ELSEVIER

Desalination xxx (2006) 256–265

DESALINATION

www.elsevier.com/locate/desal

Investigation of effects of window combinations on ventilation characteristics for thermal comfort in buildings

M.A. Hassan[a]*, N.M. Guirguis[a], M.R. Shaalan[b], K.M. El-Shazly[c]

[a]*Housing & Building Research Center, P.O. Box 1770, Cairo, Egypt*
email: waghassan@yahoo.com
[b]*Faculty of Engineering, Zagazig University, Zagazig, Egypt*
[c]*Shoubra Faculty of Engineering, Zagazig University, Egypt*

Recieved XXX; accepted XXX

Abstract

An energy-efficient building is a major target of building researchers and designers worldwide. Obviously, any portion of energy that can be saved in this respect can be directed to industrial processes, if any. Building energy consumption can be reduced through various systems such as air conditioning (a major building energy consumer), lighting, equipment, etc. In regions where energy is limited or scarce, air conditioning would have to be replaced by natural ventilation for the removal the building heat load for thermal comfort. In this article, results of an investigation of natural ventilation criteria as affected by various factors are presented. In particular, the effects of opening location and size (window-to-wall ratio) and building orientation relative to prevailing wind direction are considered. Flow visualization and finite element techniques were adapted to gain better understanding of these effects. For flow visualization, a smoke tunnel test facility was constructed at the Housing & Building Research Center and utilized to test several idealized building models. On the other hand, a Computational fluid design software (ANSYS FLOTRAN) was applied to the present problem. Both techniques (visualization and computation) gave clear pictures of what happen inside the ventilated space under different opening locations and combinations for various wind directions. Knowing the flow structure inside the ventilated space, it would be a simple task to evaluate the thermal comfort index ($\overline{PMV}$) for a given thermal load (internal heating, solar radiation, etc.). Also, such pictures should help architects to propose good designs of natural ventilation systems in their buildings.

Keywords:

1. Introduction

Since the mid 1970s there has been an urgent need to reduce the use of energy in the heating and cooling of buildings worldwide. In developing countries, the energy consumption for maintaining an acceptable environment in buildings constitutes the largest portion of the total energy demand. In advanced countries, the proportion of energy usage for buildings is larger than that

*Corresponding author.

Presented at the Conference on Solar Energy, Bahrain 2004

0011-9164/06/$– See front matter © 2006 Elsevier B.V. All rights reserved.
doi:10.1016/j.desal.0000.00.000

DESL0038.fm Page 257 Monday, April 17, 2006 6:39 PM

used in other purposes (industrial processes, transport, etc). Ventilation engineering is required to assess the air leakage characteristics of a given building and to design the ventilation openings required. Recently, ventilation has assumed a new dimension in the building design process as a result of the much publicized sick building syndrome.

Ohba [3] described the effect of wind on the ventilation system of such a half-enclosed building with a large opening to the outside. Video image experiments were conducted of the court space within a high-rise residential building and of the space within a domed building with a removable roof. Three-dimensional image pictures of the court space in the high-rise residential building were created using video imaging techniques and the graphics software library. This enabled better understanding of the whole ventilation system. Also, the results indicated that in the domed building with removable roof, one could obtain 7.0 air changes per hour with an opening ratio of 25% and the reference velocity of 3 m/s.

Tsutsumi et al. [6] presented full-scale measurements of indoor thermal comfort factors and numerical simulation of indoor airflow. The measurements were carried out in summer in two dwelling units of apartment buildings, one ventilated naturally through opening windows using *Tsuufuu* (Japanese word like the cross ventilation) and the other was unventilated. Full-scale measurements of the wind and thermal environmental factors in and around an apartment building were carried out to examine the effect of indoor airflow on thermal comfort. The results were compared to estimate the change of thermal comfort sensation with indoor air flow speed, showing that the value thermal comfort index *PMV* in the case of *Tsuufuu* is lower than that in case without *Tsuufuu*. Computational fluid dynamics (CFD) was applied to estimate the indoor airflow caused by *Tsuufuu*. The numerical simulation results were generally in good agreement with those of model experiments.

Riffat [8] developed a unit combining light-pipe technology and passive stack ventilation by utilizing the light pipe as an exhaust stack. The measurements of flow velocity air flow rates and natural stack ventilation through the test chambers were carried out using a constant injection tracer-gas method. CFD was used to investigate the simulated performance of the passive stack ventilation through the integrated system using software package FLUENT. The results showed that CFD could be used as a successful tool in such ventilation systems.

Gan [4] developed the CFD program ventilation of a room with turbulence and energy exchange “VORTEX” for predicting the indoor environment in mechanically and naturally ventilated room. The program produced direct simulation of the thermal comfort indices *PMV* and PPD and air quality of room. The results showed that the predicted velocity and temperature were in good agreement with measured values.

In the present work an investigation was carried out of effect of wind speed and direction on thermal comfort for different window-to-wall ratios (WWR) and the window location using flow visualization and computational fluid dynamic techniques.

2. Test facility

2.1. Wind tunnel facility

Taking into account the criteria suggested by Cermak [2], a wind-tunnel test facility was designed, built and tested at the Housing & Building Research Center (HBRC) [5]. A simplified building model was constructed for wind tunnel testing and was including heating elements (for thermal measurements). The tunnel floor was provided with a facility for setting the model at any angle (α) relative to wind direction. The wind tunnel has a working section 1.0 m wide, 1.0 m high and 2.0 m long serving the purpose of testing building models of size that permitted

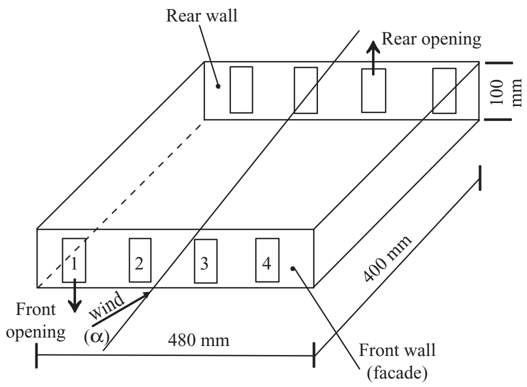


Fig. 1. Test Model.

the observation of phenomena with reasonable accuracy. The duct was provided with a standard-profile intake. Turbulence-generation grids were constructed and mounted some distance ahead of the test model to obtain turbulent shear-like (*ABL*) atmospheric boundary layer profile.

Test model: The model was constructed from Plexiglas sheet 5-mm thickness and has a dimension of 48 cm × 40cm × 10cm height as shown in Fig. 1. The model walls had double walls where thermocouples was fixed on the inner surface of the internal wall and the pressure tapping on the outer surface of the external wall thermocouples was fixed at levels corresponding to the standard levels (0.6, 1.1 and 1.8 m from the floor). The floor was provided with the heating elements. In order to complete the picture of thermal analysis, thermocouples were fixed to the upper surface of the Formica sheet (on the floor) at different points. A data logger OMEGA-32 was used to record the temperatures.

2.2. Smoke tunnel facility

Because wind is invisible, wind flow patterns around building cannot be easily conceived unless the flow is visualized. Also, the difficulty of carrying out field measurements around real buildings consolidates the importance of resorting to flow visualization which can be carried out indoor as a check of the validity of analytical and computational solutions. Such flow visualization makes it possible for architects to propose suitable solutions for wind effects in their building designs. Moreover, because of the presence of regions of circulatory flow in front of and downstream of buildings, the flow here being very complex, it would be very difficult to use hot wire, laser beam or Pitot probes to quantify the flow pattern for comparisons with analysis. It is therefore advised to visualize the flow as streaks, stagnation, and separated flow and reattachment zones around as well as inside building models. This helps architects to allow for natural ventilation systems in their designs.

Several practical methods of flow visualizing are available [1]. Such importance not only in giving a clear qualitative pictures of flow phenomena, but also, in some cases, in yielding quantitative information. The selection of a particular method depends on type of fluid, speed, temperature, turbulence, density variations, etc. in the present work, the “smoke (kerosene must) method” was adopted using a smoke tunnel facility that will be described hereafter. This technique was selected for reason of simplicity, tidiness, and suitability for detecting regions of separated flow and eddies. Fig. 2 shows the smoke tunnel test facility that was designed, built and tested at HBRC [5]. As shown in Fig. 2 the tunnel consists of a duct made up of five major sections: an inlet section, a working section, a contraction, an air source, and a smoke generator. The flow rate is controlled by means of a potentiometer from 1.5 to 3.5 m/s. The tunnel duct was mounted vertically to facilitate the flow of air-born smoke filaments in the working section.

Test model: The model is a parallelopiped-shaped, 100 × 83 × 26 mm had eight identical openings, four in front and four in rear walls. Various combinations of openings were tested (all closed but one open, some closed, all open) It was constructed from Plexiglas sheet 2-mm. All these flow visualization tests were carried

DESL0038.fm Page 259 Monday, April 17, 2006 6:39 PM

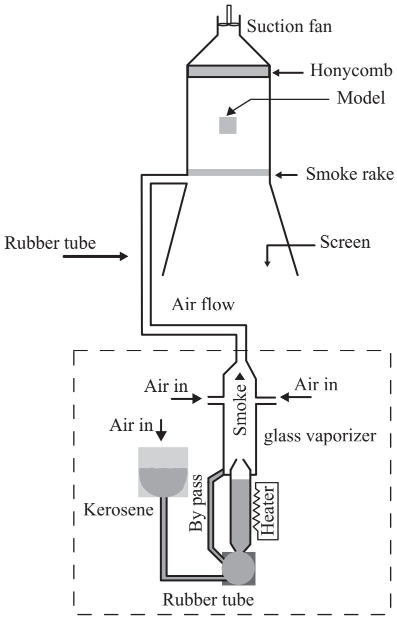


Fig. 2. Schematic for smoke tunnel.

out with same wind speed (2 m/s) and various wind directions ($a = 0^o$, 45º, 90º, and 135º). This set of tests was intended to investigate the effect of combinations of openings on flow configuration inside the model (building space), hence on thermal comfort therein.

3. CFD Technique ("ANSYS" software package)

During the last few years there has been great interest in developing CFD computer program for predicting the airflow in ventilated rooms. Numerical simulation has a lot of advantage, for example flexibility of the shape of houses and setting the exact parameters of airflow. The majority of these CFD program are based on the solution of Navier–Stokes equations, the energy equation, the momentum and concentration equation as well as the transport equations for the turbulent velocity and its scale. Because of both the natural and airflow are regard as turbulent flow, therefore, a mathematical model of turbulence is needed for the numerical simulation. The *k–e* turbulence model in which *k* is the turbulence velocity and *e* is its dissipation, is one of the most popular turbulence models. The *k–e* turbulence model is usually known as an implicit scale and currently it is most widely used because of its applicability to the wide-ranging flow problems and its lower computational domain than more complex models that are available.

In the present work an investigation was carried out using ANSYS to examine the effect of the combinations of two front windows for deferent wind speed and directions on thermal comfort in ventilated space in buildings. The ANSYS is based on the finite element method and solves both flow and thermal fields in any domain. A typical model of a building with openings in various combinations was considered for computation at different wind speed and directions under a given thermal load. The results of ANSYS consisted of (i) local velocity inside the ventilated space, which is averaged and used in the calculation of computational (PMV), (ii) average temperature inside the ventilated space which is averaged and used in the calculation of thermal comfort indices, (iii) local temperature on building walls which are averaged and used in the calculation of mean radiant temperature (T_{mrt}) which also used in the calculation of thermal comfort indices. The results will be analyzed and velocity-vectors and contours discussed in the comparison with visualizing findings (smoke-tunnel visualization). Fig. 3 shows sample of the velocity contours inside the ventilated space for two front wall windows open with no rear wall open at wind speed 3.8 m/s, in a horizontal plane corresponding to standard of 1.1 m from the floor.

4. Results and conclusions

Fig. 4 shows a comparisons between the CFD results and experimental results carried out in a smoke tunnel for flow pattern inside the test model for two different front wall openings, no

DESL0038.fm Page 260 Monday, April 17, 2006 6:39 PM

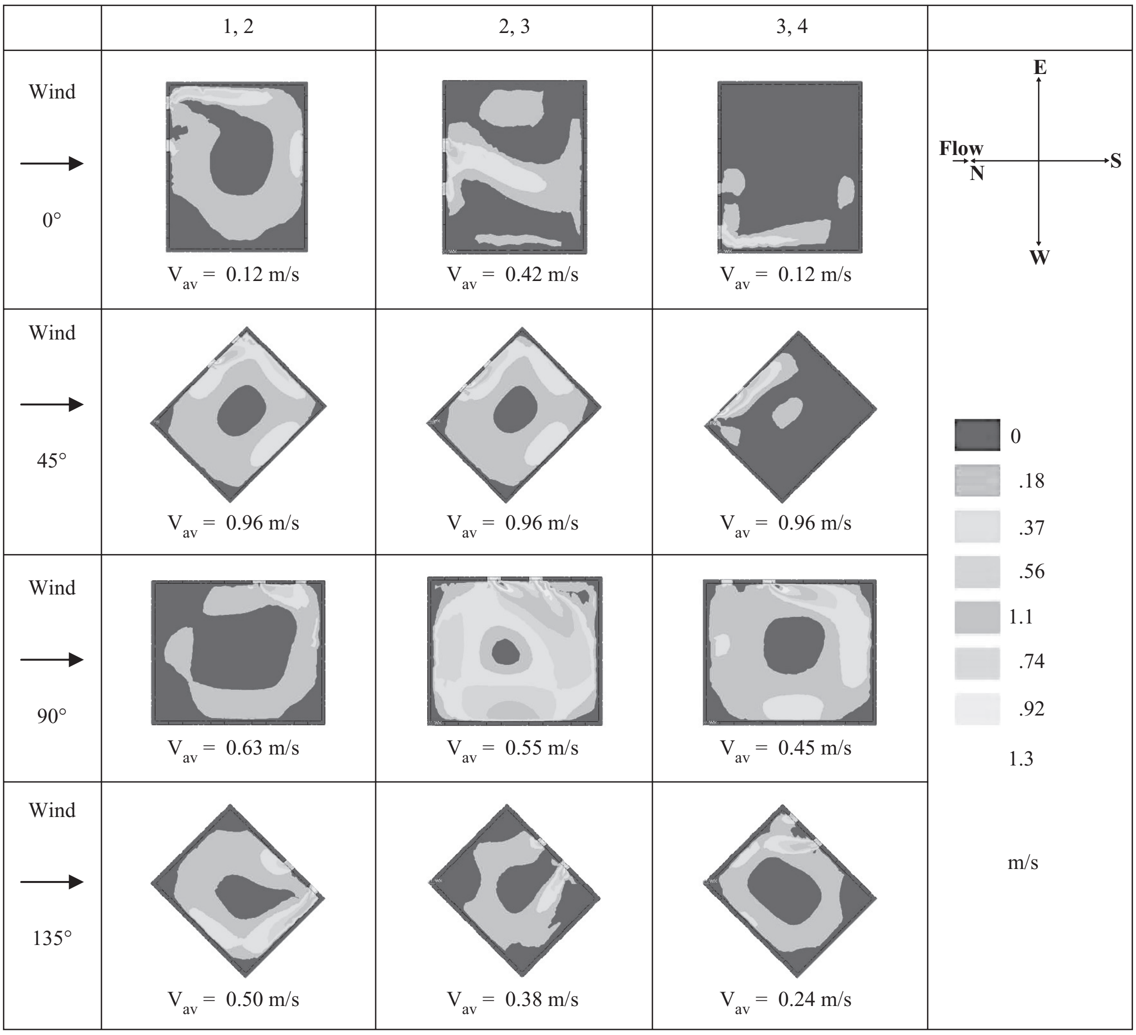


Fig. 3. Velocity contours inside ventilated space at wind speed = 3.8 m/s, in a horizontal plane corresponding to standard level of 1.1 m from the floor.

rear opening. Figs. 5–7 show comparisons between the CFD results and experimental results at different wind speed and directions for different combinations of two front windows open (while the rear windows are closed, single-sided ventilation). There is a good agreement between the CFD and the experimental results with accepted difference for all wind speed and directions. It may be concluded that:

1) The CFD techniques can be used successfully in the field of thermal comfort investigation with accepted results, which can be used by the architects and designer engineering in their design of buildings. A better flow pattern (hence better ventilation) is obtained in all cases with two openings compared with one opening.
2) In single-sided ventilation with two adjacent openings it does not matter whether they are right-, left-, or center-located.
3) In single-sided ventilation with two non-adjacent openings (corner located and center

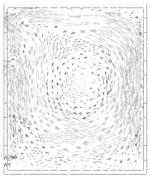
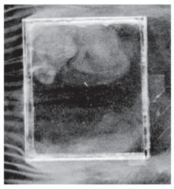
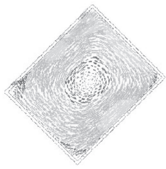
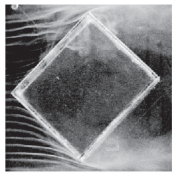
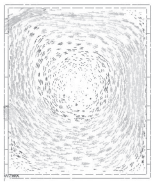
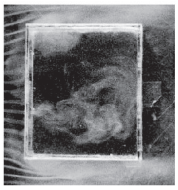
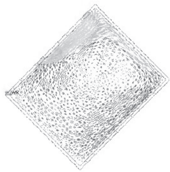
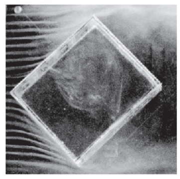
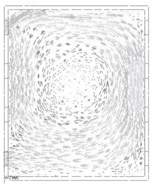
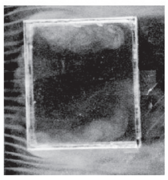
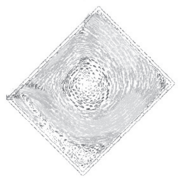
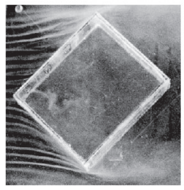
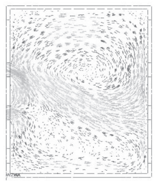
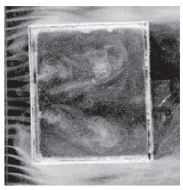
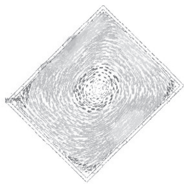
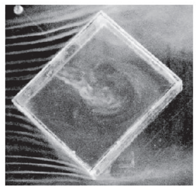
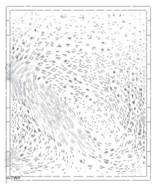
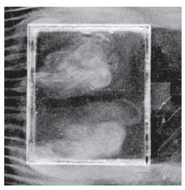
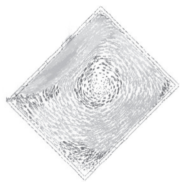
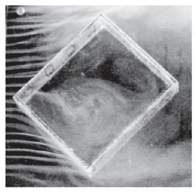
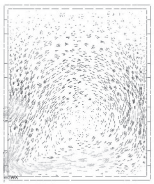
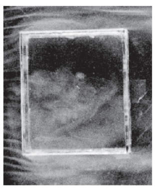
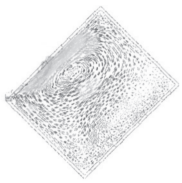
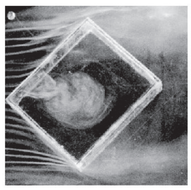
Front Wall Opening	0°		45°	
	Expected flow pattern inside model	Picture	Expected flow Pattern inside model	Picture
1, 2				
1, 3				
1, 4				
2,3				
2, 4				
3, 4				

Fig. 4.

DESL0038.fm Page 262 Monday, April 17, 2006 6:39 PM

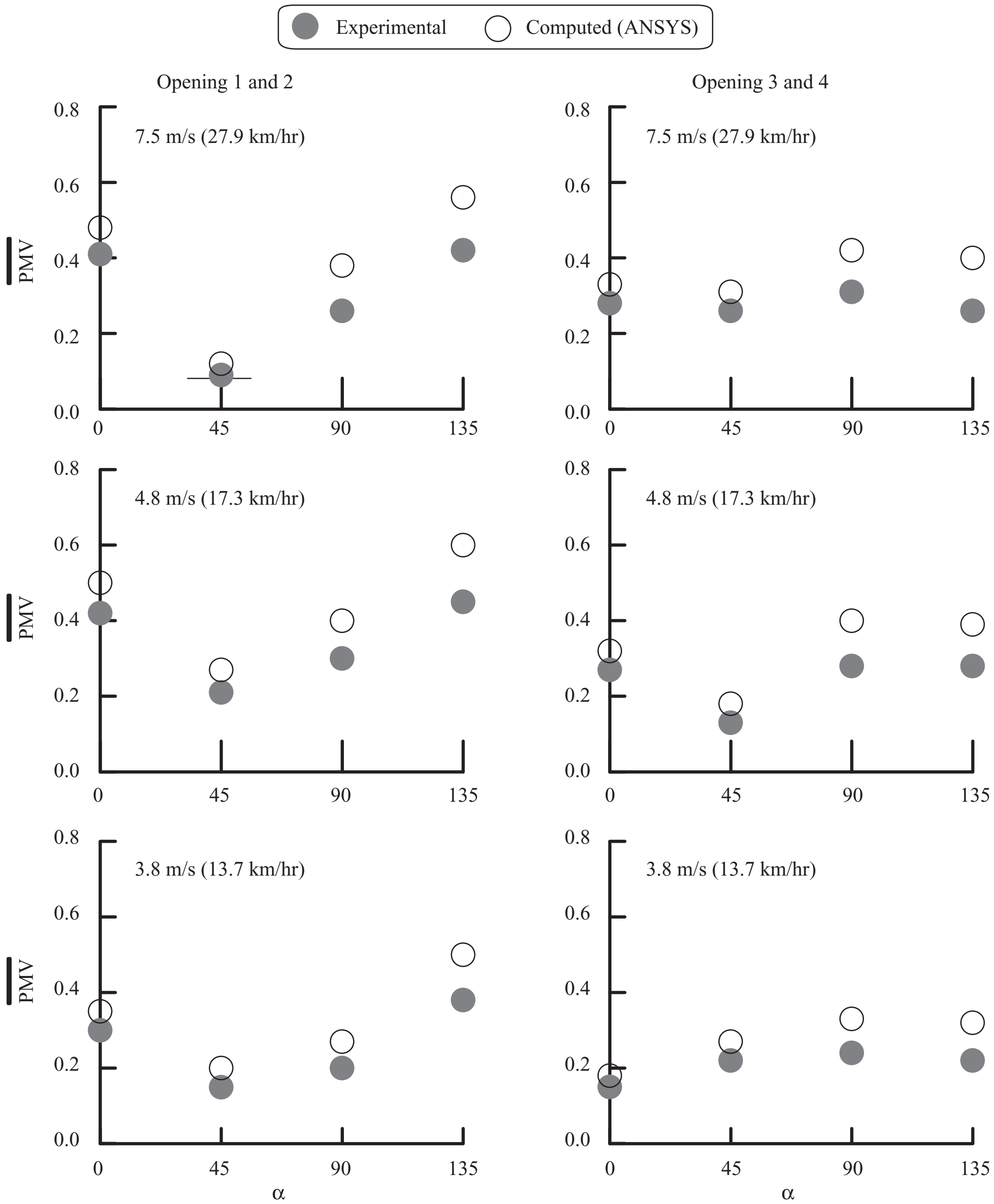


Fig. 5. Comparison between normalized experimental and computed $\overline{PMV}$ for different wind speeds and directions (single-sided ventilation with two adjacent openings).

located) there is some improvement over the case of two adjacent openings. In single-sided ventilation with two non-adjacent openings (one far left and one far right) give better ventilation than two adjacent openings (center-located).

DESL0038.fm Page 263 Monday, April 17, 2006 6:39 PM

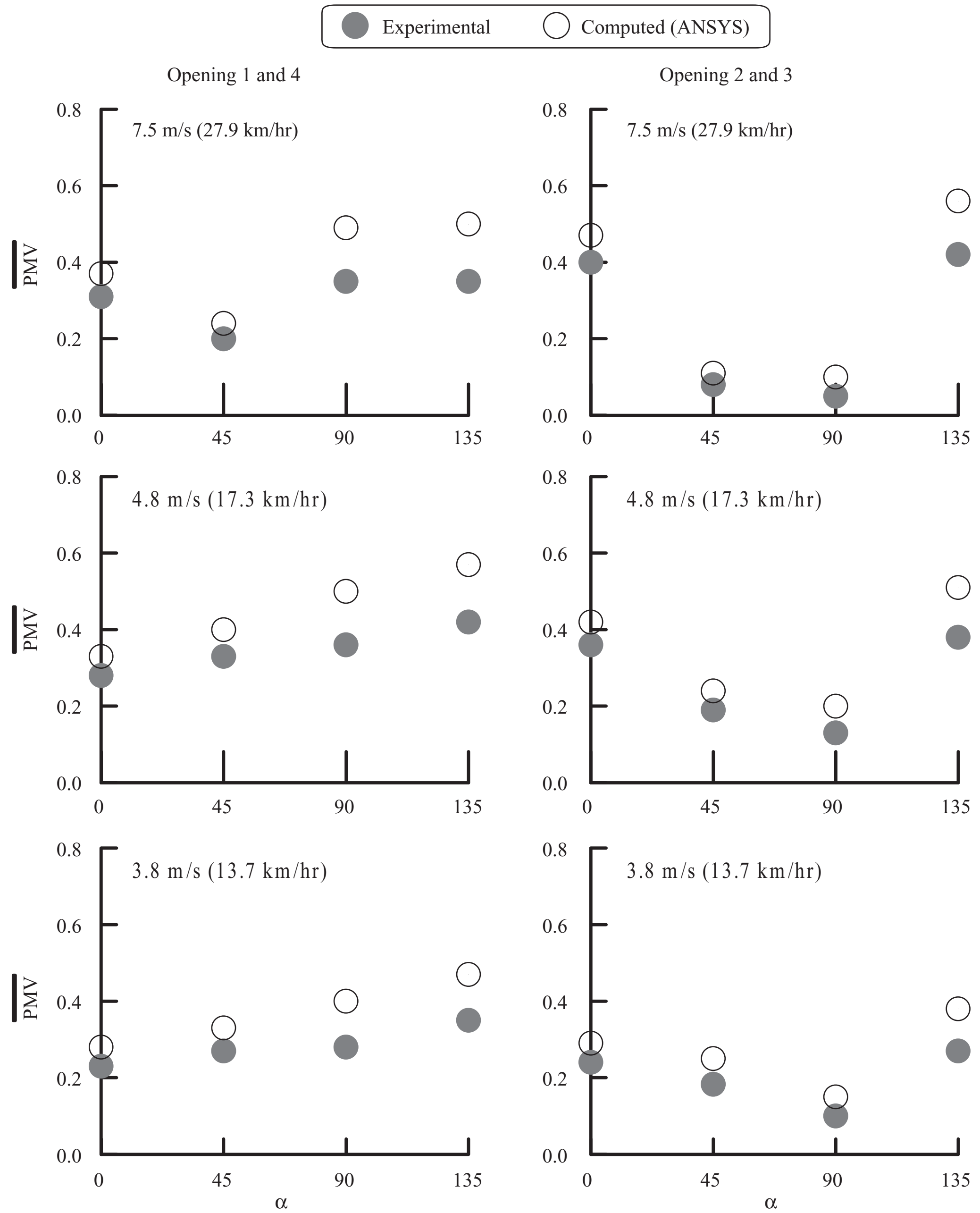


Fig. 6. Comparison between normalized experimental and computed $\overline{PMV}$ for different wind speeds and directions (single-sided ventilation with two openings).

DESL0038.fm Page 264 Monday, April 17, 2006 6:39 PM

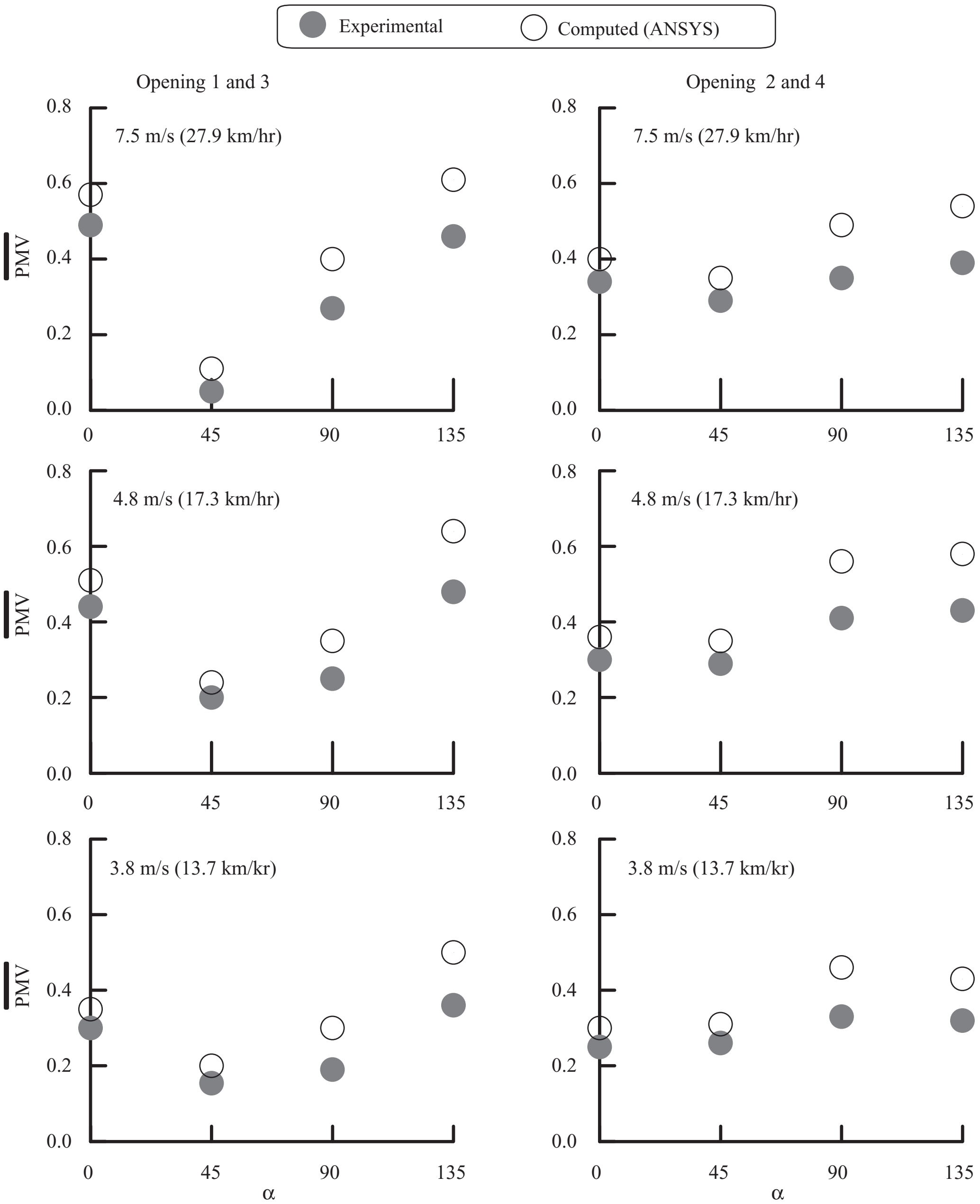


Fig. 7. Comparison between normalized experimental and computed $\overline{PMV}$ for different wind speeds and directions (single-sided ventilation with two non-adjacent openings).

DESL0038.fm Page 265 Monday, April 17, 2006 6:39 PM

References

[1] R.C. Pankhurst, Wind-Tunnel Technique, Book/ Isaac Pitman, London, 1968, 156–158.

[2] J.E. Cermak, Wind simulation criteria for wind-effect tests. J. Structural Eng., 110 (1984) 328–339.

[3] M. Ohba, Study on three-dimensional characteristics of natural ventilation in half enclosed buildings using video image technique, J. Wind Eng. Ind. Aerodyn., 46 &47 (1993) 831–836.

[4] G. Gan and H.b. Awbi, Numerical simulation of the Indoor Environment, Building and Environment, 29(4) (1994) 449–459.

[5] N.M. Guirguis, M.A. Hassan, M.R. Shaalan and G.B. Hanna, Construction and testing of a wind tunnel for In-door investigation of environmental effect of building, World Renewable Energy Congress, Denver, CO, USA, 1996.

[6] J. Tsutsumi, T. Katayama, A. Ishii, P. He and T. Hayashi, Investigation and numerical simulation of wind tunnel effects on thermal comfort in a house, J. Wind Eng. Ind. Aerodyn., 60 (1996) 267–280.

[7] M. Nabil, A.H. Mahmoud, R. Mohamed and B. George, Design and construction of a smoke tunnel as a visual assessment tool, Proceeding of Al-Azhar Engineering Sixth International Conference, September 1–4, 2000.

[8] S.B. Riffat, S.J. Smith and G. Oakely, Triplsave, The investigation and monitoring of a combined natural daylighting and stack ventilation system, Proceeding of World Renewable Energy Congress VII, Germany, 2002.

		Ver No.:100
Quality Management System	**Author Query Sheet**	**W.E.F 2nd Jan 2006**
		Page 1 of 2

Customer Name	Book / Journal Code	Name of the Book / Journal	Article ID
Balaban	DESL	Desalination	0038

Copy Editing:

1. Kindly provide a set of 3–5 keywords for your manuscripts.
2. Please check whether corresponding author is okay as identified.
3. Kindly note that the Reference [7] is uncited in the text.
4. Kindly provide caption for Figure 4.
5. Please check that figure captions for figures 5–7 seem to be same.

Graphics:

Nil

Pagination / QC:

Nil

Remarks:

Nil