ATRIUM-ASSISTED NATURAL VENTILATION OF MULTI-STOREY BUILDINGS

Y Ji* and MJ Cook

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

ABSTRACT

Buoyancy-driven displacement natural ventilation in a simple multi-storey building structure is investigated. The storeys are connected to a common atrium and top-down-chimneys, and the internal heat gains of the storeys are represented by localised point heat sources on each floor. The heat sources generate thermal plumes which entrain the surrounding air and transport warm air upwards. Simple analytical models are used to describe the main flow features, such as interface height, temperature gradient and ventilation flow rate. A two-storey building structure is also investigated using computational fluid dynamics (CFD). Using the RNG k-epsilon turbulence model the predicted airflow patterns, temperature profiles and ventilation flow rates agreed favourably with the analytical models. The work demonstrates the potential of using CFD for modelling buoyancy-driven displacement ventilation in complex building structures and the accuracy that can be expected.

INDEX TERMS

Atrium, Natural ventilation, buoyancy, CFD, RNG turbulence model

INTRODUCTION

Natural ventilation has been the subject of much research over recent decades due to its potential for offering good indoor air quality for occupants and relatively low energy usage compared with mechanical ventilation. The driving force for passive natural ventilation is generated by the temperature difference between the inside and outside of a building (known as the stack effect). Architectural features such as atria can be used to enhance the stack effect.

To study natural ventilation flow in buildings at the design stage it is useful to use modelling techniques to investigate the possible ventilation flow rates, temperature gradients and stratification within the ventilated spaces. For example, simple analytical models have been developed to study natural ventilation in a single and connected space by Linden et al (1990) and Holford & Hunt (2003) respectively. The work of Hunt & Holford (1998) investigated natural ventilation flow in multi-storey spaces connected to a tall atrium with top-down-chimneys (TDCs) used to bring fresh air into the spaces from high level. These analytical models were validated by small scale experiments and used to offer a better understanding of the general principles of natural ventilation. As an alternative modelling technique, CFD has been increasingly used to model natural ventilation flows. The work of Cook & Lomas (1998), Ji et al (2004) showed that CFD was successfully used to model natural ventilation flows in a single space and a single space connected to a tall atrium. The simulation results were compared with both the analytical models and the salt bath experiments, and favourable agreement was achieved. However, there was only one airflow path in the flow systems investigated. In order to further investigate the performance of CFD for modelling natural ventilation flow in multiple spaces with more than one airflow path simultaneously a general analytical model for multi-storey spaces connected to a tall atrium is presented in this paper. The model is then used to verify the CFD predictions for this type of natural ventilation flow. The intention of this research is to offer a general analytical model for multiple ventilated spaces connected to a common atrium and how best to model this type of flow using CFD.

THE ANALYTICAL MODEL

A natural ventilation flow system of a two-storey building connected to an atrium is shown in Figure 1. For each storey, the air flow path includes a TDC inlet, a TDC, a storey inlet, a storey outlet and a common atrium outlet. These are denoted by the labels $i=1$ to 5 with cross-sectional areas of a_{j1} to a_{j5} respectively (j =0, 1, 2, …, denotes the jth floor). The effective opening area of an individual opening in an airflow path and the total effective opening area for an individual storey may be written as follows:

* Corresponding author email: yingchun@dmu.ac.uk

$$A_{ji} = \sqrt{2} C_{ji} a_{ji} \text{ and } A_{js}^{-2} = \sum_{i=1}^{4} A_{ji}^{-2} \quad \text{(1a, b)}$$

where C_{ji} is used to account either dissipation losses (due to expansion or discharge) associated with individual ventilation openings or the friction losses along a TDC.

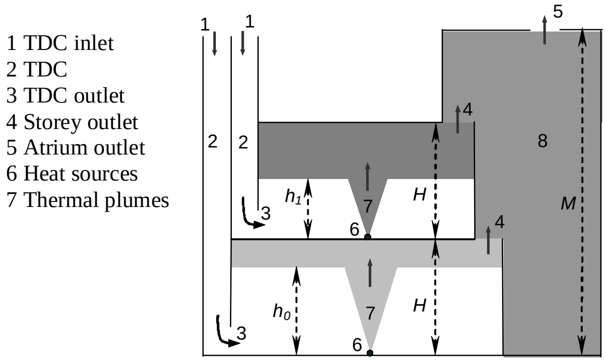

***Figure 1** Geometrical description of a two-storey building connected to an atrium*

Mathematical expressions for predicting the volume flux Q and the reduced gravity G' in a thermal plume generated by a localised point heat source with strength B as a function of the distance z from the source were given by Morton et al (1956). Assuming zero initial momentum and volume flux, Q and G' are expressed as:

$$Q(x,B) = C(Bz^5)^{1/3} \text{ and } G'(x,B) = (B^2 z^{-5})^{1/3} / C \quad \text{(2a, b)}$$

Using the plume theory of Morton et al (1956), Hunt & Holford (1998) presented design curves for sizing ventilation openings in natural ventilation systems in multi-storey buildigns. A simultaneous equation to determine the ventilation flow in each storey was also given by conservation of volume and heat fluxes. Extended from Hunt & Holford (1998) a simultaneous equation which determines the interface heights (h_j) of a multi-storey building may be written as follows when the heat input of each storey is the same:

$$C^3\left(A_{js}^{-2} h_j^{10/3} + A_{j5}^{-2} \left(\sum_{j=0}^{n-1} h_j^{5/3} \right)^2 \right) = h_j^{-5/3}\left(H_j - h_j\right) + n\left(M - \sum_{k=0}^{j} H_k\right) / \sum_{j=0}^{n-1} h_j^{5/3} \quad (3)$$

where C=0.143, a constant related to the plume entrainment coefficient. Equation (3) shows that when natural ventilation is driven solely by single localized heat sources, the interface heights, separating the buoyant upper layer (T_j) and the lower layer at ambient temperature (T_0), is independent of heat input, and depends only on the geometrical factors (the storey height H_j, the stack height M and the effective areas of the storey A_{js} and the atrium A_{j5}) and the constant C.

Considering an n-storey building with constant storey height ($H_j = H$) connected to a tall atrium where the ventilation strategy is to achieve the same ventilation flow rate for each storey, a simple analytical model for the non-dimensional interface height x_j $(= h_j / H_j)$ for the jth storey can be obtained using equation (3)

$$A_{jt} / H^2 = \left(C^3 x_j^5 / ((M - jH)/H - x_j)\right)^{1/2} \text{ where } A_{jt}^{-2} = A_{js}^{-2} + n^2 A_{j5}^{-2} \quad \text{(4a, b)}$$

Equation (4) gives a general description of natural ventilation flow driven by a localised point heat source in a single space or connected spaces, e.g. for a single space ($M = H$, j =0) or a single space connected to an atrium ($M > H$, j =0) the model equations (4a, b) is consistent with the models of Linden et al (1990) for a single space and Holford & Hunt (2003) for a single space connected to a tall atrium.

Using the same approach used in Holford and Hunt (2003) the non-dimensional forms of the volume flow rate and the reduced gravity (g'_j also equals to $g(T_j - T_0)/T_0$) at the interface height can be written as follows:

$$Q_{js}/CB^{1/3}H^{5/3} = (h_j/H_j)^{5/3} \text{ and } g'_j/B^{2/3}H^{-5/3} = (h_j/H_j)^{-5/3}/C \qquad \text{(5a, b)}$$

As an illustration, the model predictions are shown in Figure 2 (with $M = 4H$).

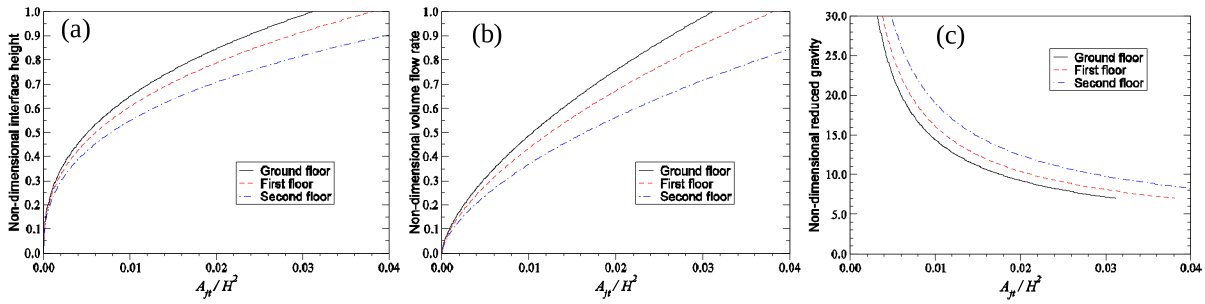

***Figure 2** Analytical model predictions of non-dimensional interface heights, volume flow rates and reduced gravities for ground, first and second floors of a multi-storey building.*

As shown in Figure 2a, increasing the total effective opening areas for a storey causes a rise in the interface height for that storey. Interface heights on the other storeys are different when the total effective opening area is the same for each storey due to the different stack effect acting. This implies that in order to achieve the same ventilation flow rate for each storey (with the same interface height) the total effective opening areas of each storey should be different. The higher the storey is, the less stack effect is provided by the atrium, so the larger the total effective opening area that is needed. The same trend can be examined in Figure 2b for the ventilation flow rates, while the reduced gravity decreases with the increase of the total effective opening areas (Figure 2c).

THE CFD MODEL

A CFD model of a simple two-storey building connected to an atrium was constructed. In the CFD model in this work (Figure 3) the key dimensions are $M = 9$m and $H = 3$m. In reality the flow inside the connected spaces is mixed by conduction, convection and radiation effects. In this work, conduction and radiation effects were neglected to meet the assumptions of the mathematical models. Ambient temperature was T_0 =15°C.

Following the successful modelling of natural ventilation flow in a single space (Cook & Lomas 1998, Ji 2004) and connected spaces (Ji et al 2004) the RNG k-epsilon turbulence model with Boussinesq approximation for buoyancy effects is again used to conduct the numerical simulations.

Pressure boundaries are set at the ventilation openings which connect to the exterior air (TDC inlets and atrium outlet). This means that unity can be assumed for the discharge coefficient at these locations (C_{j1} =1, C_{j5} =1) because the numerical airflow does not experience dissipation losses across the pressure boundaries. The cross section area of the TDC is much smaller than the cross section area of the storey therefore $C_{j3} \approx 1$ (Nakayama & Boucher 1999, pp 119). For the openings at location 4, a value of $C_{j4} \approx$ 0.63 was assumed.

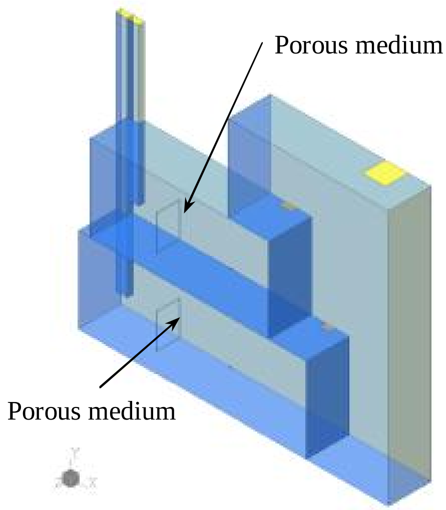

***Figure 3.** An illustration of the fluid domain (half) used in the CFD model*

In the work of Hunt & Holford (1998) the loss coefficient along the TDC due to the friction losses were determined by the ratio of the length (L) and diameter (D) of the TDC and a friction factor f to give $C_{TDC} = D/(4Lf)$. f was treated as a constant. There was only a minor effect (less than 2%) due to C_{TDC} over the calculation of the total effective opening areas and therefore neglected in the work of Ji et al (2004). In this study, the effect of TDC friction losses can be large due to the high ratio of L/D therefore can not be ignored. In order to accurately calculate the volume flow rate through each storey to meet the predefined criteria (same ventilation flow rates) the friction factor f is not treated as a constant but related to the Reynolds number of airflow inside the TDCs. An empirical equation, $f = 0.0008 + 0.05525\,\mathrm{Re}^{-0.237}$ (Nakayama & Boucher 1999, pp 116) is introduced to calculate the friction factor of the TDC in which the Reynolds number (Re=UD/n) can be calculated using the airflow velocity U, the hydraulic diameter D of the TDC and the viscosity (n = 14.61×10^{-6}m^2/s at T_o =15ºC). The frictional loss coefficient C_{j2} along the TDC is then determined by the friction factor f_j ($C_{j2} = \sqrt{D_j/(4L_j f_j)}$, Hunt & Holford 1998). For a given total effective opening area (A_{jt}) the interface height (x) and the flow rate (Q_j) at this height can be obtained using the analytical equations (4a and 2a). The flow rate is then used to estimate the airflow velocity ($U_j = 4Q_j/pD_j^2$) within a TDC, and then the Reynolds number can be calculated. Individual opening sizes along each airflow path can then be obtained from equation 4b for use in the CFD simulations.

RESULTS AND DISCUSSIONS

It took about 5000 iterations for the cases studied in this paper to achieve convergence using a mesh of approximately 2.3×10^5 cells. The qualitative quantitative predictions of the CFD model are shown in Figures 5 and 6. The well developed vertical thermal plumes entrain their surrounding air and transport warm air into the upper layer. The porous boundary effectively prevents the incoming horizontal momentum from hitting the plume which assists the development of a vertical thermal plume to meet the assumption of Morton et al (1956). Interfaces are clearly formed which separate the warm upper layer from the cool ambient layer below (Figure 4b).

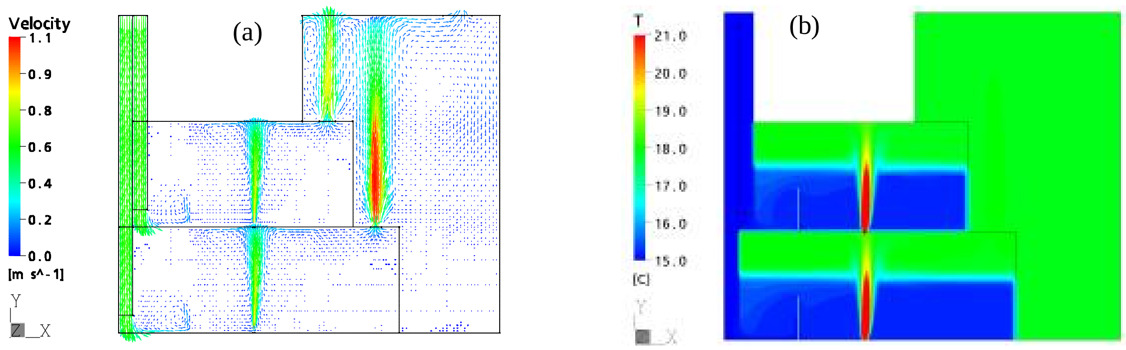

Figure 4 *Qualitative results of the CFD model (a) velocity and (b) temperature contour*

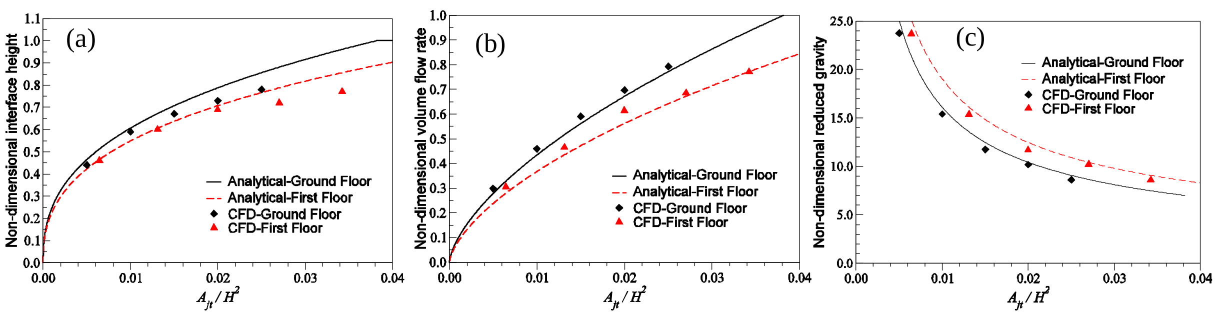

Figure 5 *comparisons between the analytical model and the CFD simulation results*

The quantitative results of CFD are compared with the predictions of the analytical model. The interface heights predicted by the CFD model agree well with the analytical predictions when the total effective opening areas are relatively small (Figure 5a). For large total effective opening areas the interface heights are clearly under-predicted. The similar performance of the CFD model in the prediction of the interface height for a single storey connected to a tall atrium was observed by Ji et al (2004). This is thought to be caused by the analytical assumption of a homogenous buoyancy layer above the interface for all total effective opening areas. In the analytical model complete ventilation (x_j =1.0) can be predicted for both floors when the total effective opening areas are increased to some critical value. However, in the CFD simulations, when the interface approaches the ceiling, the buoyant layer becomes unsteady and non-uniform. And a layer of warm air close to the ceiling always exists. The comparisons between the analytical model and the CFD simulations in the prediction of non-dimensional volume flow rates and reduced gravities show close agreement for all ranges of the total effective opening areas investigated (Figure 4b, c). From the analytical equations, when the interface height is under-predicted a lower volume flow rate and a higher reduced gravity may be expected (equation 5a, b). However, in the CFD simulations, the volume flow rate is determined by the physical opening sizes and the buoyancy source strength rather than the interface heights, while the reduced gravity of the ventilated space depends on the ventilation flow rate.

CONCLUSIONS

Atrium assisted displacement natural ventilation flow in a multi-storey space has been studied using both mathematical analysis and numerical simulations. The analytical model can be used to estimate the airflow rate, temperature gradient and the stratification level for a steady state flow. CFD simulations were conducted using the RNG k-epsilon turbulence model and the CFD results were compared with the predictions of the analytical model and showed favourable agreement. Discrepancies were observed in the prediction of interface height for large ventilation opening size which may be caused by the unsteady airflow in the upper layer when the interface approaches the ceiling. The use of a porous medium region was necessary for reducing the incoming horizontal momentum and assisting the development of a vertical plume.

This paper demonstrates how pressure boundary conditions can be imposed directly at the ventilation openings which connect to the exterior of the domain. Using this boundary condition, the physical dissipation losses across these openings were neglected and their effect was modelled using a physically reduced opening size. This paper also demonstrates how the friction factor is calculated in association with theoretical volume flow rates which enables the use of a suitable loss coefficient of the TDC in each floor for different cases.

ACKNOWLEDGEMENTS

This work is part of an EPSRC-funded project (GR/N37346). The financial support of the EPSRC is gratefully acknowledged. The authors are also grateful to Dr. Gary Hunt (Imperial College, London) for his guidance and discussions during this research.

REFERENCE

Cook MJ., Ji Y. and Hunt GR. 2003. “CFD modelling of natural ventilation: combined wind and buoyancy forces,” *International Journal of Ventilation*, vol. 1, pp. 169-180.

Cook MJ. and Lomas KL. 1998. “Buoyancy-driven displacement ventilation flows: Evaluation of two eddy viscosity models for prediction,” *BSERT*, Vol 19, No 1, pp 15-21.

Holford JM. and Hunt GR. 2003. “Fundamental atrium design for natural ventilation,” *Building and Environment*, Vol 38, pp 409-426.

Hunt GR. and Holford JM. 1998. “Top-down natural ventilation of multi-story buildings,” *19th Annual AIVC conference*, Oslo, Norway, pp. 197-205.

Ji Y. 2004. “Numerical studies on natural ventilation flow in an enclosure with both buoyancy and wind effect,” *CIBSE national conference*, London, UK.

Ji Y., Cook MJ. and Hunt GR. 2004. “CFD modelling of atrium-assisted natural ventilation,” *The 9th International Conference on Air Distribution in Rooms*, ISBN: 972-97973-2-3.

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

Morton BR., Taylor GI. and Turner JS. 1956. “Turbulent gravitational convection from maintained and instantaneous sources,” *Proc. Roy. Soc.* A 234. pp 1-23.

Nakayama Y. and Boucher RF. 1999. “An introduction of fluid mechanics,” ISBN 0340 676493.

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