INNOVATIONS IN VENTILATION TECHNOLOGY

21ST ANNUAL AIVC CONFERENCE
THE HAGUE, NETHERLANDS, 26-29 SEPTEMBER 2000

AIRFLOW THROUGH LOUVERS: AN EXPERIMENTAL AND CFD STUDY

Steve Sharples* and Amjed Maghrabi+

*** Centre for the Built Environment, School of Environment and Development, Sheffield Hallam University, Sheffield S1 1WB, UK**

+ School of Architecture, University of Sheffield, Sheffield S10 2TN, UK

SYNOPSIS
In this study a series of parametric laboratory measurements were made of the velocities outside, v_e, and inside, v_i, a full-scale louver positioned in a rectangular opening located in a vertical wall forming part of a 1m x 1m x 2m deep model room. The louver parameters examined, for external incident wind speeds from 0.6 to 2.5 m/s, included louver blade depth (L), aperture (d) and distance from the louver in to room (R_d). For this initial study the blades of the louvers for each experiment were horizontal with an inclination angle θ of 0^o. It was concluded that louver aperture dimensions and distance from louver both played major roles in determining the magnitude of the velocity drop, whilst the effect on v_i/v_e of louver depth dimensions was smaller. Experimental results were compared against velocity drop values predicted by the computational fluid dynamics (CFD) program FLUENT. A comparison between the experimental data and the CFD predicted results showed a generally good agreement for positions close to the internal surface of the louver. Further in to the room the comparisons showed a greater divergence.

INTRODUCTION
In recent years there has been a renewed interest in the use of natural ventilation systems in buildings as a sustainable means of supplementing or replacing mechanical ventilation and air conditioning (see, for example, NatVent[1]). For large, non-domestic buildings, in particular, the natural ventilation inlets will often consist of large wall openings with louvered covers. Night-time ventilation of buildings can also make use of secure louvered openings.

A recent study of through-the-wall ventilators was performed at the Building Research Establishment (White *et al*[2]). This indicated that interactions between components in the ventilators, such as louvers, insect screens and internal flowpaths, meant that the airflow performances of the ventilators were highly variable and not easily described by standard flow equations. One way of gaining a better understanding of these interactions is through analysis of the performance of the individual components. For louvers there is only limited design information on the airflow performance of louvers and how louver parameters (such as blade depth, blade thickness, spacing between blades and blade orientation) interact to produce the overall louver flow and velocity characteristics. This study sort to gain a better understanding of some of these interactions.

A number of researchers have investigated the effect of various window opening parameters, such as size, location and orientation, on improving indoor airflow quantity and distribution, for example Brandle and Boehm[3], Jong and Bot[4] and Sobin[5]. Others have investigated the specific mechanisms of airflow through louvers and the various parameters governing indoor air motion such as louver depth, aperture and inclination angle. Yakubu and Sharples[6] and Pitts and Georgiadis[7] focused on pressure drops across the louvers. Recently, Maghrabi and Sharples[8] investigated the influence of various louver configurations on pressure flow characteristics (ΔP) under controlled volume-flow rates (Q). They used a pressurised room chamber technique, originally developed by Baker *et al*[9], to investigate airflow through cracks. Recent louver airflow work was carried out by Oliveira and Bittencourt[10], who used louvers with dimensions similar to those of the present study.

However, in their work no consideration was given to the reduction of airflow due to the room depth and louver depth

This study describes part of a current research project to investigate the overall airflow characteristics of louvers. The louver parameters, which were examined under a series of wind speeds ranging from 0.6 to 2.5m/s, included louver depth L, aperture opening height d and room depth R_d. For this series of tests all of the louvers investigated were kept horizontal (i.e. inclination angle $\theta = 0^o$). The aim of the experimental study was to investigate any possible relationships that might exist between the velocity drops across the louver opening and the parameters given above. The laboratory results were then compared with values predicted from the computational fluid dynamics package FLUENT.

METHODOLOGY

Ten full-scale louver models with different configurations and a fixed frame measuring 0.48m high by 0.32m wide were constructed. The louver blades were 10mm thick and were made of smooth pinewood in order to reduce any surface friction effects. The louvers were adjusted to a horizontal position with an inclination angle $\theta = 0^o$. Three louvers depths (L) were chosen: 0.04, 0.06 to 0.08m. Six blade gap dimensions (d) were used: 0.01, 0.02, 0.03, 0.035, 0.05 and 0.07 m (see Table 1 and Figure 1).

Table.1: Louver configurations used in the study

Louver Number	Blade Depth L (m)	Blade Aperture d (m)	Number of Blades N	Porosity (p%)
1	0.04	0.01	24	50
2	0.04	0.02	16	66
3	0.04	0.03	12	75
4	0.06	0.02	16	66
5	0.06	0.03	12	75
6	0.06	0.035	11	77
7	0.06	0.05	8	83
8	0.08	0.03	12	75
9	0.08	0.05	8	83
10	0.08	0.07	6	87

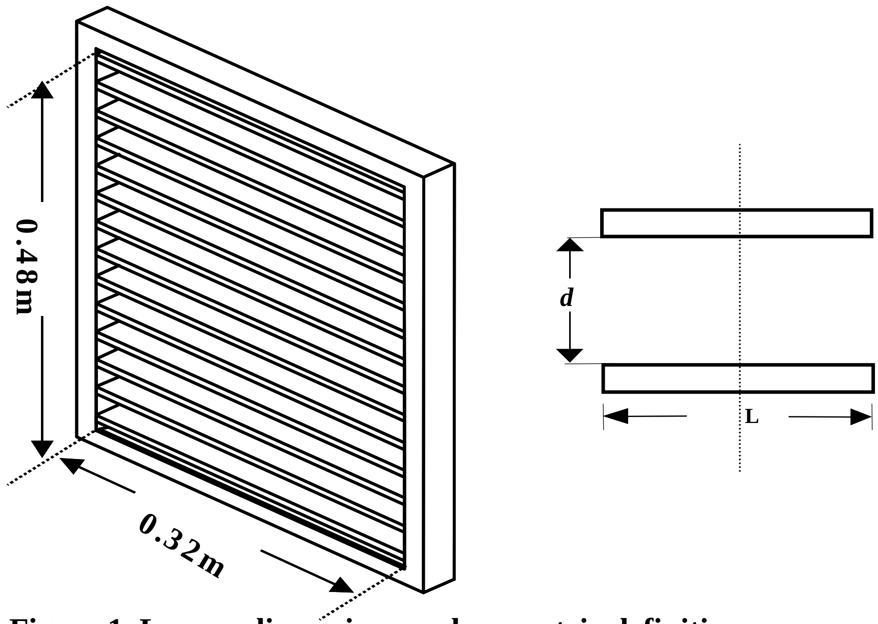


Figure 1 Louver dimensions and geometric definitions

An attempt was made to trace the flow reduction in the occupied zone along a room. For this a room chamber of 1×1×2 *m* long was constructed to represent the environment where internal velocity was recorded. Model louvers were fitted at one end of the chamber and the other end, the air-exiting side, was left open in an attempt to investigate flow patterns that were solely due to the various louver configurations. Pre-set holes on the floor of the chamber were made to represent locations where a high precision hot wire anemometer was positioned to provide a convenient readout of air velocity. Calibrations of the anemometer were carried out to provide a linear output of 0 to 5 volts DC for a velocity range of 0 to 5 m/s. The anemometer's voltage signals were logged on a computer and the software was adjusted to take three readings per second and give an average velocity reading over 30 seconds, 60 seconds and 120 seconds. The first average timing (30 seconds) was chosen to represent the mean values for velocities v_i and v_e as no major differences were found with the longer averaging times. Another hot wire anemometer was used to measure the external air velocities. This anemometer was placed outside the louver opening room chamber to measure wind velocities at pre-set distances of up to 0.75m from the louver. It was evident that the resistance to airflow caused by both the chamber and the louver resulted in discrepancies to readings taken too near to the panel. Beyond a distance of 0.50m from the louver these discrepancies were diminished and so all external velocities were measured at this distance.

For internal velocities detailed analysis of velocity drops along horizontal sections of flow was attempted. Six different locations were selected along the length of the chamber as shown in Figure 2. The pre-set distance between them was 0.25m. The inaccuracies in readings at R_d<0.25m due to the presence of jet flows led to R_d = 0.25m acting as the first measuring location inside the room.

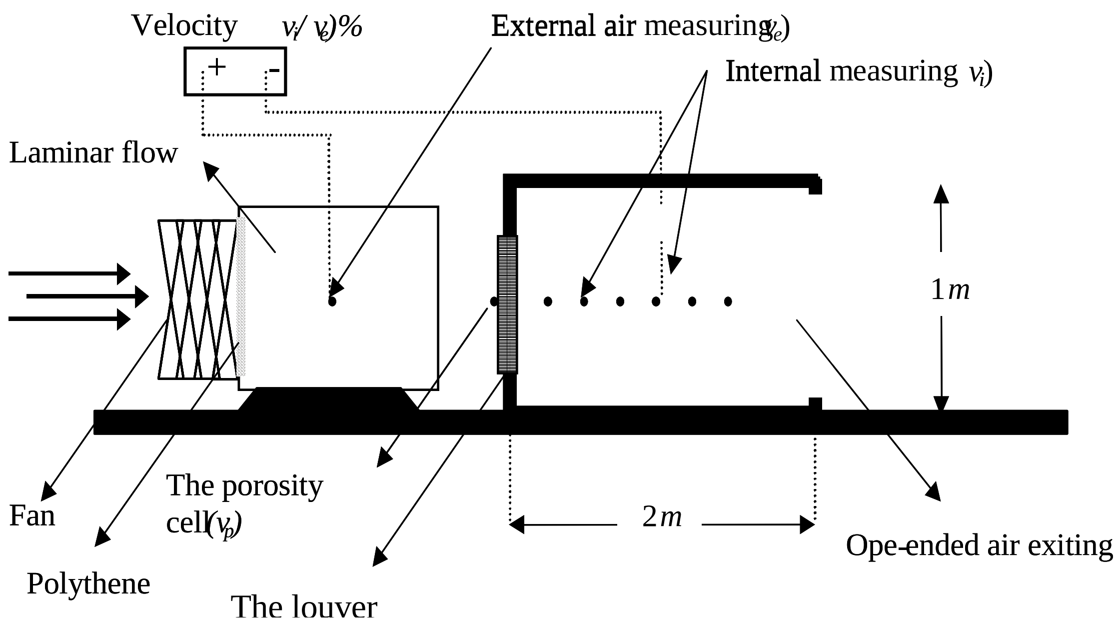


Figure 2 Schematic diagram of the experimental set-up.

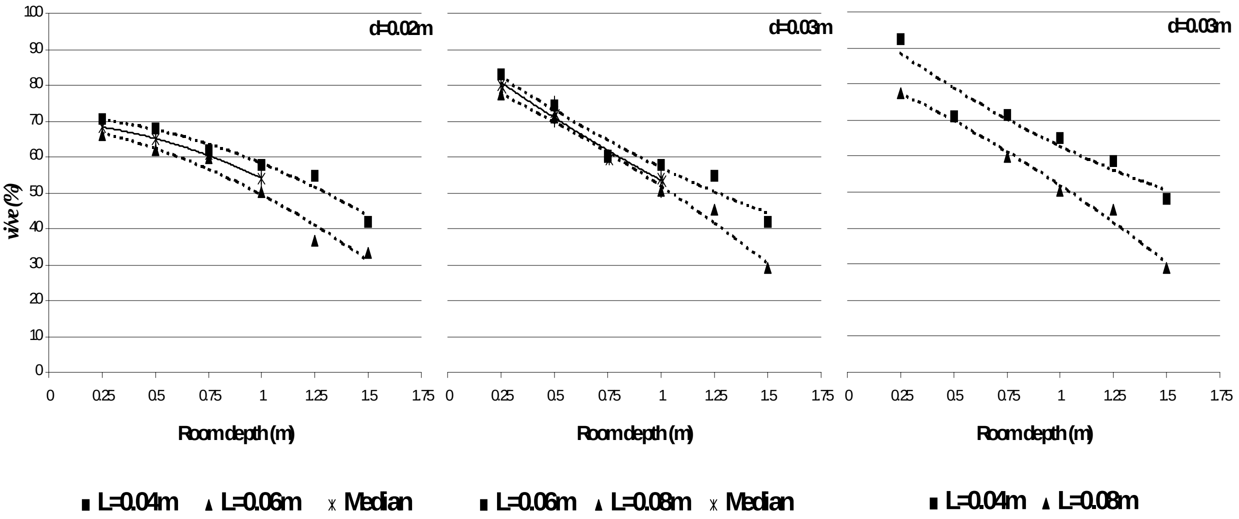
Two 0.48m diameter fans placed in series and facing the front panel of the model room were used to generate the external wind velocity v_e. Initial velocity measurements showed that the fans tended to generate higher velocity profiles at the edges of the fans compared to the central axis. To produce a more uniform incident velocity field a polythene sheet (0.01m thick) was placed at the fan's air exit. Additionally, proper alterations were made to assure laminar flow towards the examined units. The measurements were then carried out after ensuring a steady flow of air using procedures detailed in Etheridge and Sandberg [11]).

RESULT

This study investigated the velocity drop across the louver model configurations given in Table 1. The louver depth L, external wind speed v_e and aperture d of the louvers were investigated, together with the effect of room depth R_d.

Louver depth *L*

The measured variations of the velocity ratio v_i/v_e along the room length as a function of louver depth L is shown in Figure 3.



(a) ***(b)*** ***(c)***

Figure 3 Variation of v_i/v_e as a function of louver blade depth L

Figures 3(a) and 3(b) show that increasing louver depths by small amounts had little effect on the velocity ratio close to the louver. However, with increasing distance in to the room the ratio values start to diverge, with the longer blade depth exhibiting a larger reduction in velocity. Figure 3(c) shows the impact of doubling the louver blade length from 0.04 to 0.08m. There is now a significant and relatively constant velocity ratio difference of 15% to 20% along the length of the room, with the 0.08m blade showing the larger reduction. The external wind speed v_e used in all of these experiments was measured as 2 m/s.

Wind speed v_e

Internal velocities at various locations along the room depth were recorded against external wind speeds that ranged from 0.6 to 2.5m/s. For the smaller louver apertures examined, i.e. $d \leq 0.03$m, the ratio v_i/v_e was found to vary significantly with the magnitude of the external wind speed and distance R_d in to the model room. For larger apertures variations in external wind speed had much less effect on v_i/v_e close to the louvers (low values of R_d), but there were greater variations at some distance in to the room. Table 2 shows the ratio of v_i/v_e at 2.5m/s to v_i/v_e at 0.6m/s. A ratio value of 1.00 would indicate that the change in external wind speed was having no effect on the value of v_i/v_e. All values shown in Table 2 are for measurements along the centre line of the model room

Table 2 Ratio of v_i/v_e values for high to low external wind speeds

	d = 0.02 m L = 0.06 m	d = 0.03 m L = 0.06 m	d = 0.05 m L = 0.06 m	d = 0.07 m L = 0.06 m
Room depth R_d	Ratio	Ratio	Ratio	Ratio
0.25 m	1.21	1.13	1.08	1.01
0.50 m	1.34	1.24	1.08	1.03
0.75 m	1.44	1.40	1.17	1.06
1.00 m	1.47	1.37	1.32	1.14

A possible explanation of the results in Table 2 could be the resistance to airflow caused by the narrow louver spacings. For low *d* values the low wind speed will lose most of its kinectic energy overcoming the flow resistance of the louver gap, and so the flow on the other side of the gap inside the room will be weak. As *d* increases the low and high wind speeds easily push the air through the wider gaps and so the internal velocity close to the louver is higher and steadier.

Louver aperture *d*

The effect of louver aperture *d* on the velocity drop across the louvers v_i/v_e was investigated for several room depth locations R_d along the centre line of the room for a fixed value of louver depth *L*. For high wind speeds with the largest aperture (*d* = 0.07m) the air speed in the room at a depth of 1.0m was still nearly 80% of the external air speed. For the same location an aperture of *d* = 0.02m could only generate 55% of the external air speed. Figure 4 shows the experimental points and the least squares fit curves produced for the louver apertures as a function of *d* and R_d.

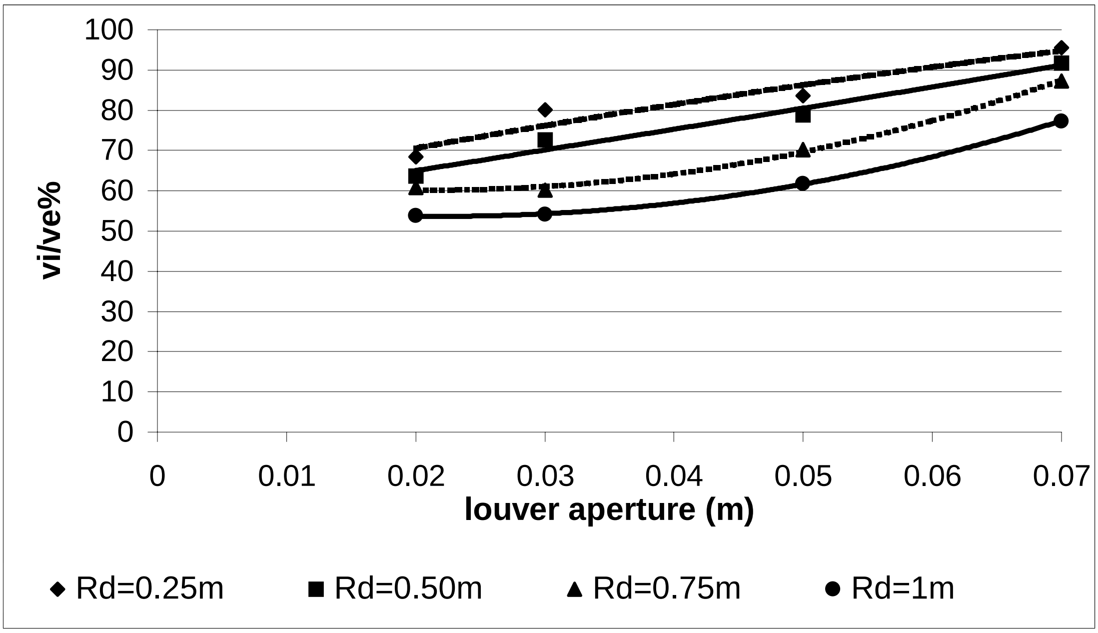


Fig. 4: v_i/v_e produced for louver apertures measured at various R_d.

CFD APPRAISAL.

Computational Fluid Dynamics (CFD) can be used to predict the indoor velocities and overall ventilation effectiveness of natural or mechanical air distribution systems. In this study FLUENT [12] software was used for the CFD simulation of the louver-room configuration. A number of case studies were tested first to simulate the laboratory work. The specification of boundary conditions was carefully established as the assumed boundary conditions play a vital role in the results predicted from the simulations. A refined grid domain of 250 x 100 nodes was established in the room and the vicinity of the louvers. The k-ε model was implemented in the simulation to represent the flow turbulence. The k-ε model is a widely used method to represent turbulence in CFD (Jones and Whittle, [13]). The results of velocity readings were taken when the simulation process was fully converged or when the global absolute residuals from the iterations reached a very low limit (approximately less than $1x10^{-4}$).

The CFD analysis was used to predict the ratio of v_i/v_e for comparison with the experimental data. Two sets of comparisons were made. In the first the measured and predicted values of v_i/v_e were compared at a point close to the louver (R_d = 0.25m) for louver (L, d) values of (4, 2), (6, 3.5) and (8, 5). The results from this are presented in Figure 5. There is good agreement for the reasonably strong airflows encountered very close to the room side of the louvers, which gives encouragement that CFD can be used to predict the local air velocities around louvers. In the second comparison the room depth was varied from 0.25 to 1.5m and the v_i/v_e ratios from CFD and experiment were examined. The results for two louver settings (L=0.08, d=0.03 and L=0.08, d=0.07) are shown in Figure 6. These results are representative of the trends observed in other louver combinations The agreement is still good close to the louver, and the general form of the variation with R_d is similar. However, there is a marked divergence between CFD and experimental results as R_d increases, and towards the back of the room model the differences become very large (around 50%). The design of the room model left the back of the model open with no obstructing wall. It may be that this boundary condition needs to be better defined in future studies as the airflows in this location will be weaker and so more susceptible to large variations over small spatial distances.

CONCLUSIONS

This parametric and CFD study of airflow through louvers has highlighted the following:

- airflow velocities in a room containing a louver covered opening result from an interaction of louver geometry, room geometry and prevailing wind conditions
- the key parameters for louver performance are the aperture opening (distance between louver blades) and the distance from the louvers in to the room
- small changes in the depth of the louver blade do not have a significant effect on airflow velocities
- CFD analysis has been successfully used to predict air velocities in the region close to the room side of the louvers

- CFD analysis was less successful in predicting air velocities towards the rear of the room at some distance from the louver opening

- Further CFD simulations are required to identify appropriate boundary conditions to improve the agreement with the experimental data for all locations within the room.

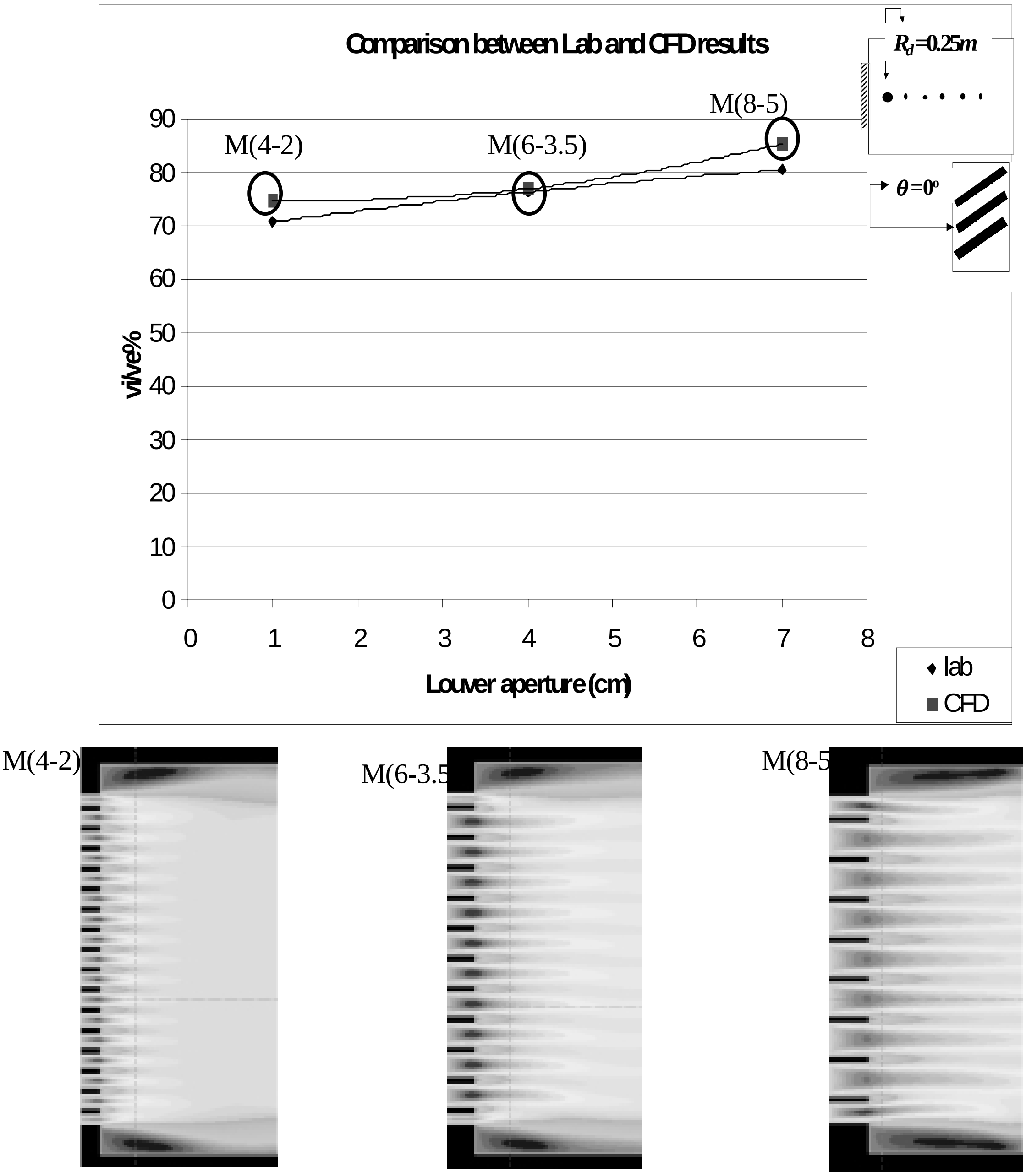
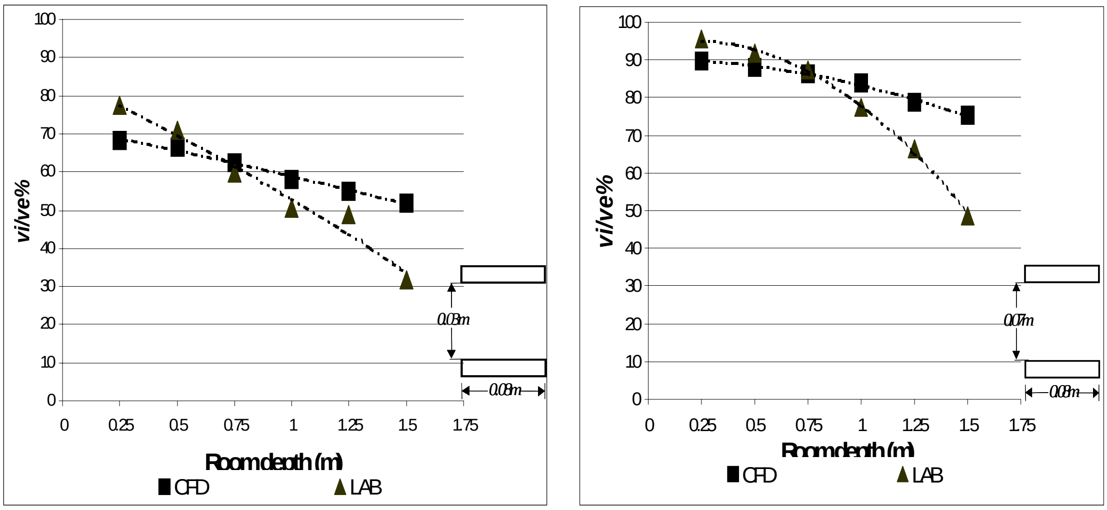


Figure 5 Comparison of CFD and experimental v_i/v_e values at R_d =0.25m



(a) **(b)**

Figure 6 Comparison of CFD and experimental v_i/v_e values for two louvers configurations (a) 0.08-0.03 (b) 0.08-0.07 as a function of room depth R_d

REFERENCES

1. V. Kukadia and M D A E S Perera
NatVent™: aims and vision.
Proc. 19th AIVC Conference, 388-397, Oslo, 28th -30th September, 1998

2. M White, G McCann, R Stephen and M Chandler
Ventilators: ventilation and acoustic effectiveness.
BRE Information Paper IP4/99, 1999

3. K Brandle and P E Boehm
Airflow windows: performance and application. .
Proc. ASHRAE/DOE Conference. "*Thermal performance of the exterior envelope of the building* II", 361-379, 6th-9th December 1982

4. T Jong and G P A Bot
Flow characteristics of one-side mounted windows.
Energy and Buildings **19**, 105-112, 1992.

5. H Sobin
Window design for passive ventilative cooling: an experimental model-scale study.
Proc. International Conference on Passive and Hybrid Cooling, 191-195, Miami, 1999

6. G S Yakubu and S Sharples
Airflow through modulated louver systems.
Building Services Engineering Research and Technology **12** (4), 151-155, 1991.

7. A C Pitts and S Georgiadis
Ventilation air flow through window openings in combination with shading devices.
Proc. 15th AIVC Conference, 432-439, Buxton, UK, 27th-30th September, 1994.

8. A A Maghrabi and S Sharples
Air flow characteristics through Modulated Louvered Windows.
Proc. PLEA Conference, 507-514, Brisbane, 22nd-24th September, 1999.

9. P H Baker, S Sharples and I C Ward
Air flow through cracks.
Building and Environment **22** (4), 291-304, 1987

10. F Oliveira and L S Bittencourt
Air flow through louvered windows in small rooms.
Proc. PLEA Conference, 393-396, Lisbon, 1st-3rd June, 1998.

11. D Etheridge and M Sandberg
Building Ventilation: Theory and Measurements.
John Wiley & Sons, 1996.

12. FLUENT (version 4.51). 1999. New Hampshire, USA, Fluent Inc.

13. P J Jones and G E Whittle
Computational Fluid Dynamics for building air flow prediction-current status and capabilities.
Building and Environment **27** (3), 321-338. 1992.