

A comparison of predictive techniques for natural displacement ventilation of buildings

S. A. Howell [1] and I. Potts [2]
(1) Cundall, Johnston and Partners, Horsley House, Regent Centre, Newcastle-upon-Tyne, NE3 3LU.
(2) Department of Mechanical, Materials and Manufacturing Engineering, Stephenson Building, University of Newcastle-upon-Tyne, Newcastle-upon-Tyne, NE1 7RU.

Previous work by Linden, Lane-Serff and Smeed (1990) has developed a simple mathematical model for natural displacement ventilation of an enclosure. The work also introduced the experimental salt-bath technique, which uses salt solutions and fresh water to generate buoyancy forces that are analogous to those found in naturally ventilated buildings. The work claims that a good correlation exists between the predictions of the simple mathematical model and the results obtained using the salt-bath technique.

The present paper reports further, independent experimental work using a test enclosure with air as the working fluid. A Computational Fluid Dynamics computer package is also used to predict the flow through the enclosure. The potential of each of the predictive techniques investigated is discussed. Although a simple mathematical model would be desirable, the conclusion of this paper is that such a model is not suitable for use in a design situation, and that alternative predictive techniques are preferred.

1 Introduction

Concern about damage to the environment caused by excessive energy usage has caused a reawakening of interest in natural ventilation of buildings. The theory of natural ventilation, however, must be developed to provide the engineer with the tools to design a naturally ventilated building successfully.

Previous work by Linden et. al. (1990) has developed a simple mathematical model which describes the fluid mechanics of natural displacement ventilation of an enclosed space. A turbulent, buoyant, entraining plume will emanate from each heat source within the space, and the resulting convection currents maintain a steady flow of ambient air into the space via openings at low-level, displacing the warmer air, which leaves the space via openings at high-level. The mathematical model describes the displacement flow for a single source of buoyancy, and multiple equal sources of buoyancy at floor level. This work has recently been extended by Linden and Cooper (1996) to describe the mechanics of a displacement flow motivated by multiple sources of buoyancy of differing strengths at floor level. The mathematical models suppose that a stable stratification is established within the space between a layer of ambient air at low-level and a layer of fully mixed, warmer air at high-level. The model assumes that the two layers are separated by a well-defined interface.

A major implication of the existing work is that the height of the interface between the two layers of air is independent of the strength of the source of buoyancy, and is entirely dependent upon the dimensions of the space and upon the area of the openings. For the case of a single source of buoyancy at floor level in an enclosure of height H, the non-dimensional height of the interface $\xi = h/H$ is given by

$$\frac{A^*}{H^2} = C_q^{\frac{1}{2}} \cdot \sqrt{\frac{\xi^5}{(1-\xi)}} \quad , \qquad (1)$$

where A^* is an effective area of the openings to the enclosure, and the constant $C_q = \pi^2 \cdot (^3/_4) \cdot (^{6\alpha}/_5)^4$, where α is the entrainment constant for the plume. The effective area A^* is given by

$$A^* = \left(\frac{A_i \cdot A_o}{\sqrt{A_i^2 + A_o^2}} \right) \cdot \sqrt{2} \quad , \tag{2}$$

where A_i and A_o are the areas of the low-level inlet and the high-level outlet respectively. The model also predicts that as the area of the openings is increased, the volume flow rate will increase and the temperature difference across the interface will fall.

The salt-bath technique was also introduced to provide some experimental data for comparison with the predictions of the mathematical model. The procedure uses salt solutions and fresh water to generate density differences. The resulting buoyancy forces motivate a natural displacement flow that is analogous to those found in naturally ventilated buildings.

The work by Linden et. al. (1990) claims that a good correlation exists between the predictions of the simple mathematical model and the results obtained using the salt-bath technique.

The present paper presents the results of independent experimental work. A test enclosure was constructed and air was used as the working fluid. A point heat source was placed in the centre of the floor of the enclosure, generating a turbulent plume and the resulting displacement flow. Temperatures were measured using a thermocouple arrangement, and inlet velocities were recorded using the Laser Doppler Anemometry Technique, (described by Drain, 1980). A commercial Computational Fluid Dynamics computer package has also been used to predict the flow through the test enclosure.

Although a simple mathematical model would be desirable for design purposes, several deviations between the mathematical model of Linden et. al. (1990) and the ventilation of the test enclosure are identified. This paper recommends that such a simple mathematical model is not suitable for use in a design situation, and that alternative predictive techniques are preferred.

2 Symbols used

A^*	Effective area of the openings	m^2
A_i, A_o	Area at inlet/outlet	m^2
C_q	Constant for plume equations	Dimensionless
h	Height of interface above floor level	m
H	Height of ceiling of enclosed space	m
q	Volume flow rate	m^3/s
α	Entrainment constant	Dimensionless
$\xi = h/H$	Non-dimensional height of the interface	Dimensionless

3 Experimental Test Rig

The experimental test enclosure was essentially a Perspex box built around a wooden frame. The enclosure measured 1.0m × 1.0m × 0.6m. Although this was an order of magnitude smaller than any normal sized room, the mathematical model yields relationships in a non-dimensional form. Consequently, the mathematical model should be applicable to the experimental rig.

The openings at high-level and low-level were arranged symmetrically, in order to maintain the stability of the plume; an unsymmetrical arrangement tended to cause the plume to bend over to one side. A series of blank panels, 20mm × 100mm, were used to adjust size of each opening during successive experiments.

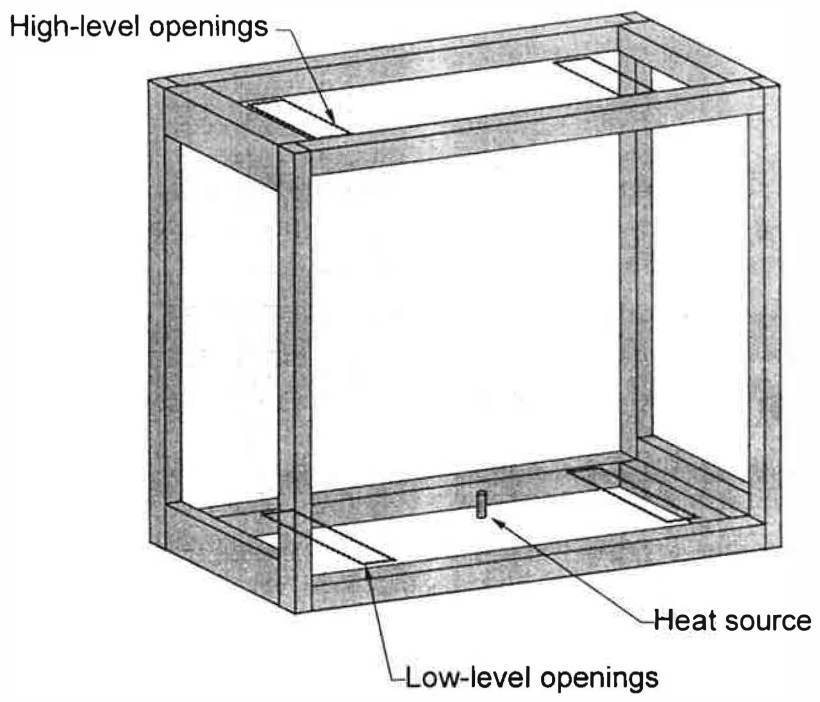

Figure 3.1 - Experimental test enclosure

A turbulent plume emanated from the heat source, and a displacement flow was established as illustrated in Figure 3.2. The inlet velocity was measured using the Laser Doppler Anemometry (LDA) technique. Temperature differences were recorded using a thermocouple arrangement, which comprises two thermocouple junctions: one that was traversed the height of the rig and the other that was fixed in the floor. Temperature differences were measured at four positions in the horizontal plane, in the region surrounding the plume.

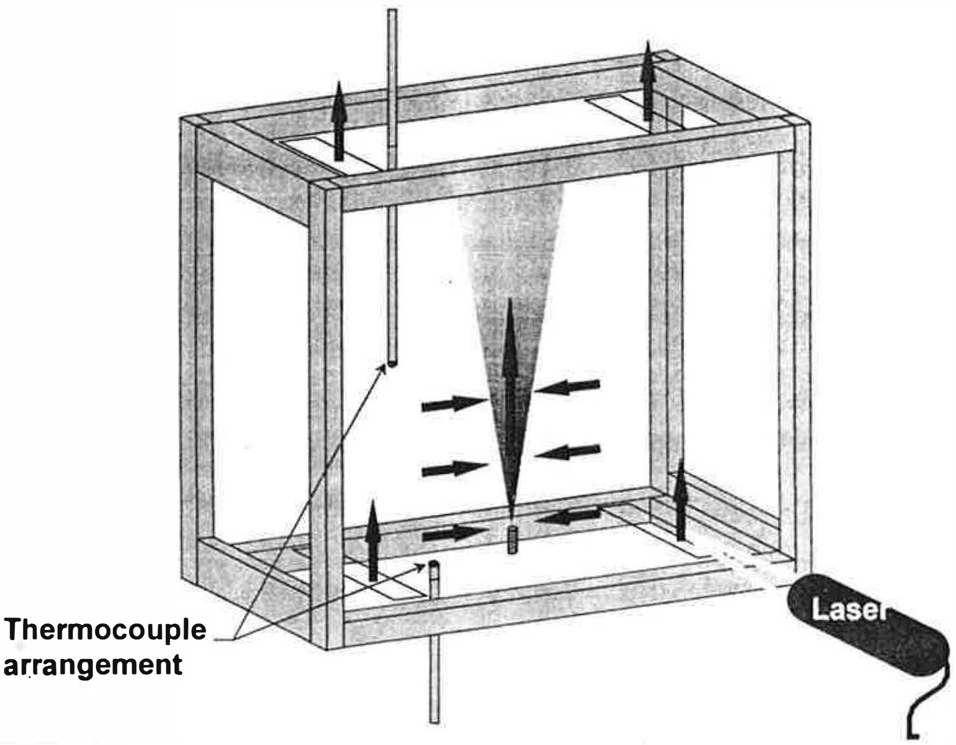

Figure 3.2 – Measurement of temperature differences and inlet velocities.

4 Computational Fluid Dynamics package, Fluent

The commercial Computational Fluid Dynamics computer package Fluent, version 4.31 was used to predict the flow through the test rig. Fluent uses the SIMPLE algorithm ofPatanker and Spalding (1972), together with a multi-grid solver for the pressure correction and enthalpy equations to solve a finite-volume discretisation of the continuity, Navier-Stokes and energy equations. Because of the symmetry of the test rig, only half of the rig was modelled, using 15312 computational cells. Constant pressure boundaries were used to model the inlet and outlet regions, whilst the plume source was provided by a region of boundary cells with specified heat flux. Initially, the flow through the test enclosure was modelled assuming entirely laminar flow. Later, the effect of applying the Renormalisation Group (RNG) turbulence model was also investigated. The RNG model defaults to laminar-like flow in regions of low turbulence, and is therefore recognised as the most appropriate turbulence model for laminar-turbulent transition flows such as the flow through the test enclosure.

5 Results

5.1 Experimental test enclosure

5.1.1 Temperatures

The temperature profiles measured are presented in Figure 5.1. It is observed that for all areas of opening, the temperature is approximately constant below a non-dimensional height of 0.4. Above this height, temperature appears to increase in a linear manner with height, although the actual temperature does fall with increasing area of the openings.

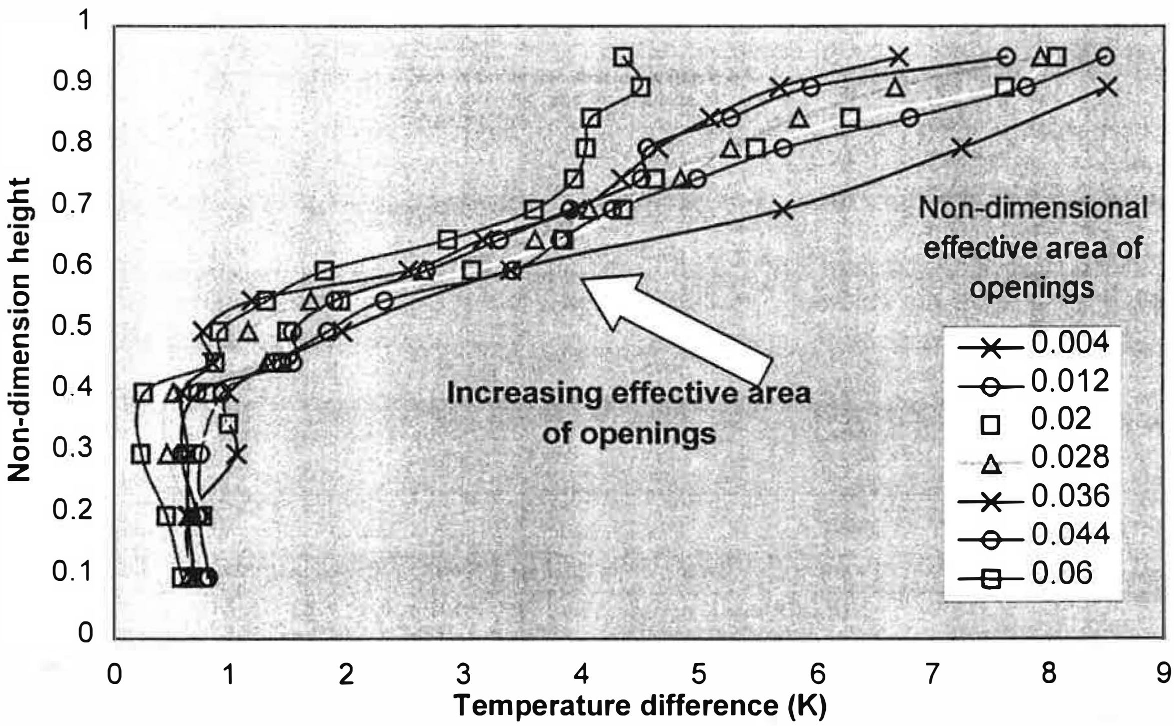

Figure 5.1 - Graph to show temperature profiles in the region outside the plume for each effective area of opening.

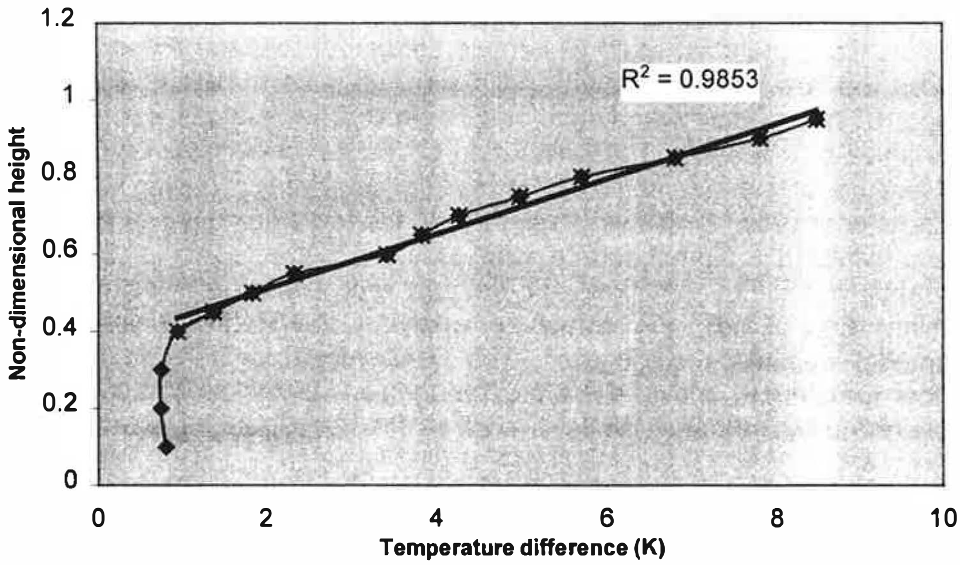

Figure 5.2 - Graph to show temperature profile in the region outside the plume for a non-dimensional effective area of opening A^*/H^2 of 0.012. This graph illustrates the linear nature of temperature rise above a non-dimensional height of 0.4.

5.1.2 Inlet velocity

The velocity at the centreline of the inlet is measured using the LDA technique. It is observed that the inlet velocity falls as the effective area of the openings to the enclosure is increased.

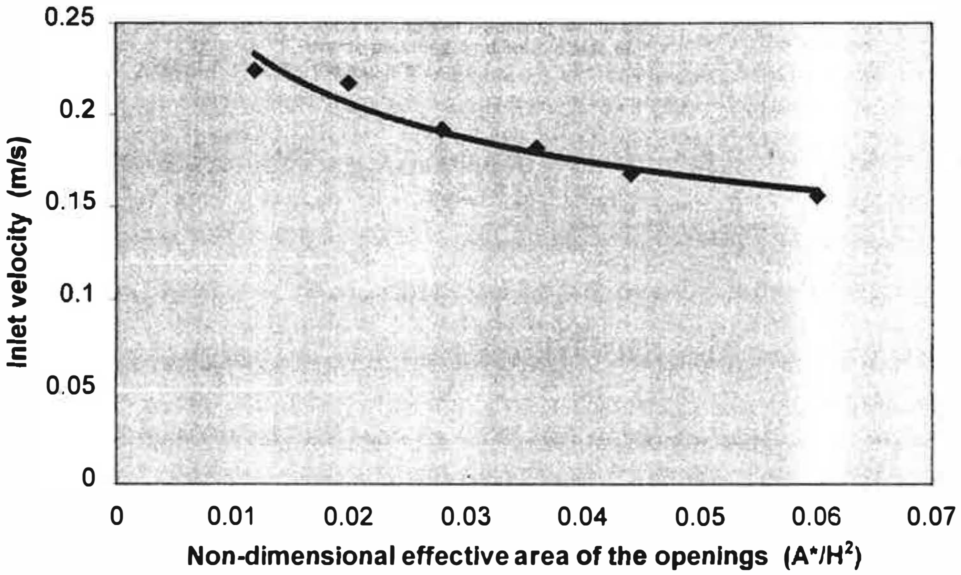

Figure 5.3 - Graph to show inlet velocity against effective area of the openings.

It is not valid to assume that the flow at inlet can be described by a 'top-hat' profile. As a consequence of this, it is not possible to use this data to predict volume flow rates through the enclosure. The data is useful, however, for a meaningful comparison to the CFD predictions presented in 5.2.2.

5.2 Computational Fluid Dynamics

5.2.1 Temperatures

The temperature profiles predicted by each of the CFD models for a non-dimensional effective area of the openings A^*/H^2 = 0.004 are shown in Figure 5.4, together with the actual temperature profile measured for that effective area of the openings. It is observed that the laminar CFD model, (no turbulence model used) yields the best prediction of temperature profile at this effective area of opening; the temperature rises only by a small amount below a non-dimensional height of 0.4, and above this the temperature rises in a linear manner with height.

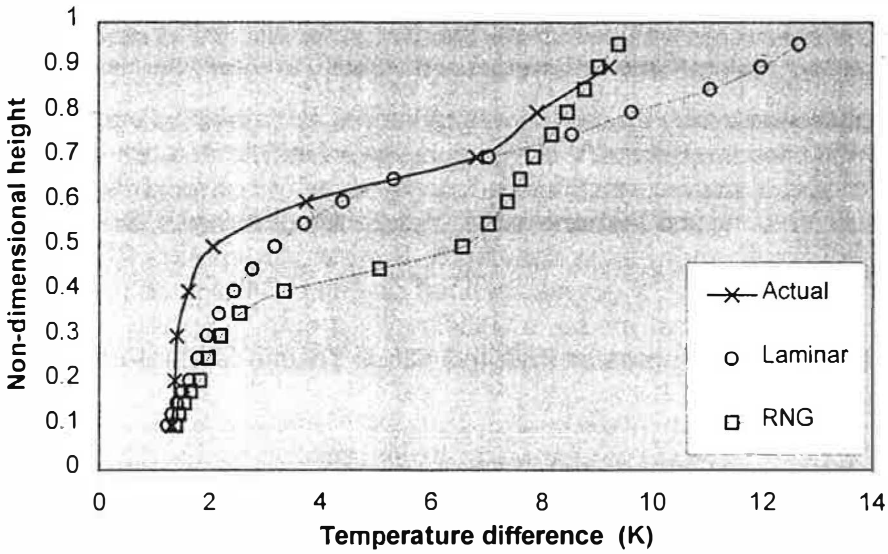

Figure 5.4 - Graph for comparison of the actual temperature profile measured, and those predicted using the CFD models, for a non-dimensional effective area A^*/H^2 of 0.004.

For the CFD predictions, if the effective area of the openings is increased, so is the height below which the temperature remains constant.

It is observed that for larger effective area of openings, the CFD model coupled with the RNG turbulence model will give the best prediction of temperature profile within the enclosure. Although this model predicts that the temperature will remain constant in the region outside of the plume below a non-dimensional height of 0.7 (compared to a non-dimensional height of 0.5 for the measured temperature), the RNG profile still exhibits the correct overall shape and a similar temperature gradient to the measured profile.

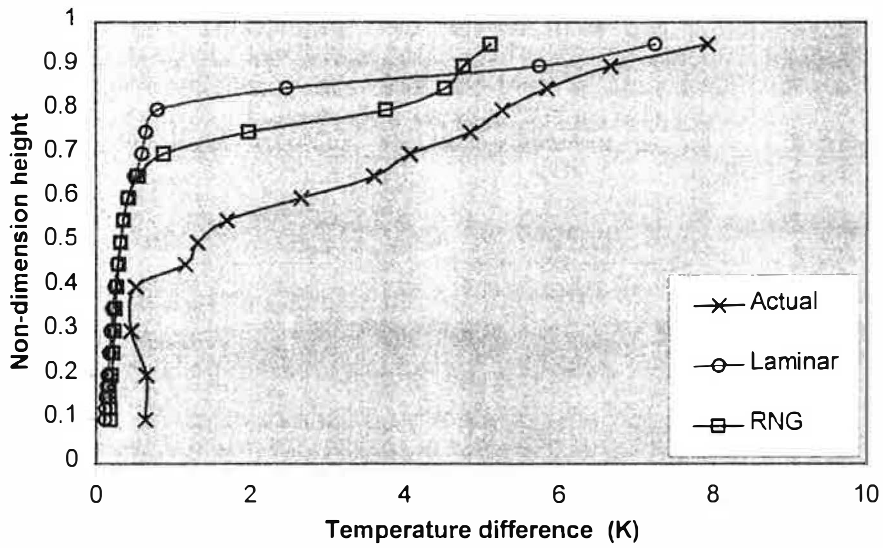

Figure 5.5 - Graph for comparison of the actual temperature profile measured, and those predicted using the CFD models, for a non-dimensional effective area A^*/H^2 of 0.030.

5.2.2 Inlet velocity

The CFD predictions for the air velocity at the centreline of the inlet are shown inFigure 5.6, together with the inlet velocity measured using the LDA technique. It is observed that for larger openings, the laminar and RNG CFD predictions both give good agreement with the inlet velocity measured. As the area of the openings is reduced, however, the CFD predictions deviate from the measured values. Of all the CFD models, it is the laminar prediction that gives the closest approximation to the measured velocity at inlet.

The inlet velocity predicted by the mathematical model gives a good agreement to the velocity measured at the inlet throughout the entire range of effective area of the openings. The mathematical model, however, assumes a top-hat velocity profile at the inlet, i.e. the vertical velocity of the air entering the test enclosure at the inlet opening is a constant, positive value above the opening only, and is zero elsewhere. Contrary to this, the CFD predictions suggest a significant variation in velocity across the inlet opening. Consequently, the volume flow rate predicted by the mathematical model is significantly greater than the volume flow rate of the CFD predictions, and will be greater than the actual volume flow rate through the enclosure. This is illustrated in Figure 5.7.

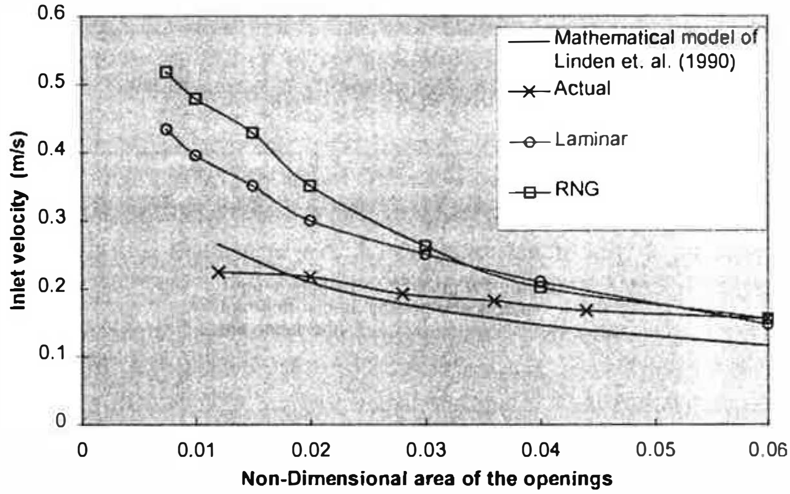

Figure 5.6 - Graph to show inlet velocity against non-dimensional effective area of the openings for experimental rig, each of the CFD predictions and that predicted by the mathematical model.

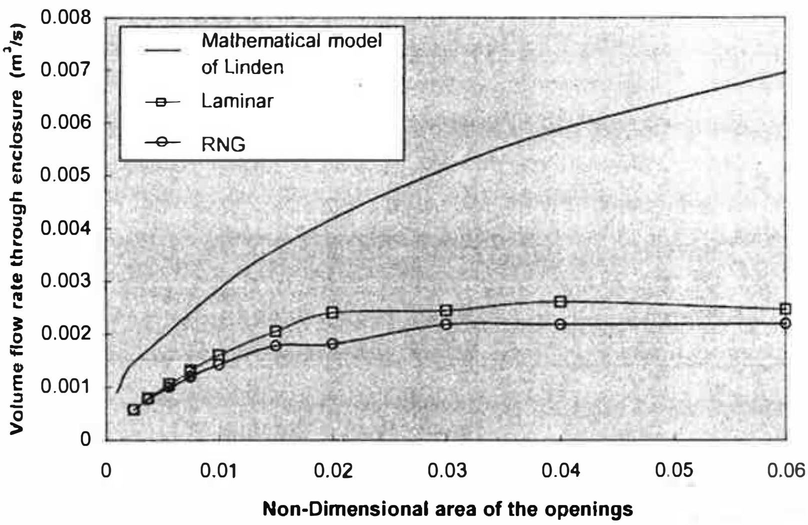

Figure 5.7 - Graph to show volume flow rate against non-dimensional effective area of the openings for each of the CFD predictions and that predicted by the mathematical model.

6 Discussion

The mathematical model presented by Linden et. al. (1990) suggests that a stable stratification will develop within the enclosure. A layer of cooler air at low-level will be separated from a fully mixed (i.e. constant temperature) layer of warmer air at high-level by a well-defined interface. Furthermore, the thickness of the layer of cooler air is entirely dependent upon the area of the openings to the enclosure.

From the experimental work presented in this paper it is observed that there does indeed exist a layer of cooler air at low-level. The air at high-level does not, however, appear to be fully mixed, but appears to exhibit an approximately constant temperature gradient. This contradicts one of the fundamental assumptions of the mathematical model of Lindenet. al. (1990). Also, the non-dimensional thickness of the layer of cooler air appears to remain constant at 0.4, and is therefore independent of the area of the openings to the enclosure.

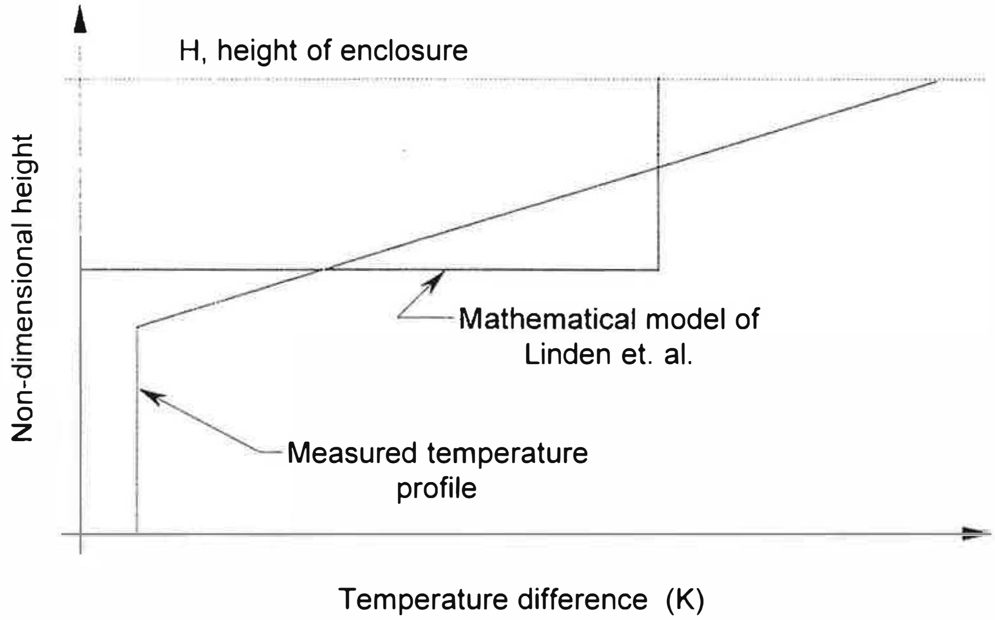

Figure 6.1 - Graph to show a comparison between the temperature profile predicted by the mathematical model and the actual temperature profile measured.

The CFD predictions can give reasonably accurate results for the flow through the enclosure. For small openings the best results are obtained from a laminar CFD prediction. For larger openings the best results are obtained from a RNG CFD prediction.

7 Conclusion

When designing a naturally ventilated building, the engineer must have reliable design tools and procedures. Whilst a simple mathematical model would be desirable, the model of Linden et. al. (1990) investigated in this paper cannot be used for the design of such buildings because the fundamental assumptions of the mathematical model appear to contradict what is observed and measured using a real experimental enclosure.

Acceptable predictions of natural ventilation flows can be obtained using CFD with existing turbulence models. Turbulence modelling, however, is still an area of intensive research and for buoyancy-driven flows, such as the flow discussed in this paper, there is still much development and validation work to be done. With further improvements in turbulence modelling and the continuing progress in affordable computing platforms, CFD will in the near future become the preferred way to predict and design for natural displacement ventilation.

8 References

LINDEN, P. F. , LANE-SERFF, G. F. AND SMEED, D. A. (1990).
Emptying filling boxes: the fluid mechanics of natural ventilation.

J. Fluid Mech. 212, 309-335.

COOPER, P. AND LINDEN, P. F. (1996).
Natural ventilation of an enclosure containing two buoyancy sources.

J. Fluid Mech. 311, 153-176.

LINDEN, P. F. AND COOPER, P. (1996).
Multiple sources of buoyancy in a naturally ventilated enclosure.

J. Fluid Mech. 311, 177-192.

MORTON, B. R., TAYLOR, G. I., AND TURNER, J. S. (1956).
Turbulent gravitational convection from maintained and instantaneous sources.

Proc. R. Soc. Lond. A234, 1-23.

WILCOX, D. C. (1994).
Turbulence modelling for CFD.

DCW Industries, 460 pp.

PATANKER, S. V., AND SPALDING, D. B. (1972).
A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows.

Int. J. Heat and Mass Transfer, Vol. 15, 1787→

FLUENT (1997).
Users Manual, Version 4.4.7

Fluent Europe Limited, Sheffield, UK.

SCHILD, P. G., TJELFLAAT, P. O., AND AIULFI, D. A. (1995).
Guidelines for CFD modelling of atria.

ASHRAE Technical Data Bulletin, Volume 11, Number 3: Aspects of atrium design.

DRAIN, L. E. (1980).
The Laser Doppler Technique.

John Wiley & Sons, 241 pp.