is $C_\mu \rho k^2/\varepsilon$; k, the turbulent kinetic energy; and ε, the turbulent energy dissipation rate. As a common practice, the constants C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and σ_ε were 0.09, 1.44, 1.92, 1.0 and 1.3, respectively. In this study, the ambient pressure was 1 atm. A zero normal-pressure gradient condition was imposed on the solid wall surface, k and ε profiles, respectively, as the incoming wind flow condition [21].

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

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

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

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

The signs z_0 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 U_h at a reference height h. The sign U_{ABL}, the mean inlet velocity at the height of z, can be obtained via Eq. (7) to generate a velocity profile within the ABL with the turbulence kinetic energy and dissipation rate computed via Eqs (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 [22, 23]. According to the updated Davenport roughness classification [24], the aerodynamic roughness z_0 was set to be 0.1 m for simulating the peripheral areas of the high-rise buildings in Danhai as roughly open terrain. The constant static pressure boundary condition was also applied at the outlet of the calculation domain. When the top and lateral boundaries were placed far away from the boundary layer, the symmetry boundary conditions were used 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 the finite control volume approach. An iterative semi-implicit method for pressure-linked equations consistent (SIMPLEC) [25] numerical method was adopted for velocity-pressure coupling. In this study, an accurate steady-state flowfield was obtained with convergence of the normalized residual errors of flow variables to 10^{-6} and the mass balance check under 0.5% for establishment of the wind flowfield of indoor and outdoor environments.

IV. Results and Discussion

Simulations were conducted through solving the interaction of airflow with buildings to examine the wind field at the pedestrian level for probing the influence of the neighboring design on the urban ventilation effect and improving the microclimate around the residential area. Figure 2 illustrates the mesh system for a typical high-rise building setup in Danhai, Taiwan. The mesh system comprised two main sections, including the neighborhood domain formed by 8 high-rise residential buildings and 2 lower level facilities as well as the surrounding domain (simplified as a flat surface) around the site. The averaged cell size was around 0.45m near the high-rise building with the smallest spacing of 0.03m to resolve steep variations of flow properties. Computations were performed on the total number grids of 7296469, 9777268, and 12710448. The numerical predictions of the flow velocities along the perimeters and vertical centerlines of eight high-rise residential buildings at three different grids suggested that the satisfactory grid independence can be achieved by a mesh setup of 9777268 grids.

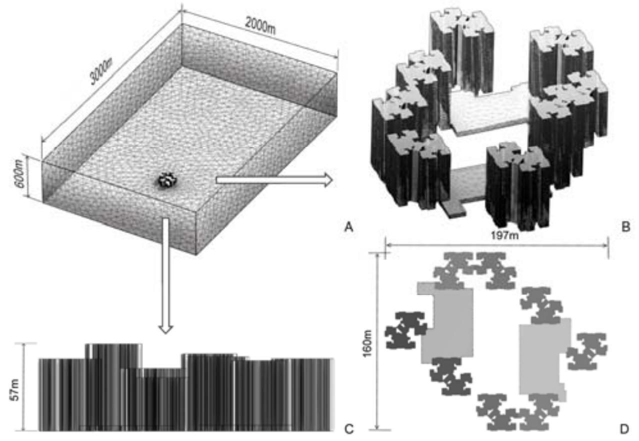

Fig. 2 Mesh systems for a typical high-rise building setup in Danhai

It generally requires 240 hours of central processing unit (CPU) time to acquire a converged steady-state solution on an Intel®Core™i7X900-3.47GHz (24.0GB RAM) personal computer. In this paper, we present the CFD simulated results of outdoor flowfield for the wind across the Danhai high-rise residential buildings. In calculations, the density and viscosity were 1.19 kg/m3 and 1.84× 10^{-5} N-s/m^2 at the annual mean temperature of 29ºC [26] in Danshui for the baseline case. From the recorded data of a local meteorological station, the annual mean wind speed of 3m/s along the x direction was used to calculate the ABL velocity. Figure 3 shows the predicted streamlines, streamwise velocity magnitude and pressure contours of wind field for the incoming ABL flow from the sea. In Fig. 3-A, B, the neighborhood mainly consisted of two parts: the high-rise buildings and low-level facilities with two sections at the heights of 1.5 and 5m, respectively. The central open space was enfolded by residential units for recreation or children playgrounds of the community. The wind effect on the residents was then investigated via the current layout and building design with the wind flowing through the spacing of the high buildings to low level facilities, and then exhausting through the other side of the neighborhood. In Fig.3-C, we can observe the low wind velocity of about 1m/s (light blue color) in the 1.5m pedestrian level of the central open space.

ISBN: 978-988-19252-8-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

The high pressure areas of about 101.33kpa (marked in an orange color) were also noted in the windward of the first high-rise building demonstrated in Fig 3-D. For the section level at a height of 5m in Fig3-E, F, the wind velocity in the central area of community is up to 2m/s due to disappearance of lower facilities.

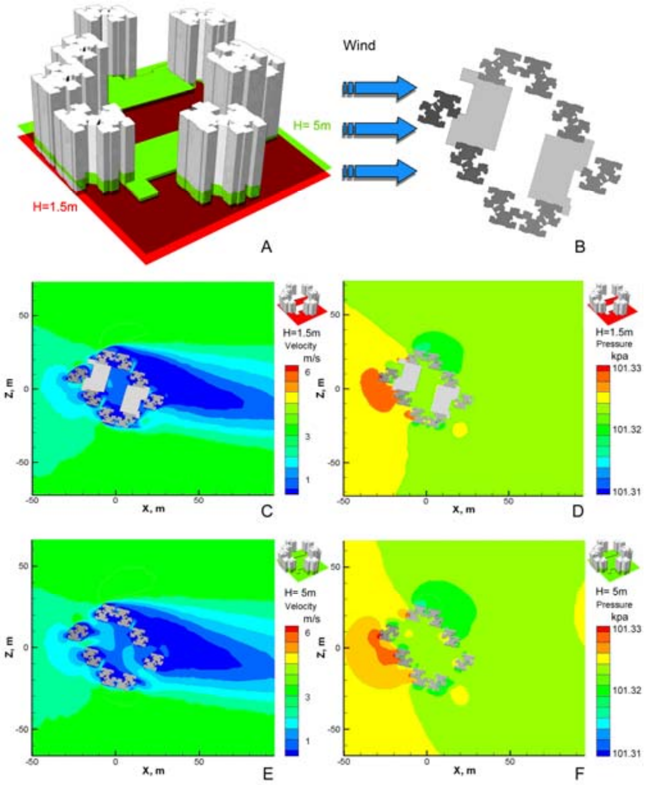

Fig. 3 Predicted streamwise velocity magnitude and pressure contours of wind field for the incoming ABL flow from the front of community

Figure 4 illustrates the close-up views of the predicted axial velocity and pressure contours of wind field in the windward and leeward sides of the buildings. Considering the wind coming from the front of the neighborhood at a height of 1.5m as shown in Fig.4-A, It was observed that the wind velocity was decreased dramatically when the flow was approaching the solid buildings with the velocity magnitudes reduced from 3m/s (in green color) to 1m/s (in light blue color). We can identity some recirculating and low wind speed areas in the back of buildings especially in the central open space. Figure 4-B demonstrates the high pressure areas (the orange bubble) in the windward of this neighborhood. In Fig.4-C, the low wind velocity was below 1m/s (in deep blue color) with the free stream in the leeward of high-rise buildings. Conversely, the pressure distribution was more even in the leeward of the first high-rise building exhibited in Fig4-D.

Figure 5-A-D illustrates the predicted crosswise velocity and pressure contours of wind field at different cross-sections across the central open space of the community. Figure 5-B shows a very low-speed area (in a deep blue color) with the crosswise velocity magnitude under 1m/s occurred in the middle between the two high-rise buildings attributable to the hindering influence of the first single high-rise building in the windward. Figures 5-C and D show the transverse velocity magnitude in the middle of the neighborhood. The streamlines pass over the lower facility with a low-speed wake flow (in light blue color) formed behind the tall one. A massive decreasing (in both light and deep blue color) velocity magnitude in the right side of the last high-rise building, quiet obviously, due to the presence of other buildings as obstacles at the wind pathway.

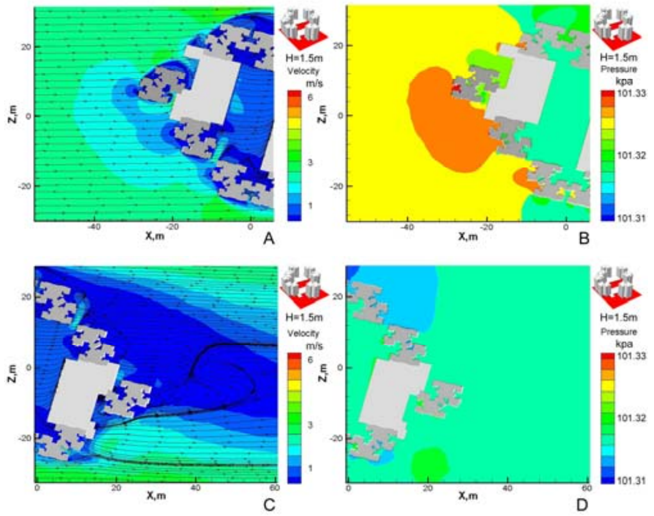

Fig. 4 Close-up views of predicted axial velocity and pressure contours of wind field in the windward and leeward sides of the buildings

Figure 6-A-D demonstrates the predicted streamwise velocity and pressure contours of wind field across the central open space. The local flow separation appeared at the leading edge of the top of lower facility, while the free-stream flow was recovered in the area beyond the separated shear layer. A low-speed vortex developed between two buildings in Fig. 6-B indicated that the spacing of buildings is an important factor to the wind effect on the layout planning. The wind speed in the central open space was less than 1m/s due to the blockage of the both high-rise and lower buildings in Fig. 6-C. The urban ventilation is highly influenced by the layout of high density and much closer spacing of high-rise buildings. A large low-speed area (less than 1m/s and marked in deep blue color) were also noted behind the high-rise building in Fig. 6-D.

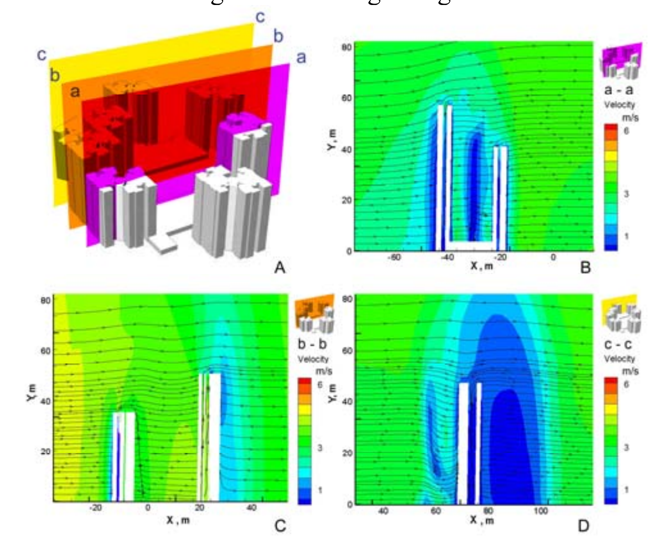

Fig. 5 Predicted crosswise velocity and pressure contours of wind field across the central open space for the incoming ABL flow from the front of the community

To deal with the insufficient urban ventilation in Danhai, Fig. 7 illustrates the predicted streamlines, axial velocity magnitude and pressure contours of wind field for enhancement of urban ventilation over the new neighborhood. As shown in Fig. 7-A, the obstacle (i.e. the first high-rise building) at the front of the wind pathway was eliminated with 7 buildings left in total. A larger opening was formed to

ISBN: 978-988-19252-8-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

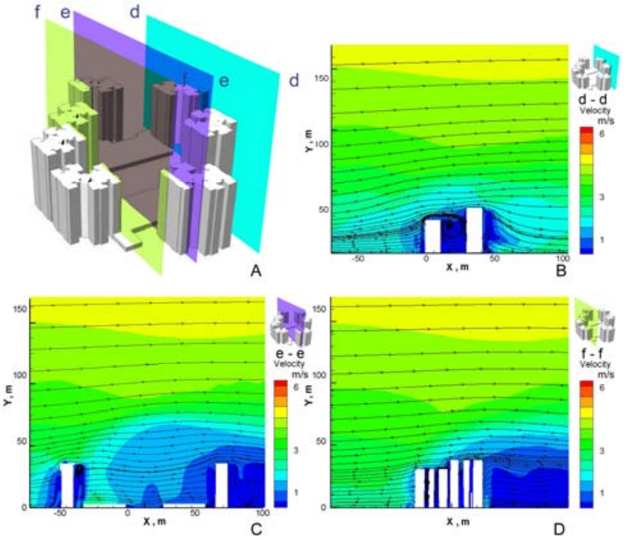

Fig. 6 Predicted streamlines, streamwise velocity magnitude and pressure contours of wind field across the central open space for the incoming ABL flow from the front of the community

induce the wind toward the central open space for enhancing the ventilation. In Fig. 7-B, the stereo streamlines at a height of 1.5m demonstrate the wind passing through the neighborhood. The wind speed was also increased to approximately 1.5m/s (in light blue color) in central open space as illustrated in Fig. 7-C. As compare to Fig 3-c, the low-speed wind area was reduced via removing a single building. In Fig.7-D, we can observe that the high pressure areas become smaller and roughly divided into two sectors. For the plane at a height of 5m in Fig7-E, the wind flowed into the central part of the community much smoother with the velocity increased up to 2.5m/s in the central area of community due to withdrawal of the first high-rise building. In Fig7-F, the high pressure concentration reduced and the effect areas are become smaller, as compare to Fig. 3-F. In

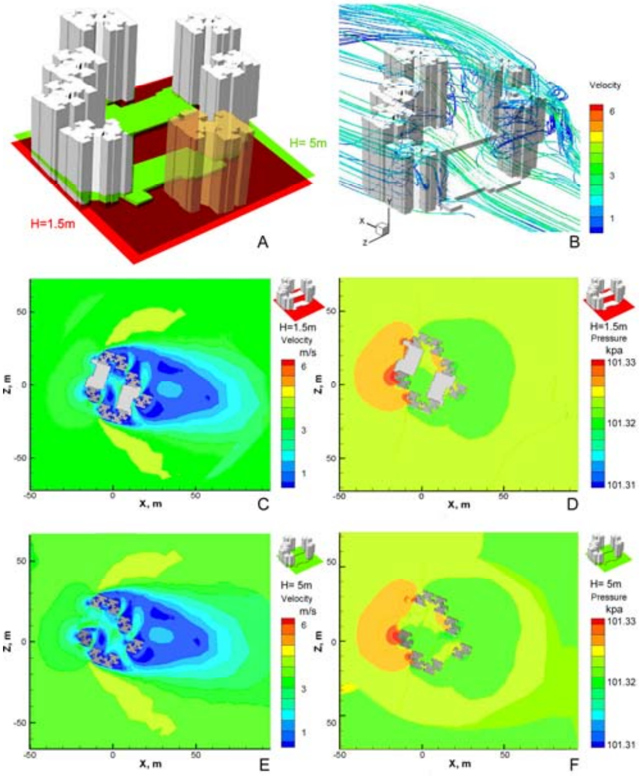

Fig. 7 Predicted streamlines, axial velocity magnitude and pressure contours of wind field for enhancement of urban ventilation for the incoming ABL flow from the front of the improved neighborhood layout in Danhai

brief, the benefit from changing the building layout is quite evident in strengthening the ventilation mechanism for achieving a better residential living environment.

According to the simulation results of new layout, Figure 8 illustrates the Close-up views of predicted streamlines, axial velocity magnitude and pressure contours of wind field in the windward and leeward sides. In comparison with Fig. 4-A, ventilation in the central open space was improved at the pedestrian level (in light blue color). The sizes of the recirculating and low-speed areas in the wake of buildings became smaller or even invisible. In Fig. 8-B, the high pressure areas (the orange bubble) in the windward were observed as two smaller parts in the two wings. In Fig.8-C, the low-wind velocity areas in the leeward of high-rise buildings are also reduced..

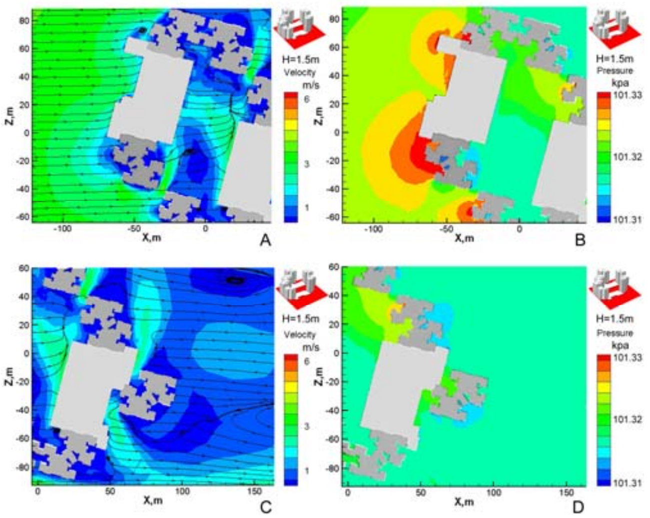

Fig. 8 Close-up views of predicted streamlines, axial velocity magnitude and pressure contours of wind field in the windward and leeward sides for the incoming ABL flow from the front of the improved layout in Danhai

In Fig. 9, removing the first single high-rise building in the windward tended to enlarge the inflow area of the wind with disappearance of those low-speed areas (less than 1m/s and in a deep blue color) in the middle between two high-rise buildings. Alternatively, the streamlines passed through the lower facility and two low-speed areas (in light blue color) formed behind buildings. Figure 9-D clearly indicated a substantial decrease in the low-speed areas in the right side of the last high-rise building, as compared to Fig. 5-D.

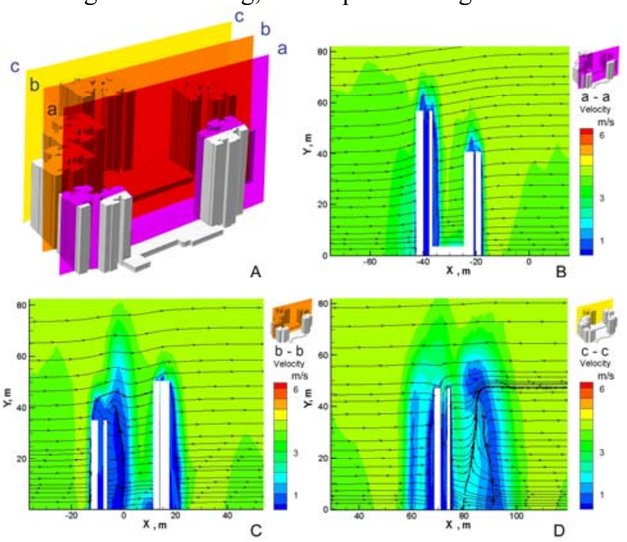

Fig. 9 Predicted streamlines, crosswise velocity magnitude and pressure contours of wind field across the central open space for the incoming ABL flow from the front of the improved layout in Danhai

ISBN: 978-988-19252-8-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

V. Conclusions

The present study has described a computational framework to investigate the CPA design concept in Danhai New Town, with a main focus on examining the building geometric layout and its interaction with the urban wind environments. Applying the CFD as an effective tool to simulate the urban wind flowing across the neighborhood, city planners can better understand a conceivable physical environment of the urban areas with the predicted streamlines, velocity and pressure distribution at the pedestrian level. The micro-climatic conditions of the city can also be characterized before the development and settlement of design strategies. Adjusting and reshaping the layout of the buildings over the neighborhood based on the feedback of the simulated results can be much effective and convincible. This research intends to integrate the detailed analysis via the CFD-based simulations serving as a digital wind tunnel with the urban planners' concept from realistic physical sense for design an optimal city. The urban ventilation process for the wind over the Danhai New Town has been examined in this study with the major results summarized as below:

(1) In response to insufficient frontal area for directing the airflow into the central open space of community, the improved design for strengthening the effectiveness of urban ventilation at the normal wind indicated a significant growth in the central open space from less than 1m/s to 2.5 m/s via an increase of the aeration area in conjunction with removal of a single high-rise building along the flow pathway.

(2) The spacing between the buildings is an important factor to control the wind field for layout planning. The urban ventilation is highly influenced via the layout of density and spacing of high-rise buildings. Though the design of the Danhai New Town may not be perfect from the fluidic vision, we can conclude with certain that it is quite promising to apply the CFD approach for developing the design strategy of great city in the future.

Acknowledgment

This paper represents part of the results obtained under the support of the National Science Council, Taiwan, ROC (Contract No. NSC101-2627-E-027-001-MY3).

References

[1] R.Yoshie, G. Jiang, T. Shirasaw, J. Chung,CFD simulations of gas dispersion around high-rise building in non-isothermal boundary layer, Journal of Wind Engineering and Industrial Aerodynamics, vol. 99, 2011, pp.279-288.

[2] Y.Tominaga, A. Mochida, T.Shirasawa, R.Yoshie, H.Kataoka, K.Harimoto, T. Nozu, Cross comparisons of CFD results of wind environment at pedestrian level around a high-rise building and within a building complex, Journal of Asian Architecture and Building Engineering , vol. 3, 2004, pp.63-70.

[3] R. Yoshie, A. Mochida, Y. Tominaga, H. Kataoka, K. Harimoto, T. Nozu, T. Shirasawa, Cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan, Journal of Wind Engineering and Industrial Aerodynamics, vol..95, 2007, pp.1551–1578.

[4] J.O.P. Cheung, C.H.Liu, CFD simulations of natural ventilation behavior in high-rise buildings in regular and staggered arrangements at various spacings, Energy and Buildings, vol. 95, 2011, pp. 1149–1158.

[5] H. Kotani, T. Goto, M. Ohba, T. Kurabuchi, Review of cross-ventilation research papers—from the Working Group for Natural Ventilation and Cross-Ventilation of the Architectural Institute of Japan, International Journal of Ventilation, vol. 8, 2009, pp. 233–241.

[6] Q. Chen, Ventilation performance prediction for buildings: A method overview and recent applications, Building and Environment, vol. 44, 2009, pp. 848–858.

[7] T. Ayata, O. Yıldız, Investigating the potential use of natural ventilation in new building designs in Turkey, Energy Building, vol. 38 ,2006, pp.959–963.

[8] W.K. Chow, Wind-induced indoor-air flow in a high-rise building adjacent to a vertical wall, Applied Energy, vol. 77, 2004, pp. 225–234.

[9] B. Blocken, J. Persoon, Pedestrian wind environment around buildings: Literature review and practical examples, Journal of Thermal Envelope and Building Science, vol. 28, 2004, pp.107-159.

[10] B. Blocken, J. Persoon, Pedestrian wind conditions at outdoor platforms in a high-rise apartment building: generic sub-configuration validation, wind comfort assessment and uncertainty issues, Wind and Structures, vol.11, 2008, pp.51-70.

[11] X.Wang, C.Huang, W. Cao, Mathematical modeling and experimental study on vertical temperature distribution of hybrid ventilation in an atrium building, Energy and Buildings, vol. 41, 2009, pp. 907–914

[12] T. Hiramatsu, T. Harada, S. Kato, S. Murakami, H. Yoshino, Analysis of indoor thermal environment with cooling condition and natural ventilation – Study of thermal environment in real-scale atrium Part3-, Journal of Architecture Planning and Environment Engineering, AIJ, vol. 74, 1998, pp. 23-30.

[13] E.Mouriki, P.Karava, A.Athienitis, KW Park, T. Stathopoulos, Full-scale study of an atrium integrated with a hybrid ventilation system, 3rd Canadian Annual Solar Buildings Conference, Fredericton, Canada, 2008.

[14] S. Hussain, P.H. Oosthuizen, Validation of numerical modeling of conditions in an atrium space with a hybrid ventilation system, Building and Environment, vol. 52, 2012,pp.152-161.

[15] H.Kotani, M.Narasaki, R. Sato, T. Yamanaka, Environmental assessment of light well in high-rise apartment buildings, Building and Environment, vol. 52, 2012, pp.152-161.

[16] J.M. Horan, D.P. Finn, Sensitivity of air change rates in a naturally ventilated atrium space subject to variations in external wind speed and direction, Energy and Buildings, vol. 40, 2008, pp.1577–1585.

[17] Construction and Planning Agency, Ministry of the Interior. Taiwan, R.O.C., http://www.cpami.gov.tw/chinese/index.php?option=com_content&view=article&id=14707&Itemid=50, 2012.

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

[19] 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, UK., 1989.

[20] 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.

[21] B. Blocken, T. Stathopoulos, J. Carmeliet, CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric Environment, vol. 41, 2007, pp.238-252.

[22] D.M. Hargreaves, N.G. Wright, On the use of the k–ε model in commercial CFD software to model the neutral atmospheric boundary layer, Journal of Wind Engineering and Industrial Aerodynamics, vol. 95, 2007, pp.355–369.

[23] J.P. Van Doormaal, G.D. Raithby, Enhancements of the SIMPLE method for predicting incompressible fluid flows, Numerical Heat Transfer, vol. 7, 1984, pp.147-163.

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

[25] 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.

[26] Central Weather Bureau of Taiwan, Weather Information for Danshui, http://www.cwb.gov.tw/V7/observe/real/46690.htm, 2012.