Numerical studies on natural ventilation flow in an enclosure with both buoyancy and wind effects

Yingchun Ji, PhD student

Institute of Energy and Sustainable Development, De Montfort University, 1.05, Queens Building, the Gateway, Leicester, LE1 9BH, UK

Summary

Computational Fluid Dynamics (CFD) is used to predict natural ventilation flows in an enclosure with upper and lower openings connected to the environment. The natural driving forces are wind combined with buoyancy. CFD simulations were carried out by varying the strengths of buoyancy under a constant assisting wind effect. Simulation results of the volume flow rates through the enclosure, the stratification level and temperature distribution were compared with the corresponding experimental measurements and analytical studies by [1] and [2]. With the application of the RNG k-epsilon turbulence model and the proper considerations on the fluid boundaries, the overall computed airflow patterns and the airflow rates through the space agreed well with [1] and [2]. The close qualitative and quantitative agreement gave confidence in CFD technique for modelling natural ventilation flows in simple geometry buildings.

1. Introduction

Natural ventilation has proved to be an effective means to achieve good indoor air quality for occupants with relatively low energy usage in comparison with mechanical ventilation systems. The conventional techniques on modelling natural ventilation are simple mathematical models and small scale experiments, and these methods have been the subject of much research (e.g. [1, 3]). As an alternative in recent decades, CFD has been increasingly employed in the area of building environmental and natural ventilation strategies (e.g. [4, 5]). With the recent advances in computing power, the process of creating a CFD model and analysing the results is much less labour-intensive, reducing the time and therefore the cost. However, the stage that CFD can be used to model the real physical world routinely has not come yet due to the uncertainty of the numerical models in CFD. Validation work is still needed before presenting CFD results for engineering applications.

CFD techniques provide much more detailed information on the airflow of interest compared with the conventional methods of small scale experiments and mathematical analyses, e.g. air speeds, temperature profile, pressure distribution in the flow field, as well as the ventilation rates through the ventilation spaces. Previous studies on natural ventilation driven by combined forces of wind and buoyancy in a space with high and low level openings were conducted by [5], in which the main aims were to investigate numerical methods for modelling airflow patterns and temperature distributions within the fluid field of interest. This work is an extension of [5] and the objectives of this paper are to assess the predictions of ventilation flow rates through the space by varying the heat loads of the source. The flow patterns and the temperature profiles will be examined to test the consistency of the CFD application. The cases considered here show the effects on the airflow of a naturally ventilated space when its internal heat loads are altered.

This paper is set out as follows. Mathematical models and experiments are described in section 2. Section 3 discusses the CFD model. Results and discussions are given in section 4 followed by conclusions in section 5.

2. Mathematical models and experiments

The plume theory of [6] provides mathematical forms to calculate the volume flux Q and the reduced gravity G' in a plume generated by a localised point heat source with strength B along

vertical distance x. Assuming zero initial momentum and volume flux, Q and G' are defined by:

$$x = C^{-3/5} Q^{3/5} B^{-1/5} \tag{1}$$

$$G'(x, B) = (B^2 x^{-5})^{1/3} / C \tag{2}$$

where C is a parameter dependent upon an entrainment coefficient α, can be expressed by :

$$C = 4^{1/3} \left(1.2\alpha (0.9\alpha)^{1/3} \pi^{2/3} \right) \tag{3}$$

Experiments were conducted to quantify the volume flow rates using saline plumes [1]. Experimental saline plumes are not ideal plumes as assumed in the mathematical analysis since the initial momentum and volume fluxes can not be ignored. A plume with non-ideal source was proved to be equivalent to a pure point source plume, located at an asymptotic virtual source with a distance $x = -x_{avs}$ below the actual source at $x = 0$ (Figure 1). In [2] the reduced gravity in the warm buoyant layer is defined as

$$g' = g\Delta\rho / \rho_o \tag{4}$$

and the relative strength of buoyancy and wind is measured by a dimensionless quantity Fr, expressed by

$$Fr = \sqrt{(\Delta / \rho_o) / (B / H)^{2/3}} \tag{5}$$

where Δ is the pressure drop induced by wind across the space (Figure 1)
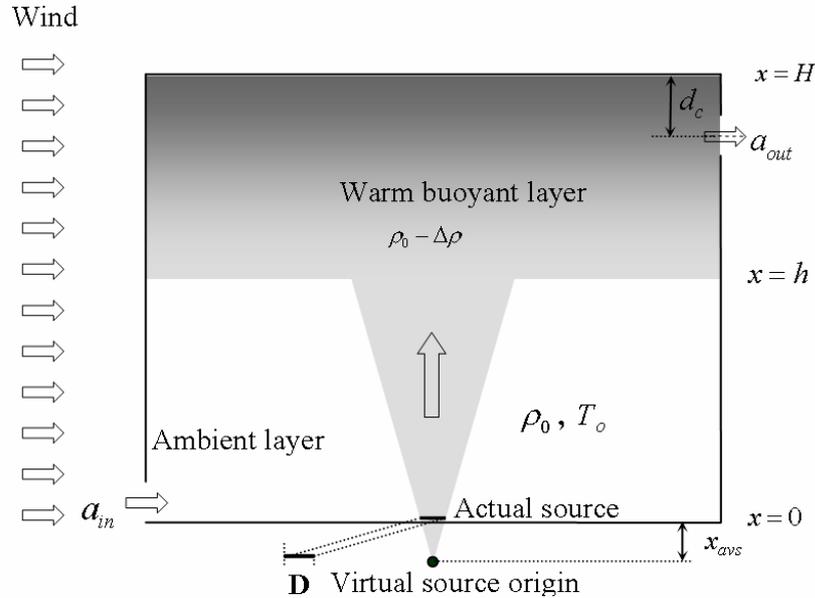
Figure 1 Schematic figure showing the locations of the actual source with diameter D ($x = 0$) and the virtual source origin ($x = -x_{avs}$).

Dimensionless results of the experiments without corrections of the virtual origin x_{avs} are shown in Figure 2 together with the theoretical prediction (equation 1) at different interface heights h (with volume flow rate $Q_h(x=h)$). The measurements of volume fluxes are linearly over-predicted by the experiments due to the initial conditions of the saline plumes.
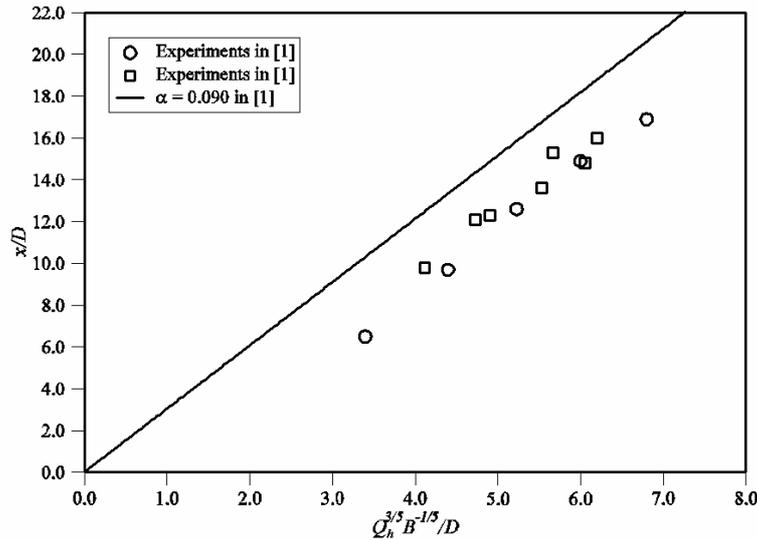
Figure 2 Experimental measurements for a turbulent plume with uncorrected virtual origin of the source. Theoretical prediction is plotted for α =0.090 (equation 1, after [1]).

Although the experimental conditions considered in [1] are buoyancy-driven alone cases the data plotted here can be used to validate natural ventilation flow rates induced by buoyancy combined with wind effects [2] due to the same assumption: for both cases, the volume flow rates through the space are equal to the volume flow rates through the plume at the interface (Figure 1, $x=h$), which is the basis of the experiments in [1].

3. The CFD model

The cases considered for CFD simulations are buoyancy combined with wind (Figure 1), the CFD model is 10 times greater than the perspex box used in the experiments [2] (Figure 3). The flow inside the enclosure in reality is mixed by conduction, convection, radiation and mass and heat transfer. In this work, conductive and radiation effects are neglected.
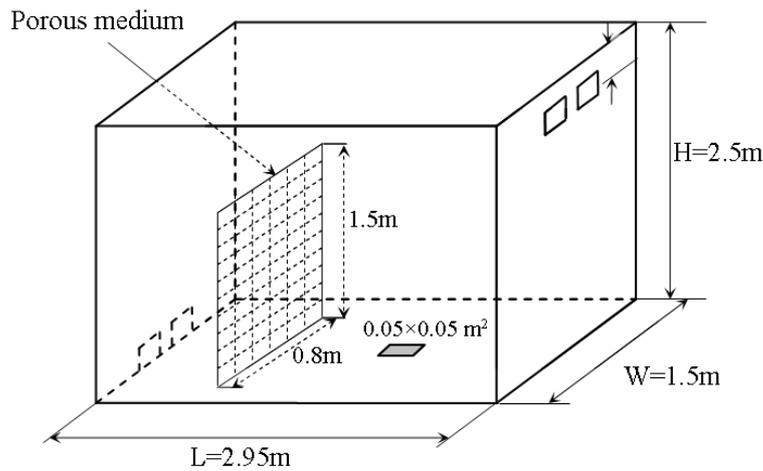
Figure 3 Numerical fluid domains for CFD simulations

3.1 Turbulence model

The eddy-viscosity two equation models are generally feasible to model indoor air flows, especially when the enclosure is not well occupied (empty box for this study) and the first order fluid parameters are of the main interests (e.g. mean temperature, velocity etc.) rather than turbulent fluctuation details [4]. Previous studies suggested that the Renormalisation Group (RNG) k-epsilon model [7] performed better than other eddy-viscosity models in the prediction of indoor airflow features [4, 8]. In this work, the RNG k-epsilon model with Boussinesq approximation was applied to conduct all the numerical simulations.

3.2 numerical methods

The software used here was CFX4, version 4.4 (2001) [9], in which the hydrodynamic equation set is solved in a segregated manner. The enclosure is an empty rectangular box, a Cartesian coordinate system with a structured mesh system is selected. Central differencing scheme is used to discretize the mass flow equation; hybrid scheme is employed for energy and momentum equations. Pressure coupling is treated by SIMPLEC algorithms and Rhie-Chow [10] is used to overcome the problem of decoupling due to co-located meshes.

A coarse mesh with 75,000 cells was used. The mesh was refined by doubling the cell number. y+ values were examined by adding it as a user scalar in the solver. In most wall areas, the y+ values of coarse and fine mesh are among 20~50, thus both meets the requirements of the RNG

model. The computed mass flow rates through the enclosure for the two mesh structures differed by 0.5%. The mean velocity at different locations also showed that increasing the resolution did not lead to significant differences (Figure 4).
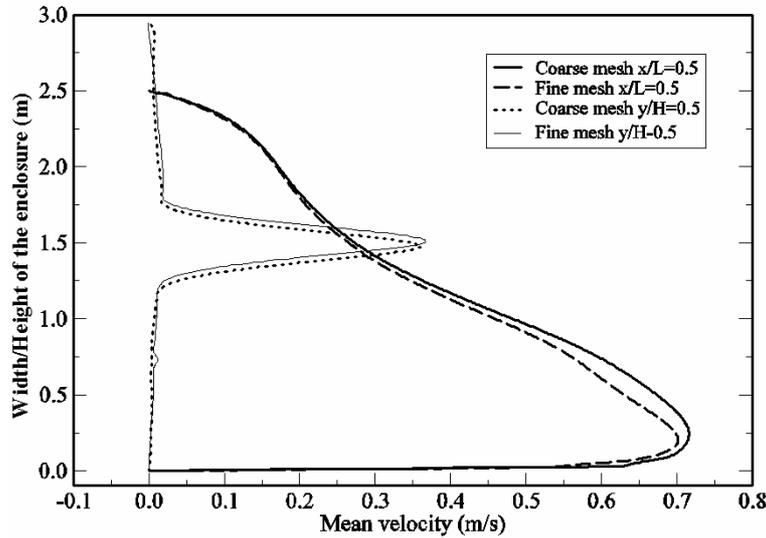
Figure 4 Mean velocities at different locations of the domain

3.3 Boundary conditions

The ceiling, the floor (except the heat source) and the side walls of the enclosure were modelled as WALL type boundaries with "no slip" condition and zero heat flux. The heat source was also modelled as WALL type boundary but with a constant heat flux. A constant pressure boundary was imposed across the inlets and outlets to account for the wind effects. Considering the effects of contraction and expansion on the ventilation openings, coefficients of C_D (=0.6) and C_e (=0.5) were introduced in analytical models respectively [2]. These effects were modelled in CFD simulations by reducing the ventilation opening sizes [5]. A porous medium boundary was applied in the middle way between the inlet and the heat source (Figure 3) to represent the perforated plate used in the same way in the experiments [2]. Its resistance constants were selected using tries and errors. It is used to block the horizontal momentum to maintain an upright plume. In practice, ventilation systems are usually designed to minimise draughts in buildings, so the indoor airflow is commonly dominated by the thermal objects within the natural ventilated spaces.

4. Results and discussions

Simulations varying the heat source strength were conducted. The conditions are summarised as follows: source strength E (in W) = 2000, 500, 150, 100, 75, 50, 20 and 5 for each case, wind mean velocity U =0.65m/s for all cases [2]. Typical stratification contrasts between experiments and CFD agreed well (Figure 5).
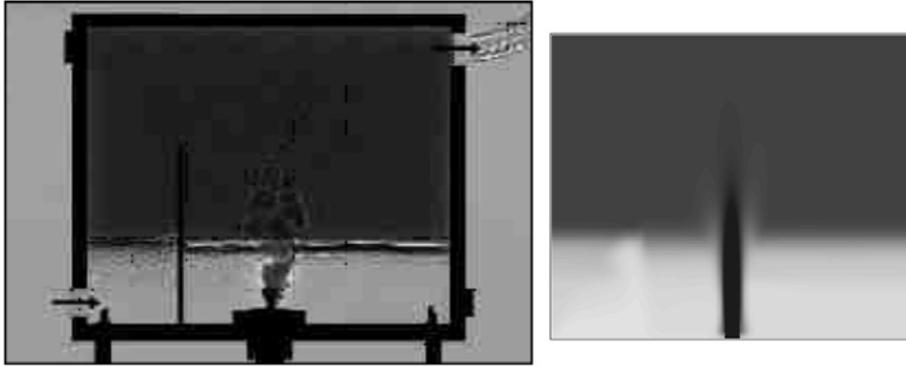
Figure 5 Typical flow patterns predicted by experiment (left) and CFD (right).

As observed in the CFD simulations, constant pressure boundary across an inlet opening produced a cluster of strong horizontal momentum. This momentum was partially blocked by the porous boundary to avoid it impinging on the thermal plume directly. The inflow air speed was therefore slowed down and its direction altered, making it isotropic after the porous boundary. The numerical uprising plume generated a relatively strong vertical momentum due to the continuous release of heat from the point source, which transported its surrounding air up to the top of the enclosure (Figure 6).
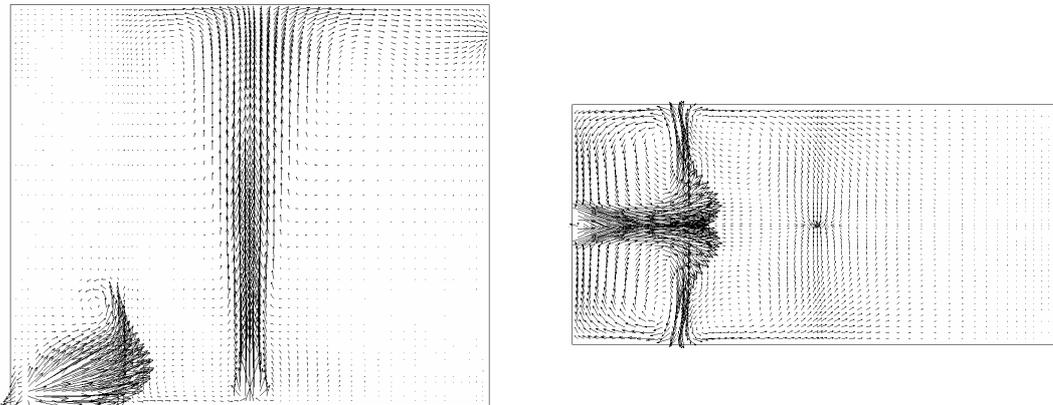
Figure 6 Speed vectors of the domain. Section view (left) and plan view (right).

From Figure 6, although the physical flow nature (contraction and expansion) at the openings was not captured due to the constant pressure boundary with reduced opening sizes it can be negligible when the flow rates through the space are of the main interest.

If the air surrounding the plume was isotropic with uniform speed the numerical entrainments induced by the vertical momentum would be the same from all directions, then the numerical plume could be perfectly developed and maintained as the dynamics assumed in theory. However, this was not approachable with the application of the porous medium boundary condition. The boundary can only minimise the effects of horizontal momentum to a level at which buoyancy induced vertical momentum begins to dominate the flow. Therefore, the stronger the heat sources, the more stable the uprising plume, and the closer the prediction between theories and CFD (Figure 7, 8).

The simulation results for ventilation flow rates through the fluid domain are plotted on the theory prediction lines (equation 1 at $x = h$, $\alpha = 0.09$ for theory [1], $\alpha = 0.083$ for theory [2]) together with the experimental measurements corrected by virtual source origin x_{avs} (Figure 7).
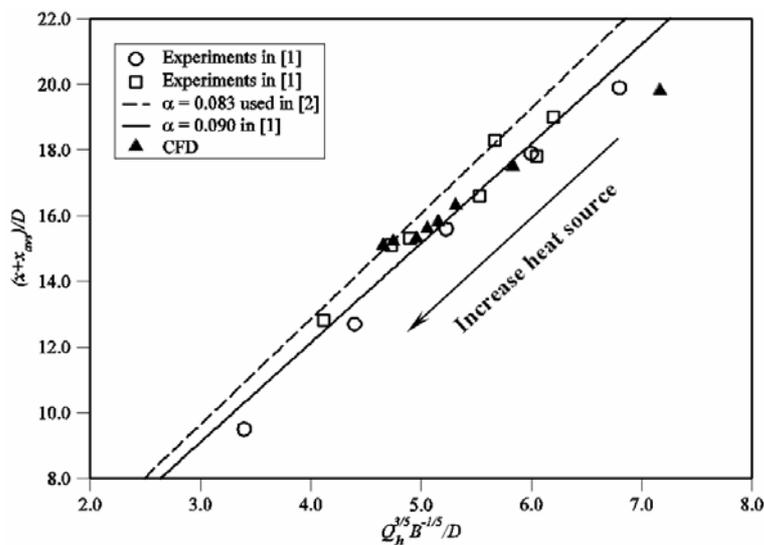
Figure 7 Flow rates predicted by theory, experiments and CFD.

CFD predictions closely agreed with both the mathematical model and experiments except the 5 watts heat input (the buoyancy force is too small to get a clear stratification level). The CFD data points on the reference line ($\alpha = 0.083$, used in [2]) showed the closest prediction of the entrainment into the thermal plume assumed by [2] at high heat inputs.

- 8 -
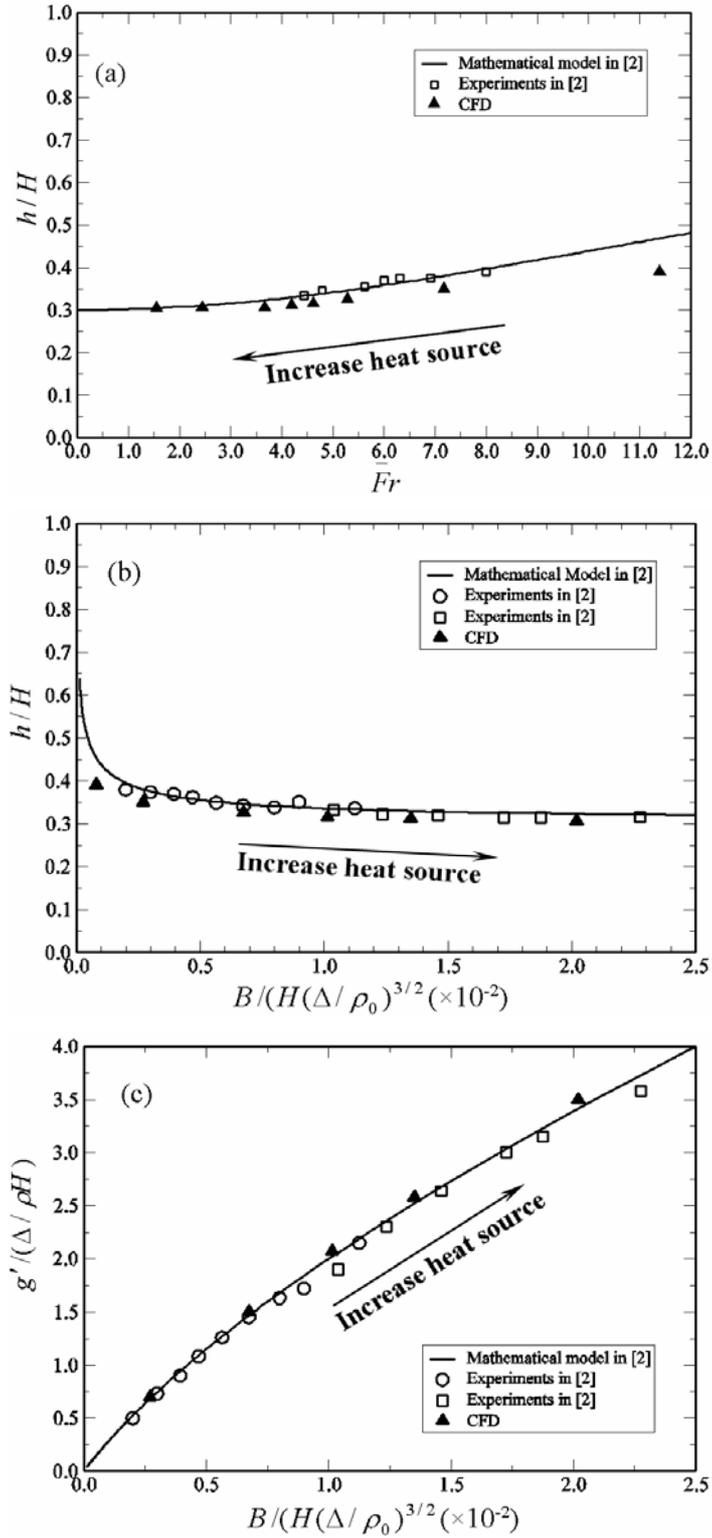
Figure 8 Predictions of analytical model, experiments [2] (with α =0.083) and CFD for Fr vs h/H, $B/(H(\Delta/\rho_0)^{3/2}$ vs h/H and $B/(H(\Delta/\rho_0)^{3/2}$ vs $g'/(\Delta/\rho H)$.

The CFD predictions also agreed the theory in [2] for varying buoyancy fluxes on the effect of the interface prediction (Figure 8a, b) and the reduced gravity (Figure 8c). Again, the strong heat sources gave more accurate predictions than small heat inputs when compared with theory.

5. Conclusions

This work illustrates the ability of CFD for modelling natural ventilation driven by combined wind and buoyancy. The wind force was arranged to assist the buoyancy and the buoyancy force was varied for different heat loads in the space. The comparisons of volume flow rates through the space, flow patterns, interface positions and the reduced gravities (induced by buoyancy forces) between the theoretical model, experiments and CFD have been made and close agreement was achieved.

The application of pressure boundary conditions with reduced physical opening sizes proved to be a practical way to model the ventilation loses due to expansion and discharge across openings. Compared with the modelling technique used in [8] this method has the advantage of lower computing cost. In the work, the computing cost was further reduced by calculating half of fluid domain. Both mathematical models [1, 2] are based on a vertical buoyant plume, porous medium boundary was essential for maintaining an upright plume for numerical studies. The parameters of the porous medium were selected using trials and errors. This was feasible for this study since one pair of settings was suitable for all the simulations with the same mesh size.

The close agreement between the three modelling techniques indicates that, with proper considerations of the boundary conditions, CFD is able to predict the natural ventilation parameters (ventilation rates through the space, thermal stratification levels, temperature gradients, pressure fields) with details on the ventilated enclosure considered in this study. Compared with the experimental studies on this type of flow, CFD is much less labour-intensive and time consuming.

References

[1] Hunt GR and Kaye NG. "Virtual origin correction for lazy turbulent plumes". J. Fluid Mech., 435, pp.377-396, 2001.

[2] Hunt GR. and Linden PF. "Steady-state flows in an enclosure ventilated by buoyancy forces assisted by wind" J. Fluid Mech. vol. 426, pp. 355-386, 2001.

[3] Linden PF., Lane-Serff G. and Smeed DA. "Emptying filling boxes: the fluid mechanics of natural ventilation" J. Fluid Mech. Vol. 212 pp. 309-335, 1990.

[4] Chen Q. "Comparison of different k- models for indoor air flow computations" Numerical Heat Transfer, Part B, 28; pp. 353-369, 1995.

[5] Cook MJ, Ji Y & Hunt GR. "CFD modelling of natural ventilation: combined wind and buoyancy forces". International Journal of Ventilation, vol. 1, pp. 169-180, 2003.

[6] Morton BR, Taylor GI & Turner JS. "Turbulent gravitational convection from maintained and instantaneous sources". Proc. Roy. Soc. A 234. 1-23, 1956.

[7] Yakhot V, Orszag SA, Thangham S, Gatski TB and Speziale CG. "Development of turbulence models for shear flows by a double expansion technique", Phys. Fluids A, 4(7), pp. 1510-1520, 1992.

[8] Cook MJ and Lomas KJ "Evaluation of two eddy viscosity turbulence models for predicting buoyancy driven displacement ventilation. BSERT, 19 (1), pp. 15-21, 1998.

[9] CFX "User Guide Version 4.4", CFX International, Harwell, UK, 2001.

[10] Rhie CM and Chow WL. "Numerical study of the turbulent flow past an airfoil with trailing edge separation". American Inst. Aeronautics and Astronautics J. 21 (11), pp. 1527-1532, 1983.