DIRECTIONAL AIRFLOW PERFORMANCE OF VENTILATORS FOR NATURAL VENTILATION

S. Sharples and N. Chilengwe

Centre for the Built Environment, School of Environment and Development
Sheffield Hallam University, Sheffield S1 1 WB, UK
s.sharples@shu.ac.uk   n.chilengwe@shu.ac.uk

ABSTRACT

The use of natural ventilation systems continues to be a popular feature in low energy, sustainable building design. One feature of natural ventilation is that, depending upon the prevailing climatic or thermal conditions, the airflow through a ventilator can be bi-directional. Aerodynamically, the ventilator, depending upon its construction, may not perform in the same way for the two different flow directions. The geometry of the internal flow-path, the inclination of louver blades and the location of meshes and acoustic linings are features that stop the ventilator being aerodynamically symmetrical for different flow directions. However, in most cases ventilator airflow performance data relate to tests conducted with the ventilator set with an 'inlet' configuration. This is partly because ventilator manufacturers are concerned about rain being driven through the ventilator, and so test for this 'worst case' scenario. This paper describes a series of experimental measurements on commercial louver ventilators that investigated the influence that flow direction and magnitude of pressure differentials can have on airflow performance. Louver face velocities of 0.05, 0.25 and 0.5 ms^{-1} were generated in the study. The commercial ventilators were also modelled as CAD files and then imported in to the CFD software FLOVENT v3.2. The experimental data were compared with CFD predicted values of airflow parameters. This analysis tested the feasibility of using CFD / CAD as a tool for designing and optimising the performance of ventilators.

KEYWORDS

Natural ventilation, ventilators, mesh screens, CFD, airflow direction

INTRODUCTION

The need to make buildings more energy efficient has resulted in a requirement to reduce the energy consumption associated with mechanical ventilation and air conditioning. This involves either supplementing or replacing these energy-intensive services with passive and more sustainable schemes such as natural ventilation [Allard 1998]. At the heart of natural ventilation systems are the facade openings that allow fresh air to flow in and stale air to flow out of a building. These openings can take the form of simple holes, openable windows, trickle ventilators and through-wall ventilators. From an aerodynamic perspective a ventilator can be a combination of relatively large openings (such as louvers) with flow properties independent of the flow rate (Reynolds number independent), and very small openings (such as insect screens) that may have some Reynolds number dependence [as indicated in Baker *et al*. 1986]. Thus, the design, selection and combination of components to form a ventilator directly affect the airflow performance of the ventilation openings and hence play an important role in determining the success of natural ventilation systems. A recent study by the UK's Building Research Establishment [White *et al.* 1999] of twenty-one commercial trickle and wall ventilators revealed that the airflow performance of these devices could be more complex than might be envisaged from their relatively simple constructions. Some ventilators performed better than would be predicted from their free area, and others performed worse (free area is the physical size of the smallest aperture in the ventilator).

There have since been some studies of the airflow characteristics of ventilator units for natural ventilation applications [Chilengwe & Sharples (2002); Karava *et al* (2003)] in order to gain fundamental understanding of the airflow characteristics. The main objective of this study was to investigate, via a series of experimental measurements on commercial louvers and mesh-screens, the influence that flow direction and magnitude of pressure differentials can have on airflow performance of a ventilator. Experimental data was also compared with Computational Fluid Dynamics (CFD) predicted values of airflow parameters.

METHODOLOGY

The study of airflow performance through ventilation openings involves establishing a pressure difference ΔP across an opening, measuring the consequent airflow Q through that opening and then deducing a relationship between ΔP and Q. For most practical purposes, and as preferred by some ventilation researchers [Liddament 1987, Walker *et al.* 1998], the power law can be used to estimate the driving force ΔP resulting into an airflow Q. The power law takes the form:

$$Q = c\,(\Delta P)^n \qquad (1)$$

where c and n are constants (n = 0.5: turbulent flow, n = 1: laminar flow) which are assumed to depend only upon the geometry of the ventilation opening. In airflow systems mesh-screens serve a variety of important functions including production of uniformity of the velocity distribution, reduction of turbulence of the air-stream, production of artificially high turbulence, introduction of known pressure drops into experimental systems [Annand 1953] and control of dust and insect entrance into buildings [Miguel & Silva 2000]. The pressure drop caused by the presence of a mesh-screen in an airflow system is normally expressed in the form of a loss coefficient k obtained experimentally and by using the standard equation:

$$\Delta p = 0.5\,k\,\rho\,v^2 \qquad (2)$$

where ρ is the density of air and v is the velocity. CFD can be utilized to predict airflow patterns and to carry out parametric studies. However, its accuracy and quality of results rely heavily on accurate setting up of the simulation model and specification of boundary conditions [Dascalaki *et al.* 1999] governing the flow field. Some studies [Holmes & Whittle 1987, Huo *et al* 2000] have offered advice and solutions for overcoming some of the difficulties encountered in the application of CFD techniques in building ventilation. There has been very little work on CFD studies of the airflow performance of ventilators and their associated components for natural ventilation applications. In addition there are very few validation results comparing CFD and physical model testing. Therefore, parametric investigations, such as those forming the core of this study, would serve to provide a validation basis for CFD to be used with confidence in subsequent modelling and designing of ventilators.

TEST RIG, EXPERIMENTAL MEASUREMENTS AND CFD MODELLING

The test rig (Figure 1) used in this study was designed and constructed in accordance with the draft European standard (pr En 13141-1) for ventilators. Background chamber leakage was determined for pressurization and depressurization with the plenum chamber opening sealed.

Three commercial aluminium louvers, such as the one shown in Figure 2, each measuring 495 x 495 x 130 mm deep were tested. Louver X has 5 blades each inclined at 45 deg with a 100 mm pitch between the blades. Louver Y and louver Z are very similar and each has 6 blades, each inclined at 45 deg with a 75 mm pitch. Louver Z incorporates an additional offset or "rain hook" on each blade which is intended to provide additional protection against water ingress. Two mesh types each measuring 450 mm x 450 mm were also used - a thin polyethylene type insect screen with hexagon-shaped orifices with a free area of 55% and a polygon-shaped orifice welded strip bird guard with a free area of 65%. The louver design is such that meshes can be attached onto the back of the louver (35mm from the blades) and held in position by two screws. The air volume flow rate was measured using a factory-calibrated laminar flow meter and the corresponding pressure difference across the ventilator was measured using a high quality digital micromanometer - accuracy quoted by manufacturer as ± 0.25% for ΔP and ± $0.0003 m^3/s$ for air volume flow rates. The data collected (Q-ΔP readings) were then subjected to power-law regression analysis to obtain the airflow characteristic of each ventilator for both inflow and out flow configurations.

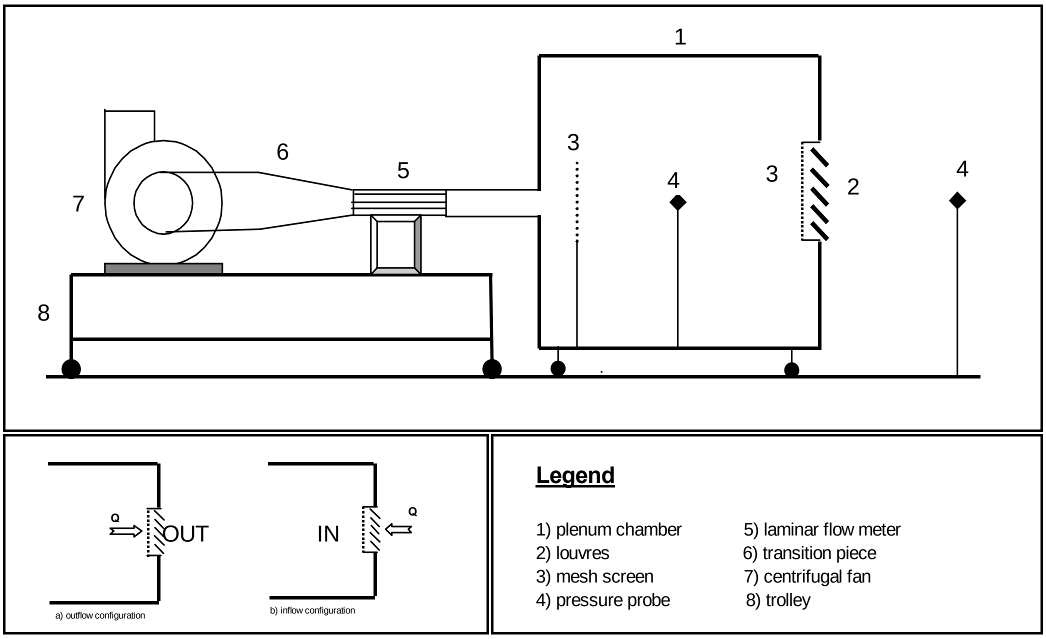

Fig 1. Schematic of experimental test rig

Fig 2. Example of louvre "Y"

CAD files of the louvers were imported into the CFD software FLOVENT using the FLO/MCAD module. The louver and mesh-screen components were incorporated into a plenum chamber model representing the actual laboratory test rig on which experimental measurements were carried out for comparison with simulated results. The louvers used in combination with the insect screen and bird guard (mesh-screens A and B) were subjected to pressure differentials resulting in velocities of 0.05, 0.25 and 0.5m/s on the louver face. The simulation results (Tables 2 & 3) were then compared with experimental measurements for both inflow and outflow configurations.

RESULTS AND DISCUSSION

The results (Table 1) show that for a given pressure differential (ΔP) across the ventilator component the resulting airflow for inlet configurations was different from that obtained for outlet configurations. The majority of the cases shown in Table 1 indicate that the ventilator performed better in outlet configuration than it did in the corresponding inlet configuration. Table 1 further shows that the percentage difference deviation between the outlet flow rate and inlet flow rate relative to the inlet flow rate was greatest at low pressure differences. The deviations for the three louvers averaged 21.2% at 0.05 Pa, 11% at 0.25 Pa, 7.4% at 0.5 Pa, 2.3% at 1.5 Pa and 1.1% at 2 Pa. Figures 3 and 4 show the airflow performance of outlet configurations relative to inlet configurations plotted as functions of pressure differentials across louver/mesh combinations. Here again it is evident that the effect of flow direction was more significant at low pressure differences and gradually levels off as ΔP increases, however, the graph obtained for louver X is different from those for louver Y and louver Z. An explanation of these differences is that airflow performance is significantly affected by variations in ventilator geometry i.e. louver Y and louver Z which gave similar graphs are also similar geometrically). Further investigations will be conducted to determine any other reasons for these differences. For the moment it is interesting to note that in all three cases the ventilator with a bird guard mesh intersects the one with an insect screen somewhere between 1 Pa and 1.5 Pa. This suggests that there is a ΔP between these limits at which the airflow performances of the two ventilators are equal. Bearing in mind that the two meshes used in this study have different geometries it is not surprising that the airflow performances of the two ventilators intersect. Table 2 showing a comparison between experimental results and CFD calculated ΔP values for inlet configurations. The Table indicates that there is reasonable agreement between the two sets of data. Although the deviations extend over a wide range nearly two-thirds of the cases considered were under ±20%. It is clear from Table 2 that only the CFD model for louver Y used in combination with meshes consistently under predicted the ΔP's. The results also show that the models used tended to over predict the ΔP at the higher louver face velocity i.e. 0.5 m/s and under-predicted ΔP at 0.05 m/s. Generally better agreement was achieved when no mesh was included in the model. It can be seen (Figure 5) that agreement between measured and CFD calculated pressure differentials improved with increasing louver face velocities. The reason for this trend is that at higher louver face velocities the pressure differentials were larger and more firmly established, hence making it easier to measure experimentally i.e. less variation than at low face velocities.

TABLE 1

Measured air flow rates for inlet and outlet configurations at various pressure drops

Louvre	Mesh type	Pressure drop across louvre ΔP = 0.05 (Pa)			Pressure drop across louvre ΔP = 0.25 (Pa)			Pressure drop across louvre ΔP = 0.5 (Pa)			Pressure drop across louvre ΔP = 1.5 (Pa)			Pressure drop across louvre ΔP = 2 (Pa)		
		Inlet Qi (l/s)	Outlet Qo (l/s)	$\frac{Qo-Qi}{Qi}$ (%)	Inlet Qi (l/s)	Outlet Qo (l/s)	$\frac{Qo-Qi}{Qi}$ (%)	Inlet Qi (l/s)	Outlet Qo (l/s)	$\frac{Qo-Qi}{Qi}$ (%)	Inlet Qi (l/s)	Outlet Qo (l/s)	$\frac{Qo-Qi}{Qi}$ (%)	Inlet Qi (l/s)	Outlet Qo (l/s)	$\frac{Qo-Qi}{Qi}$ (%)
X	no mesh	18.02	17.70	-1.8	41.83	42.68	2.0	60.12	62.36	3.7	106.8	113.7	6.5	124.17	133.09	7.2
	bird guard	13.87	15.26	10.0	33.72	36.46	8.1	49.44	53.06	7.3	90.7	96.1	6.0	106.30	112.33	5.7
	insect screen	10.78	9.10	-15.6	29.81	28.25	-5.2	46.20	46.03	-0.4	92.5	99.8	7.9	110.97	122.19	10.1
Y	no mesh	18.28	22.10	20.9	44.44	47.69	7.3	65.14	66.42	2.0	119.5	112.3	-6.0	140.00	128.83	-8.0
	bird guard	13.19	17.05	29.3	34.22	39.92	16.7	51.60	57.60	11.6	98.9	103.0	4.1	117.30	119.89	2.2
	insect screen	13.73	15.80	15.1	34.23	37.64	10.0	50.74	54.70	7.8	94.7	98.9	4.5	111.45	115.54	3.7
Z	no mesh	18.85	23.60	25.2	44.05	51.94	17.9	63.49	72.97	14.9	113.3	125.0	10.4	131.87	143.99	9.2
	bird guard	11.17	19.10	71.0	33.74	43.24	28.2	54.30	61.47	13.2	115.5	107.4	-7.0	140.67	124.26	-11.7
	insect screen	10.81	14.73	36.3	32.93	37.65	14.3	53.22	56.40	6.0	113.9	107.0	-6.0	138.97	126.60	-8.9

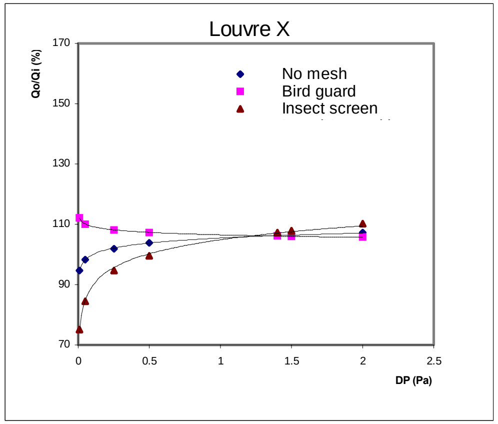

Fig.3: Qo/Qi as a function of Δp for louvre X

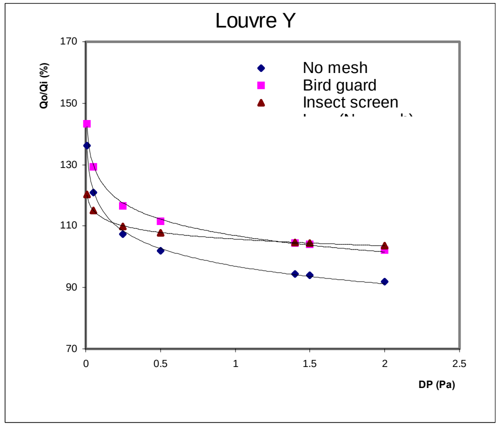

Fig.4: Qo/Qi as a function of Δp for louvre Y

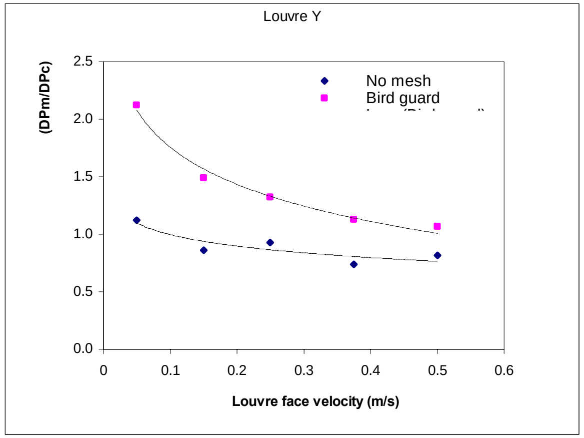

Fig.5: Ratio $\Delta P_M/\Delta P_C$ as function of face velocity

Table 2

Experimental results and CFD calculated pressure differentials for inlet configurations

Louvre	Mesh type	Louvre face velocity 0.05m/s			Louvre face velocity 0.25m/s			Louvre face velocity 0.5m/s		
		Measured ΔP (Pa)	CFD calculated ΔP (Pa)	$\frac{M-C}{M}$ (%)	Measured ΔP (Pa)	CFD calculated ΔP (Pa)	$\frac{M-C}{M}$ (%)	Measured ΔP (Pa)	CFD calculated ΔP (Pa)	$\frac{M-C}{M}$ (%)
X	no mesh	0.018	0.019	-5.6	0.382	0.361	5.5	1.448	1.453	-0.3
	bird guard	0.030	0.026	13.3	0.552	0.454	17.8	1.951	2.450	-25.6
	insect screen	0.048	0.030	37.5	0.607	0.718	-18.3	1.828	2.828	-54.7
Y	no mesh	0.018	0.016	11.1	0.336	0.362	-7.7	1.187	1.451	-22.2
	bird guard	0.034	0.018	47.1	0.512	0.388	24.2	1.659	1.555	6.3
	insect screen	0.031	0.019	38.7	0.527	0.415	21.3	1.801	1.750	2.8
Z	no mesh	0.017	0.018	-9.1	0.346	0.379	-9.5	1.298	1.501	-15.6
	bird guard	0.046	0.018	60.0	0.473	0.386	18.4	1.306	1.527	-16.9
	insect screen	0.048	0.021	56.3	0.487	0.440	9.7	1.332	1.745	-31.0

CONCLUSIONS

The airflow through ventilators is influenced by the flow direction. The influence is more pronounced at low pressure differentials. Where prevailing climatic conditions are reasonably consistent it might prove beneficial in naturally ventilated buildings to have differently designed inlet and outlet ventilators. Further work should seek to design ventilators that are aerodynamically symmetrical whilst maintaining a good airflow performance at low pressure differentials. The study has also highlighted an example of integrating CAD and CFD models as a tool for analyzing ventilators for natural ventilation applications. Since data for CFD validation of airflow through louvers/meshes are extremely scarce it follows that studies such as this will generate data which can hopefully lead to ventilator manufacturers having better design guidelines and design tools for the production of efficient and effective natural ventilation ventilators. In turn such information could improve the confidence in natural ventilation design methods to a level where they can match similar components in mechanical ventilation applications.

The authors wish to thank the UK's Engineering and Physical Sciences Research Council for funding this work (Grant No. GR/R70637).

REFERENCES

Annand W J D. (1953). The resistance to air flow of wire gauzes. *Journal of the Royal Aeronautical Society, **57**, 141 -146*.

Allard F (1998). *Natural Ventilation in Buildings: a Design Handbook*, James and James, London, UK.

Baker P H, Heap R D and Sharples S. (1986). Airflow through perforated screens at small pressure differences. *Building Services Engineering Research & Technology, **7:2**, 96-97.*

Chilengwe N and Sharples S. (2002). Parametric analysis of air flow through ventilators. *The International Journal of Ventilation, **1:2**, 109-117.*

Dascalaki E, Santamouris M. and Asimakopoulos D N. (1999). On the use of deterministic and intelligent techniques to predict the air velocity distribution on external openings in single-sided natural ventilation configurations. *Solar Energy, **66**, 223-243.*

Holmes M J and Whittle G E. (1987). How accurate are predictions of complex air movement models? *Building Services Engineering Research & Technology, **8**, 29-31.*

Huo Y, Hahighat F, Zhang J S and Shaw C Y. (2000). A systematic approach to describe the air terminal device in CFD simulation for room air distribution analysis. *Building & Environment, **35**, 563-576.*

Karava P, Stathopoulos T and Athienitis A K (2003). Investigation of the performance of trickle ventilators. **38**, 981-993.

Liddament M W. (1987). Power law rules - OK? *Air Infiltration Review, **8:2**, 4-6.*

Miguel A F and Silva A M. (2000). Effect of aperture characteristics on the mean and turbulent heat and mass flows within enclosures. *Proceedings of RoomVent Conf. Reading, UK*.

Walker I S, Wilson D J and Sherman M H. (1998). A comparison of the power law to quadratic formulations for air infiltration calculations. *Energy and Buildings, **27**, 293-299*.

White M, Mc Cann, Stephen R and Chandler M. (1999). Ventilators: Ventilation and acoustic effectiveness. *Building Research Establishment, UK, Information Paper, IP 4/99.*