

## SIMULATING NATURALLY VENTILATED BUILDINGS WITH DETAILED CFD-BASED WIND PRESSURE DATABASE

Bing Wang<sup>1</sup>, Timur Dogan<sup>1</sup>, Debashree Pal<sup>1</sup> and Christoph Reinhart<sup>2</sup>

<sup>1</sup>Graduate School of Design, Harvard University, Cambridge, MA

<sup>2</sup>Department of Architecture, Massachusetts Institute of Technology, Cambridge, MA

### ABSTRACT

Natural ventilation has long been recognized as a sustainable design strategy to replace or reduce energy associated with fans or active cooling systems in buildings. When it comes to designing a passive system based concept that involves natural ventilation, a major difficulty for design teams is to reliably predict indoor ventilation rates especially in buildings located in a dense urban context. This manuscript proposes a new concept to model ventilation rates and hourly annual indoor temperature distributions using a combination of outside CFD (Computational Fluid Dynamics) calculations and indoor airflow network calculations. In the first step, CFD calculations for eight cardinal wind directions are used to populate a custom database of pressure coefficients for each window in an urban scene. In the second step, hourly wind data and pressure coefficients method have been implemented in Rhinoceros<sup>1</sup> in combination with Phoenix<sup>2</sup> and EnergyPlus<sup>3</sup>. Example results are presented for several typical cases, indicating the improvement of this workflow on simulation results, especially for building with particular shape and urban context.

### INTRODUCTION

Buildings account for 40% of the energy used in the United States (James W.A.). Similar numbers can be found for other countries cross Europe and Asia. Natural ventilation has been considered viable techniques to replace or reduce active cooling loads and fan power in residential and commercial buildings alike. (Liddament M.W. 1986; Hagentoft C. 1996.) There is further mounting evidence that occupants in naturally ventilated buildings tend to accept higher indoor temperatures than air-conditioned spaces. This

increased tolerance further increases the potential of energy-saving through natural ventilation. (Busch J.F. 1992.) Prominent modern buildings that make use of natural ventilation are the Jubilee Campus of the University of Nottingham, UK and the Commerzbank Tower in Frankfurt a.M. in Germany.

Despite of such examples the diffusion of natural ventilation, especially in US buildings, where air conditioning tends to be the norm, the application of natural ventilation is rare. Partly due to planning uncertainties. It is not easy for a design team and/or building owner to be confident about an impeccably functioning natural ventilation system. Three hurdles come to mind; how are occupants going to operate windows and doors, what are the resulting air exchanges and are occupants going to accept the resulting indoor temperature and relative humidity levels? This manuscript is particularly concerned with predicting air exchanges and annual indoor temperature profiles in buildings located in urban environments.

To tackle the problems, a range of methodologies are used in practice; experiments, measurements, simplified formulas, multi-zone airflow network calculations and CFD (Computational Fluid Dynamics). Out of the theoretical methods, CFD is considered the most complex and time consuming but also the most flexible technique. It can be used to predict detailed wind environments around buildings. However, the process is time-consuming and only a few typical wind situations, out of all possible cases, can practically be considered. This is why CFD is more commonly used to test the thermal behavior of a building under extreme conditions, say a hot summer afternoon. This practice is considered suboptimal because a designer needs to understand how a building performs all year round and how often occupants might be dissatisfied with interior conditions. Simplified simulation methods usually do not consider actual exterior microclimate conditions, especially for space near urban canyons, and are therefore not satisfactory because the simulations cannot necessarily be expected to be accurate.

<sup>1</sup> Rhinoceros, a widely used NURBS (Non-uniform rational B-spline) modeling program for designers.

<sup>2</sup> Phoenix, a commercial CFD (Computational Fluid Dynamics) software used to simulate outdoor environment.

<sup>3</sup> EnergyPlus, a building energy simulation program for engineers, architects, and researchers.

This study aims to achieve more precise evaluation of a potential for natural ventilation by developing a workflow of annual simulation of natural ventilation that is based on detailed CFD-simulated wind environment database on site. Several case studies compare simulation results of the new method and the existing methods, showing the advance of this workflow.

## METHODOLOGY

### Wind Pressure Coefficient

Wind is an important driving force for infiltration and ventilation because wind causes variable surface pressures on buildings that change intake and exhaust system flow rates, natural ventilation, infiltration and exfiltration, and interior pressures. Wind pressure is therefore an important boundary condition for calculation of natural ventilation. Wind pressure is used in a wide range of models, from building component heat, air and moisture transfer models to airflow network programs, which are either used as a stand-alone program or coupled with building energy simulation programs. (Clarke J.A. 2001; Feustel H.E. 1990; Dols W.S., George N.W. 2002; EnergyPlus. 2009.)

Flow patterns and turbulence will influence the pressure on the façades of buildings. With the same wind pattern and turbulence, the wind pressure on the façades ( $p_s$ ) are proportional to local outdoor atmospheric pressure at the same level as in an undisturbed wind approaching the building ( $p_v$ ). Wind pressure on the building envelope is usually expressed by pressure coefficients ( $C_p$ ), defining the proportional relationship of  $p_s$  and  $p_v$  as follows:

$$C_p = \frac{p_s}{p_v}, \quad p_v = \frac{\rho_a U_H^2}{2} \quad (1)$$

Where

$U_H$  is approaching wind speed at wall height  $H$ , m/s

$\rho_a$  is ambient (outdoor) air density,  $\text{kg/m}^3$

The influence of  $C_p$  on building performance is huge because several performance indicators for both energy consumption and thermal comfort, are often very sensitive to the air change rate, which depends on  $C_p$ . The high uncertainty associated with  $C_p$  values is caused by the wide range of influencing parameters, including building geometry, detail of façade (e.g. external shading devices, balconies), shelter elements (e.g. buildings, trees), windows position on the façade, variable wind speed and wind direction, and turbulence intensity.

### $C_p$ Data Sources

In practice, there are two different kinds of source of  $C_p$  data; primary sources, which include full-scale measurements, reduced-scale wind tunnel tests, and secondary sources like modeled data through CFD simulations (Costola D., Blocken B., Hensen J.L.M. 2009.) Primary sources are generally considered to be more reliable  $C_p$  but are difficult to obtain. A brief review of the advantages and disadvantages of different sources is presented in the following text.

On-site, full-scale measurements on real building façades are generally considered to be the most accurate description of  $C_p$  but they are expensive to obtain and – obviously – not available during the design of a new building. Measurement based  $C_p$ s are hence, mainly used for validation purposes. A large amount of high-quality experimental data has recently become available through ultrasonic anemometers and pressure transducer measurements. (Richards P.J., Hoxey R.P., Short L.J., 2001; Richardson G.M., Robertson A.P., Hoxey R.P., Surry D. 1990; Levitan M.L., Holmes J.D., Mehta K.C., Vann W.P. 1991.) Despite of these advances, measure  $C_p$  data for various urban settings largely remains elusive (Reinhold T.A. 1982.) Wind tunnel experiments are generally considered as the most reliable source of pressure data for buildings during the design phase. While more affordable than on-site full-scale measurements, wind tunnel experiments are still unavailable for most projects. There also remains a scale issue related to wind tunnel experiments, even though recent studies that did pay attention to turbulence domains, demonstrated significant agreement between wind tunnel and on-site full-scale measurements (Richards P.J., Hoxey R.P., Connell B.D., Lander D.P. 2007).

Secondary sources are used to provide  $C_p$  data for infiltration and ventilation studies with relatively low cost, when primary data sources are not available. Secondary sources of  $C_p$  data include  $C_p$  databases and analytical models. The previous ones are compilations of  $C_p$  data from one or more primary sources, classifying the data according to some parameters, such as building shape and orientation to the incident wind. Analytical models are equations to calculate  $C_p$  for specific building configurations, which is a more user-friendly way to access the large amount of empirical data used in the model formulation. Two main  $C_p$  databases and three most widely used analytical models are introduced here.

Two  $C_p$  databases are currently widely available and established; the AIVA database and the ASHRAE Handbook. These two data sets are also the ones used in popular building energy simulation programs. In

1986, a compilation of  $C_p$  data was published by the Air Infiltration and Ventilation Centre (AIVC), which rapidly became an important international reference. (AIVC. 1984.) AIVC database provides  $C_p$  data for low-rise buildings with tables of averaged data, which reference buildings with up to 3 storey, as well as  $C_p$  figures with vertical profiles for high-rise buildings. In the ASHRAE Handbook of Fundamentals, a chapter dedicated to airflow around buildings has information about  $C_p$  which is provided by reproducing data from several primary sources. (ASHRAE. 2009.) Surface-averaged  $C_p$  data as well as examples of the distribution over the surface for low-rise and high-rise buildings are included. However, a main difference of ARSHRAE database is that instead of data for sheltered buildings, it provides correction factors for the reference wind speed based on sheltering factors.

Analytical models are based on wind-tunnel and/or full-scale experiments. They estimate  $C_p$  data for a broader range of building configurations including obstructions and the effect of different wind directions and speeds across a façade. Generally, analytical models use regression techniques to analyze large amount of  $C_p$  data from primary data. The reliability of these equations therefore depends on the quality of  $C_p$  experimental data used in the regression, and on the parameters considered in the analysis. The three main analytical models are the models by Swami and Chandra (Swami M.V., Chandra S. 1988.), CpCalc+ (Grosso M. 1992.) and Cp Generator (Knoll B., Phaff J.C., de GidsW.F.. 1995.). Swami and Chandra's model provides one simple equation for low-rise buildings and another for high-rise buildings, indicating a normalized pressure coefficient compared to that with orthogonal wind. CpCalc+ was developed by COMIS to provide  $C_p$  data incorporating the effect of sheltered buildings, building shape and distribution of  $C_p$  on façade, by adding new parameters including the power-law exponent of the mean wind-speed profile, the plot density, the relative building height to the surrounding buildings, the width to height aspect ratio and the position at the façade. Developed by Dutch institution TNO, the Cp Generator is a web-based application. It is similar to CpCalc+, except improvement on sheltering effect by considering discrete block-shaped obstructions instead of the neighborhood plan area density, which is used in CpCalc+.

In conclusion, except CFD simulation, all  $C_p$  data source are based on measurements, regardless of on-site full-scale measurements or wind tunnel re-scale experiments. Several limitations exist because of the cost of measurements, among which there are two main aspects that cannot be fully covered by measurements and regression formulas from measurements; that is

complex building geometry and sheltering effect by buildings and terrain. Some authors also mentioned the lack of data for complex building shapes such as L-shape or U-shape. (Swami M.V., Chandra S. 1988; Grosso M. 1992.) However, the building geometry is one of the most critical factors in designing a ventilation strategy and the sheltering effect of urban context cannot be ignored in the real world. As one of the primary sources of  $C_p$  data, CFD simulation is the only way to solve these problems now, despite it being an time consuming and error-prone process for beginners. The advantage of CFD simulation and a methodology to build on-site  $C_p$  database for building simulation through CFD will be discussed next.

### CFD Simulation

Due to the rapid development of software and hardware power, CFD simulations are becoming increasingly accessible to smaller design firms. As mentioned earlier, the main advantage of CFD simulation is that they are not limited by any particular urban scene geometry or complexity and can yield wind pressure values on each window in a scene. Analytical pressures only provide  $C_p$  values for rectangular building variances. Instead programs use one  $C_p$  value for every window in a façade and thus estimate an average, constant air exchange rate for a whole building. For a multistory building that borders a narrow urban canyon, this approximation is evidently wrong.

Several studies about reliability of CFD simulation results were conducted. AIJ (Architectural Institute of Japan) carried a series of study on CFD simulation of outdoor environment. Akashi Mochida and other researchers compared wind velocity of CFD simulation result with wind tunnel experiment data, as well as turbulence strength, indicated by turbulent kinetic energy and turbulent energy dissipation rate, for a single high-rise building, with various  $k-\epsilon$  models. (Mochida A., Tominaga Y., Murakami S. and etc. 2002.) The results showed that there were high agreements on CFD simulation results and wind tunnel experiment data, with appropriate settings. After that, cross comparisons of CFD simulation results with wind tunnel experiments data around a high-rise building and within a building complex, were conducted by Yoshihide Tominaga. (Tominaga Y., Mochida A., Shirasawa T. and etc. 2004.) Though the cases in this study were quiet complicated with a building complex in an actual urban area, relatively high agreements were achieved, with special attention to CFD simulation settings.

Also some studies on comparison of wind pressure were conducted. Morimasa Watakabe and his

colleagues compared wind pressure measurements on tower-like structure obtained from full-scale observation, wind tunnel test, and the CFD simulation results. (Morimasa W., Masamiki O., Hisashi O. and etc. 2002.) Both mean wind pressure coefficients and wind pressure coefficients are compared in the study. The results are shown as Figure 1. For descriptions of model and measurement methods, please refer to the original paper. Lou compared wind pressure on a plate cantilevered roof of an exhibition building, with wind tunnel data and CFD simulation result. (Lou W.J., Sun B., Lu D. and etc. 2007.) The study showed that the wind pressure can be predicted by CFD simulation even in complicated models. In previous research, the author simulated the wind pressure for a tall building, to compare simulation results of different turbulence model with wind tunnel data cited in ASHRAE. The comparison of wind pressure coefficients data for front wall is shown in Figure 1. (Wang B., Lin B.R. 2011.)

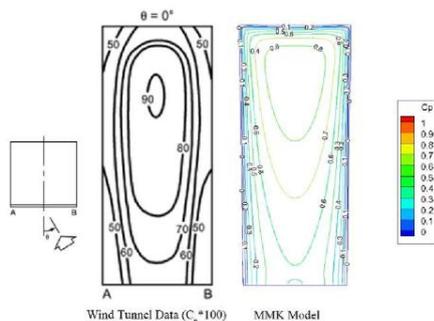


Figure 1 Comparison of  $C_p$  Data

All these studies showed that CFD simulation can predicted the wind environment well, including wind flow velocity and turbulence strength, as well as wind pressure, compared to wind tunnel experiment data and full-scale measurement data.

### Methodology

Building performance programs could simulate hourly natural ventilation for the whole year and then evaluate the potential of natural ventilation, by airflow network model, which calculate air flow rate based on pressure network. (EnergyPlus. 2009.) Normally a  $C_p$  database or an analytical model is used to provide the pressure of each window for the calculation of airflow network. Though the limitations of these methods and advantage of CFD simulation were clear to everyone, however, it is not common to use CFD simulation as source of custom  $C_p$  data for building performance simulation. The main reasons are the required level of expertise and the high cost of these simulations, both in terms of computational resources and user time. IES <VE> is the only tool that at present includes a prototype of this

integration, but at present it is less useful due to the limited number of options for the CFD simulation, limited grid options and lack of integration in the post-processing stage. (Integrated Environmental Solutions Limited. 2009). However, it does indicate a possible direction to improve the use of CFD as source of  $C_p$  data and the limitation can be overcome by the integration of pre-processing and post-processing between building energy simulation and CFD.

To overcome the limitation for using CFD simulation as a  $C_p$  data source in building simulation tool, A workflow of simulating detailed natural ventilation in building was developed in this paper. Phoenics was used as the CFD simulation tool, which is one of the most widely used commercial CFD software in simulating outdoor environment. (Ludwig J.C., Mortimer S. 2010.) EnergyPlus was used for building energy calculations by the simulation tool. Rhinoceros, the widely used NURBS modelling tool, was employed for data transfer between the two models, by self-developed tools. This workflow presented here could be used as a guideline for simulation of natural ventilated building, and also for future software development.

The first step of the workflow was to build a model in Rhinoceros that was exported as a source for Phoenics. After that, 8 CFD simulation cases were run to generate a complete wind environment. Then the pressure information of all cases was imported into Rhino. This step could be conducted automatically by using a Rhino plug-in. The next step was to locate each window in Rhino model and obtain pressure data to calculate  $C_p$  values of each window from the 8 cases. Then the  $C_p$  database were added into EnergyPlus case file and assigned to each window, replacing the original  $C_p$  values, by a text editor software, manually or automatically by scripts. The steps were clear but difficult to carry out manually and complex for beginners. Hence, several Rhino plug-ins and script were developed to conduct all the processes.

### Simulation-based $C_p$ database

As established, hourly  $C_p$  database or formula to calculate  $C_p$  is required in building simulation programs. The simulation-based  $C_p$  database will provide a set of  $C_p$  value for each window in 8 wind directions, based on CFD results. As indicated in the definition,  $C_p$  values are independent from approaching wind speed. However, this conclusion is made based on an assumption that the air flow pattern will stay almost the same with the same wind direction and the wind velocity is linear to the approaching wind speed. In this

paper several CFD simulation cases were designed to verify this assumption.

Phenics 2009 is used in this study as the CFD simulation tool. Detailed settings followed the CFD simulation guideline provided by AIJ (Architectural Institute of Japan), which is concluded by several studies on cross comparison of CFD simulation results and wind tunnel experiment data. (Tominaga Y., Mochidab A., Yoshie R. 2008.) The grid number is around 2,000,000 in an adequately sized domain. The inflow boundary condition has a vertical velocity profile, as well as the vertical distribution of turbulent energy, both of which are usually given by power laws. For lateral and upper surfaces of computational domain and downstream boundary, the normal velocity component and normal gradients of tangential velocity components were set to zero. A logarithmic law with roughness parameters was used in setting solid surface boundary conditions. Steady simulation with Simple solution algorithm and Quick differencing scheme were used in this study. The turbulence model is MMK model (a revised standard ke model). Also, grid independent study was conducted for all the cases.

Sheltering effect is the most critical factors influencing the wind environment around buildings. As explained above, there are two main considerations for surrounding buildings. First is the density and second is the relative height to surround buildings. Hussain M. studied the influence of density on flow pattern around buildings, indicating that three different flow patterns could be presented by three models with different distances between buildings: half of building height, same building height and two times of building height. (Hussain M. 1988.) Similar categories were used for relative building height. Therefore, five different urban models were chosen in this study. Shown as Figure 2 and Table 1.

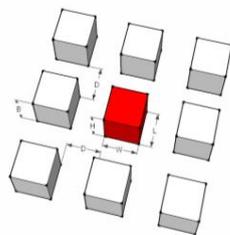


Figure 2 Sketch and Labels of an Urban Model

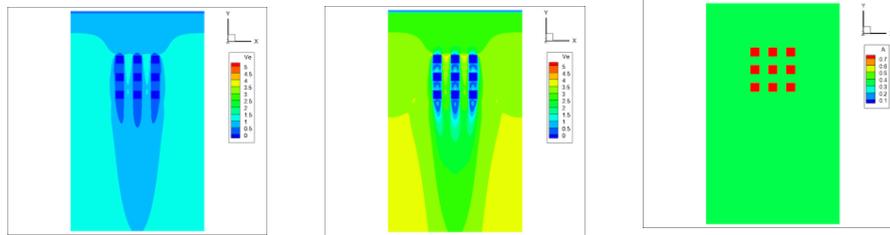
Table 1 Different Urban Models

NO.	L×W×H(M)	R(B/H)	D (M)	SKETCH
1	20×20×20	1	1	

2	20×20×10	2	1	
3	20×20×40	0.5	1	
4	20×20×20	1	2	
5	20×20×20	1	0.5	

Because of the symmetry of the model, only two approaching wind directions, 0 degree and 45 degree, were tested instead of all 8 wind directions. For the first model, three wind speeds were simulated (1m/s, 3m/s and 5m/s at 10m height) while two were used in all the other cases (1m/s and 3m/s at 10m height). Comparison of air flow at 10m height was conducted in the entire domain. Relative speed was defined as ratio of speed at a certain point in different cases with different approaching wind speed. For example, contour of wind speed and relative speed for comparison of 1m/s case and 3m/s case in model 1 is shown in Figure 3. As we can see, the flow pattern stays almost the same and the ratio of wind speed at a certain point in two cases is constant.

Similar comparisons of different wind speed were conducted for other wind directions and urban models. As example, relative speed of three different approaching wind speed for model No.1 with two wind directions are shown as Table 2. The results show that the relative speed of different cases are stable in the whole domain. The main differences exist in the turbulence area behind buildings. Turbulence is also causing more difference for 45 degree wind direction than that of 0 degree. The comparison of other urban models have a similar principle that in most areas of the domain the wind speed has a roughly linear correlation with the approaching wind speed, with a 10% maximum relative error, which is acceptable in application. This verifies the assumption that the air flow pattern will stay almost the same with the same wind direction and the wind velocity is linear to the approaching wind speed. Therefore, we can run 8 CFD simulation cases and calculate a set of  $C_p$  for each window, for any shape of building in any urban context. After that we input this  $C_p$  database into building simulation tools and link it with weather data to get accurate hourly wind pressure on each window, which leads to a detailed natural ventilation simulation.



(a) wind speed in 1m/s case (b) wind speed in 3m/s case (c) relative speed  
Figure 3 Wind Speed and Relative Speed for Two Cases with Different Approaching Wind Speed

Table 2 Relative Speed of Three Different Approaching Wind Speed

URBAN MODEL NO.	WIND DIRECTION	RELATIVE SPEED (1M/S AND 3M/S)	RELATIVE SPEED (3M/S AND 5M/S)	RELATIVE SPEED (1M/S AND 5M/S)
1	0			
1	45			

### CASE STUDY

Several typical cases were tested for the workflow, to compare the influence of natural ventilation and indoor thermal condition by changing the wind pressure coefficients. The building model was a 20m×20m×20m cubic box model. It was modeled as a 6 floors office building in DesignBuilder (a graphical user interface of EnergyPlus), with a simplified floor plan shown as Figure 4. The building was heated but not cooled. Natural ventilation were set available all the time, to evaluate the highest potential of natural ventilation. The weather condition was set as Boston, MA, U.S.A. Annual average indoor air temperature and air change rate (ACH) of each room were used in the comparisons.

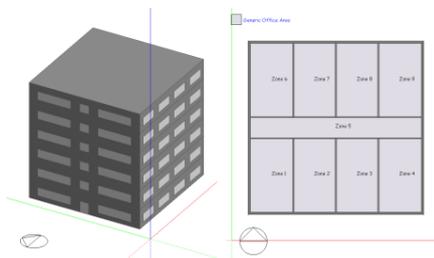
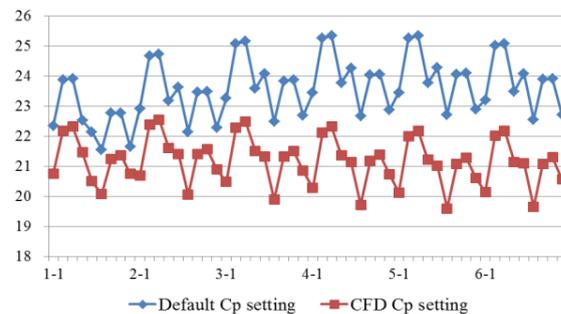


Figure 4 3D Model and Floor Plan of Basic Model

### Comparison of Air Temperature



### Comparison of ACH

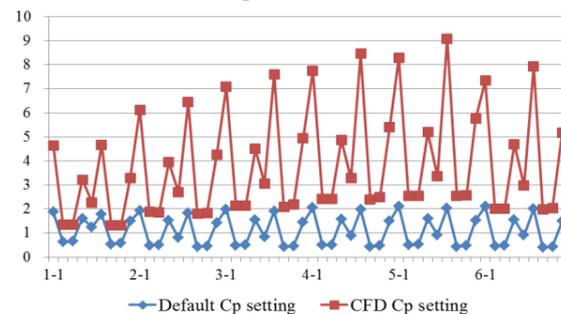


Figure 5 Comparisons of Different Cp Settings

First comparison was conducted between default EnergyPlus  $C_p$  setting and proposed CFD based  $C_p$  setting. shown as Figure 5. As introduced above, the default  $C_p$  setting in EnergyPlus is call Average-surface Calculation, which is calculated based on analytical model developed by Swami and Chandra. Instead of giving average  $C_p$  values, the proposed CFD based  $C_p$  model assigned different  $C_p$  values for each window.

As we can see from the results of comparison, the default  $C_p$  setting in EnergyPlus underestimate the potential of natural ventilation by ignoring the wind pressure difference of windows on the same façade. Also, the simulation result with default  $C_p$  setting showed that the annual average air temperature in rooms on top floor will be higher, however, it was not true because the top floor rooms had better natural ventilation with higher approaching wind speed. According to this comparison, with CFD based  $C_p$  model we can achieve more accurate and detail information of natural ventilation.

Another comparison was conducted between different urban models. Simulation results of building with no urban context and with urban model 1, 4 and 5 (see Table 2) were shown as Figure 6.

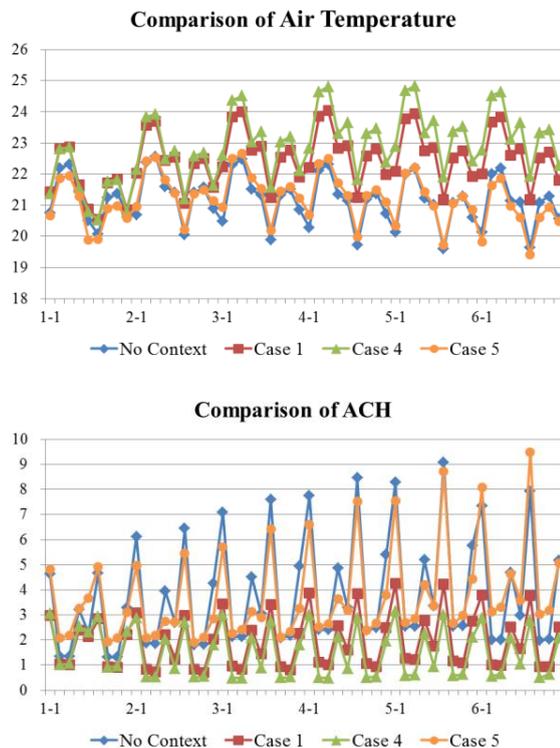


Figure 6 Comparisons of Different Urban Models

From this comparison we can easily see the influence of urban context on natural ventilation. Obviously the

denser the urban context, the worse potential of natural ventilation. In the urban model 5, when the distance of buildings is twice as the building height, the urban context had almost no influence on natural ventilation, which agreed with the study of urban of density on flow pattern around buildings conducted by Hussain M. (Hussain M. 1988.).

As we can read form the case study results, the proposed CFD based  $C_p$  model could provide detail performance simulation of naturally ventilated buildings, with any building shapes and any urban context condition. What's more, this workflow has great improvement in informing the architect with detail simulation data, for example different natural ventilation performance of each room. But the most critical improvement of this workflow is the capacity of simulating buildings with particular shape and urban context.

## CONCLUSION

Natural ventilation is one of the most widely used methods to reduce energy consumption as well as improving thermal comfort. However, the  $C_p$  values used now in most of the energy simulation software are based on  $C_p$  database or analytical models, which ignores the effect of urban context and form of building itself. A methodology to analyze natural ventilation in detail with CFD simulated  $C_p$  database was developed in this paper. Also several case studoes were introduced, indicating the great improvement in informing the architect with detail data. Studies show that the proposed CFD based workflow has great advantage in detail simulation, especially the capacity of simulating buildings with particular shape and urban context.

However, more research remains in accurate simulation of natural ventilation. For example, in DesignBuilder the control strategy of natural ventilation opens the windows when outdoor temperature is lower than set point of indoor temperature, which results in a waste of energy by having cooling system at the same time with opened window. This is one of the directions of a future study.

## REFERENCES

- AIVC. 1984. In: Wind pressure workshop proceedings AIC-TN-13.1-84. Brussels: AIVC.
- ASHRAE. 2009. ASHRAE Handbook - 2009 Fundamentals Chapter 26., Atlanta GA.
- Busch J.F. 1992. Tale of two populations: thermal comfort in air-conditioned and naturally ventilated



- offices in Thailand. *Energy and Building*. 18(3-4): 235~249.
- Clarke J.A. 2001. *Energy simulation in building design*. Oxford: Butterworth-Heinemann.
- Costola D., Blocken B., Hensen J.L.M. 2009. Overview of pressure coefficient data in building energy simulation and airflow network programs. *Building and Environment*. 44: 2027-2036.
- Dols W.S., George N.W. 2002. *CONTAMW 2.0 User Manual*. NISTIR 6921.
- EnergyPlus. 2009. *EnergyPlus Engineering Reference*. US Department of Energy.
- Feustel H.E. 1990. COMIS - an international multizone air-flow and contaminant transport model. *Energy and Buildings*. 30:3-18.
- Grosso M. 1992. Wind pressure distribution around buildings: a parametrical mode. *Energy and Buildings*. 18:101-31.
- Hagentoft C. 1996. Heat, air and moisture transfer in insulated envelope parts: performance and practice, International Energy Agency, Annex 24. Final Report, vol. 1. Leuven: Acco.
- Hussain M.. 1988. A wind tunnel study of mean pressure forces acting on large group of low-rise buildings. *Journal of Wind Engineering and Industrial Aerodynamics*. 31, 41-66.
- Integrated Environmental Solutions Limited. 2009. *Integrated Environmental Solutions, MacroFlo Calculation Methods - <Virtual Environment> 5.6*, n.d.
- James W.A. Residential passive ventilation systems: Evaluation and Design, AIVC Technical Note 54.
- Knoll B., Phaff J.C., de Gids W.F.. 1995. Pressure simulation program. In: *Proceedings of the conference on implementing the results of ventilation research*. AIVC.
- Levitan M.L., Holmes J.D., Mehta K.C., Vann W.P.. 1991. Field measurements of pressures on the Texas Tech building. *Journal of Wind Engineering & Industrial Aerodynamics*. 38:227-34.
- Liddament M.W. 1986. *Air infiltration calculation techniques - an applications guide*. Bracknell: AIVC.
- Lou W.J., Sun B., Lu D. and etc. 2007. Wind tunnel test and numerical simulation of wind loads on complex cantilevered roofs. *Journal of Building Structures*. 28(1), 107-118.
- Ludwig J.C., Mortimor S.. 2010. *PHOENICS-VR Reference Guide*. London: CHAM.
- Mochida A., Tominaga Y., Murakami S. and etc. 2002. Comparison of various k-ε models and DSM applied to flow around a high-rise building - report on AIJ cooperative project for CFD prediction of wind environment. *Wind and Structures*. 5 (2~4), 227-244.
- Morimasa W., Masamiki O., Hisashi O. and etc. 2002. Comparison of wind pressure measurements on tower-like structure obtained from full-scale observation, wind tunnel test, and the CFD technology. *Journal of Wind Engineering and Industrial Aerodynamics*. 90, 1817-1829.
- Reinhold T.A. 1982. *Wind tunnel modeling for civil engineering applications*. Cambridge: Cambridge University Press.
- Richards P.J., Hoxey R.P., Connell B.D., Lander D.P. 2007. Wind-tunnel modelling of the Silsoe Cube. *Journal of Wind Engineering & Industrial Aerodynamics*. 95:1384-99.
- Richards P.J., Hoxey R.P., Short L.J., 2001. Wind pressures on a 6 m cube. *Journal of Wind Engineering & Industrial Aerodynamics*. 89:1553-64.
- Richardson G.M., Robertson A.P., Hoxey R.P., Surry D. 1990. Full-scale and model investigations of pressures on an industrial/agricultural building. *Journal of Wind Engineering & Industrial Aerodynamics*. 36:1053-62.
- Swami M.V., Chandra S. 1988. Correlations for pressure distribution on buildings and calculation of natural-ventilation airflow. *ASHRAE Transactions*. 94:243-66.
- Tominaga Y., Mochida A., Shirasawa T. and etc. 2004. 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*. 3 (1), 63-70.
- Tominaga Y., Mochida A., Yoshie R. 2008. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics*. 96, 1749-1761.
- Wang B., Lin B.R. 2011. Comparison of turbulence models in simulating key elements of outdoor wind environment around building complexes. *Proceedings of Building Simulation 2011, Sydney*.