

Analysis of ventilation efficiency indices inside a local domain within an urban area using two building model

Mahmoud Bady^a, Shinsuke Kato^b, Hong Huang^c

^a*Graduate School of Engineering, The University of Tokyo, Komaba, Meguro-ku, Tokyo, Japan*

^b*Institute of Industrial Science, The University of Tokyo, Komaba, Meguro-ku, Tokyo, Japan*

^c*Institute of Industrial Science, The University of Tokyo, Komaba, Meguro-ku, Tokyo, Japan*

ABSTRACT: This paper deals with the determination of the ventilation efficiency (VE) indices, Purging Flow Rate, Visitation Frequency, and Staying Time of air pollutants, inside a domain located within an urban area. For the purpose of study, a wind environment of 100 x 100 x 75 m³ volume was considered and a two building model inside was adopted to perform the calculations. A void was constructed by a central street with one building on each side. Two parameters were selected to evaluate the VE indices; the first one was the street width, while the second parameter was the height of the upwind building. In all cases of the present study, the applied wind direction was assumed normal to the street direction. The evaluation of the VE indices were carried out based on the average pollutant distribution inside the domain. Results of the study showed that, both the street width and the upwind building height have great effects on the VE indices, as they can be considered the tools which control the air quality inside urban domains.

KEYWORDS: Ventilation efficiency indices, urban area, air quality

1 INTRODUCTION

During last few decades, investigation of air pollutants transportation and dispersion in urban areas has become of major concern for the protection of air quality and also for compliance with air pollution regulations. The importance of such investigation arises from the increased levels of air pollutants in the atmospheric air of urban areas due to the continuous expansion of industries and the increased usage of motor vehicles especially in large urban areas. Both numerical and physical models try to simulate the flow and evaluate the concentrations of air pollutants within built-up areas. It was considered that the estimation of pollutant concentrations only is not sufficient to reflect a complete image about the air quality of a domain [1]. Still there is a need to consider another measure (or measures) that can describe the pollutant behavior inside the study domain, such as the pollutant staying time and the return times of the pollutant to the domain, in addition to the pollutant concentration. These requirements can be attained by considering the ventilation efficiency of indoor environments. There are many indices which describe the VE of a domain such as: Purging Flow Rate (*PFR*), Visitation Frequency (*VF*), Staying Time (*T_P*), and the Six Indices (*SVEI-6*) [2]. In this paper, *PFR*, *VF*, and *T_P* will be considered only. Recently, it was thought that the VE indices for indoor environments are also effective in evaluating the removal efficiency of air pollutants by the domain local wind in outdoor areas [1].

There are many factors in urban areas which affect the wind flow characteristics (which in turn affect the dispersion of pollutants) such as the geometry and arrangements of adjacent buildings, street geometry and wind direction. For wind environment improvement, the street geometry is one of the important parameters to be considered by planners and designers. In the present work, effects of the urban street width and the upwind building height on the VE indices are studied.

2 VENTILATION EFFICIENCY INDICES

2.1 Purging flow rate

Purging flow rate of a domain is defined as the effective airflow rate to purge the pollutants outside that domain [2]. In other words, *PFR* can be considered as the net rate by which a pollutant is flushed out of the domain. The following equation is used to estimate the *PFR* [2]:

$$PFR = q / c_p \quad (1)$$

where q = pollutant generating rate (kg/s), c_p = domain average concentration (kg/m³).

2.2 Visitation frequency

The VE index that represents the number of times a pollutant particle enters a domain and passes through it is the visitation frequency, *VF*. It is estimated according to the equation:

$$VF = 1 + \Delta q / q \quad (2)$$

where Δq = inflow flux of pollutants into the domain (kg/s), with pollutant generation q (kg/s).

2.3 Average staying time

One of the most promising indices being used as an indication of the ventilation performance is the staying time of pollutants inside a domain, T_p . It can be defined as the time a pollutant takes from once coming (or being generated) into the domain to its leaving. It is expressed as:

$$T_p = V / (PFR \times VF) \quad (3)$$

where V = volume of the study domain (m³).

3 CFD MODELING

A wind environment contains two buildings of dimensions 5 x 25 x 10 m was simulated and the domain of study was taken as the pedestrian volume ($D \times 25 \times 2$ m) as illustrated in Fig. 1. A numerical solution for the flow field was performed using CFD code STAR-CD. Steady state analysis was adopted and the MARS scheme was applied to the spatial difference. The applied wind direction was assumed normal to the street direction.

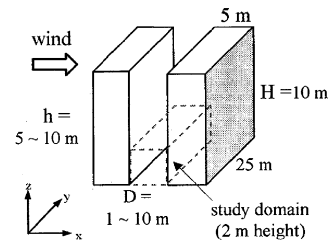


Figure 1. Building model.

4 RESULTS AND ANALYSIS

4.1 Effect of street width

Effects of street width (D) on the VE indices are displayed in Fig.2 through Fig.5. All quantities were normalized by the similar quantities estimated at the case of $D/H = 10$ %. Figure 2 shows that the pollutant average concentration decreases as the street width increases. This behavior indicates that the greater the width, the greater the volume of the domain, which increases the domain average wind speed and hence decreases the concentration. Variation of the *PFR* with D is

displayed in Fig.3 in terms of the air exchange rate ($= PFR/V$). The air exchange rate increases with increasing D since the greater the domain volume, the greater the amount of pollutants present inside it. A large flow rate (PFR) is therefore needed to purge the pollutant towards the exit.

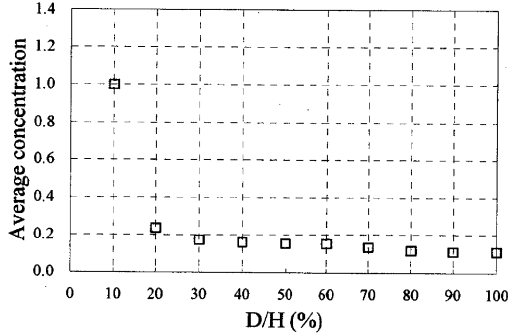


Figure 2. Normalized domain average concentration.

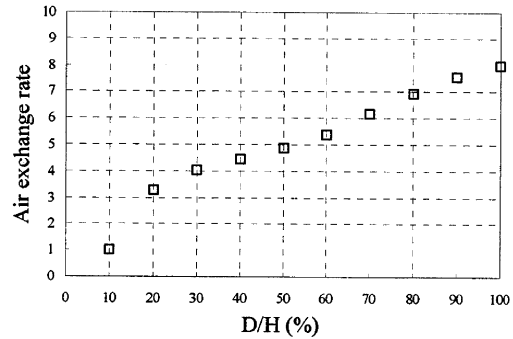


Figure 3. Normalized air exchange rate.

Effect of the street width on the visitation frequency is shown in Fig. 4. The figure indicates that, the variation of the pollutant VF with the width D is not constant. That trend can be interpreted as follows: increasing the width increases the area subjected to the flux, which increases the domain inlet flux. As a first thinking, increasing the inlet flux is expected to increase the VF . That conclusion is not absolutely correct because the VF value depends on the domain inlet flux as well as the domain volume. The ratio between these two variables determines the value of VF .

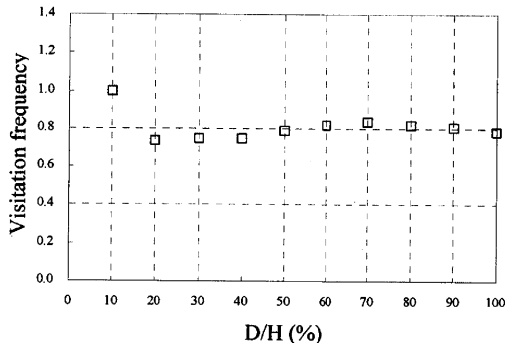


Figure 4. Normalized visitation frequency.

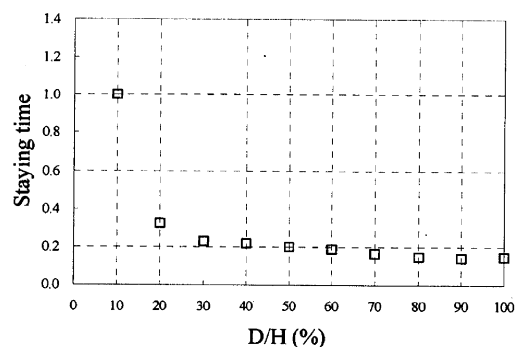


Figure 5. Normalized average staying time.

History of the pollutant staying time inside the study domain is given in Fig. 5. Clearly, increasing D has a desirable effect on T_p as it decreases as the street widens. That behavior can be referred to the increased purging capability of the domain local wind for pushing the pollutants as the domain volume increases, which reduces the time the pollutant takes towards the exit.

4.2 Effect of upwind building height

In this section, variation of the VE indices with the upwind building height (h) is presented. The street width was assumed fixed at 10 m. All quantities were normalized by the similar quantities estimated at $h/D = 50\%$. Fig. 6 shows that the average concentration increases as h increases. Also, the air exchange rate decreases with increasing h as displayed in Fig. 7. These effects are attributed to the lower average wind speed inside the domain and the decrease in the amount of fresh air supplied to it with the increase of h .

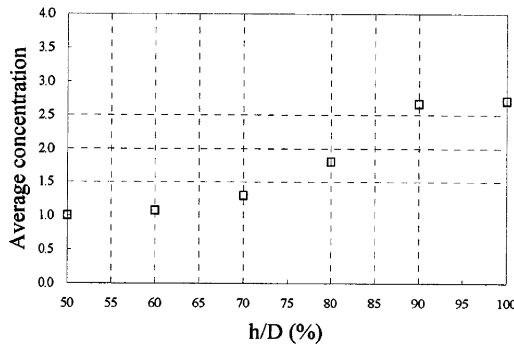


Figure 6. Normalized domain average concentration.

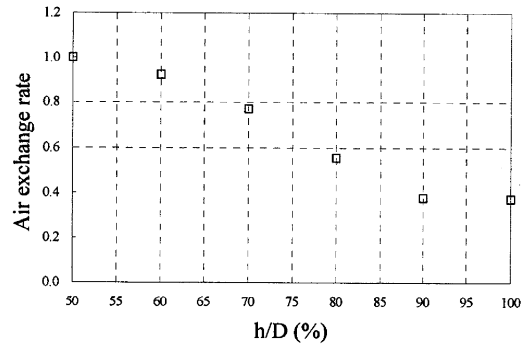


Figure 7. Normalized air exchange rate.

Figure 8 shows the effect of h on the VF of the pollutant. A little increase in the VF occurred due to a little increase in the domain total inlet flux while the study volume was kept constant.

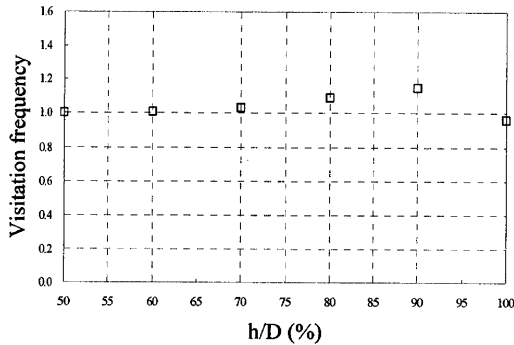


Figure 8. Normalized visitation frequency.

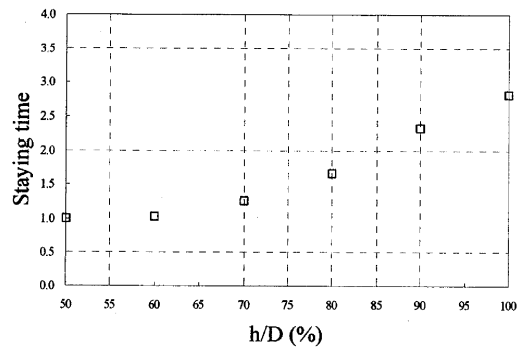


Figure 9. Normalized average staying time.

In Fig. 9, staying time increases as h increases. That behavior is expected to occur due to the low removal efficiency of the domain local wind as a result of the low volume averaged speed.

5 CONCLUSIONS

Ventilation efficiency indices (PRF , VF , and T_p) of indoor environment were introduced to analyze the air quality of a domain located within an urban area. Effects of the study domain width and the upwind building height were evaluated. In conclusion, it can be said that the two parameters have great effects on the air quality reflected by the VE indices, as they must be considered by planners when new built-up areas are designed regarding the air quality improvement.

6 REFERENCES

- 1 H. Huang, S. Kato, R. Ooka and T. Jiang, CFD analysis of ventilation efficiency around an elevated highway using visitation frequency and purging flow rate, Proc. 6th Int. Conf. on Wind Eng., Korea, 2005, pp. 752-761.
- 2 S. Kato, K. Ito, S. Murakami, Analysis of visitation frequency through particle tracking method based on LES and model experiment, Indoor Air, 13 (2003) 182-193.

Full-scale measurements and numerical simulation of cross-flow ventilation of farm buildings in a cold, windy coast climate

Thomas K. Thiis^a, Willy K. Jeksrud^{a†}, Andreas S. Flø^a

^aNorwegian University of Life Sciences, Ås, Norway

ABSTRACT: New demands in farming industry in Norwegian sub-arctic regions has caused a need for larger animal buildings. To minimize the cost of the buildings, they are built without insulation. Uninsulated animal buildings in cold climate raise an array of new engineering challenges. One challenge is to maintain the animal welfare. Because of the lack of insulation, the walls of the buildings are exposed to a high risk of condensation. This is met with a high ventilation rate to remove the excess moisture. The present study is performed on a farm at the coast in northern Norway. Numerical simulations of cross flow ventilation of an animal house are performed for several wind directions and for different openings. Both the internal and external air movements are simulated in the same simulation domain. The ventilation openings in the walls are varied to assure that the animals are not exposed to draught in periods of cold weather and high wind speed. The simulation results are used to develop an algorithm for automatic control of ventilation openings based on meteorological parameters. The air velocity inside the animal building was measured in the event of artificially produced wind pressure on the surface with the help of a fan.

KEYWORDS: Cross flow ventilation, numerical simulations

1 INTRODUCTION

Animal welfare of cattle has been studied by several. A useful parameter for assessing animal exposure to climate is the Lower Critical Temperature (LCT). "LCT is the temperature below which the animal will increase its rate of heat production above normal in order to maintain a constant body temperature" (Gregory, 1995). LCT varies according to wind speed, the condition of the coat of the animal and the feeding level. For cattle LCT is reported in table 1 (Holmes and Sykes, 1984)

Table 1. Lower critical temperatures (°C) for cattle (Holmes and Sykes, 1984)

	Maintenance feed				2xMaintenance feed			
	Dry coat		Wet coat		Dry coat		Wet coat	
Wind speed [m/s]	0.3	3.9	0.3	1.1	0.3	3.9	0.3	1.1
Calves 40 kg liveweight								
Coat depth 2 cm	9	17	16	24	-9	4	3	16
Adult 400 kg liveweight								
Coat depth 2 cm	-2	7	6	16	-19	-13	-14	1
Coat depth 3 cm	-7	3	2	11	-36	-20	-22	-5

Table 1 presents the air velocity in two steps 0.3 m/s and 3.9 m/s for dry cattle. This is usually the condition of the cattle in the houses in the study. The upper threshold value is probably also a limit of what is practical inside a building. If the air velocity should exceed 3.9 m/s, a number of

other problems would arise, such as flying debris or dust. In this study, the preferred air velocity inside the animal house is therefore set to maximum 3.9 m/s in 0.7 m height.

During the last years there has been built several un-insulated animal buildings in Norway. The climate in Norway varies from cold to polar climate according to the Köppen climate classification index. The combination of cold climate and the large moisture emission from the cattle increase the risk of condensation of water on the cold interior surfaces of the building. To avoid the condensation, the ventilation rate of the building must be high. To avoid the installation of a ventilation system, parts of the walls of the buildings are made semipermeable, thus the ventilation of the buildings is driven by the wind. The internal climate and the comfort of the animals is thus directly linked to the surrounding climate.

The indoor temperature should not fall below the LCT at the given feeding level and indoor air velocity. One can assume that the animal coat is dry when the animals are indoors. For high ventilation rates one can also assume that the outside and inside air temperature is the same. However, measurements of indoor air temperature together with outside air velocity shows that for lower ventilation rates there is a close correlation between the indoor/outdoor temperature difference and the wind velocity.

1.1 *The building in the study*

The building in the study is 29 meters long and 18 meters wide. Inside of the building, there is a section consisting of the milking area, which is ventilated separately and not included in the calculations. There is also a 1.4 meter fence dividing the building in two long rows where the cattle can walk freely around. The longest outer walls of the building consist of a semi permeable, slotted wall from the eave and 1 m down. The rest of the wall is non permeable to air.

2 METHODS

2.1 *Numerical method*

To simulate the wind field around and inside the building, a general-purpose finite volume CFD code was applied. The CFD code solves the incompressible, time averaged Navier-Stokes equations, using a $k-\epsilon$ turbulence model to close the equations. The turbulence model is known to compute excessive turbulence near the edges of the windward walls, which also can affect the downwind wakes. It is however widely used, and it is capable of producing realistic wind patterns around buildings. The number of grid cells in the simulations is around 3.6 million with grid refinement near the surfaces.

The simulation of the wind pattern around and inside the building is performed for 12 wind directions; 0, 30, 60, 90, 120, 150, 180, 210, 240, 270, 300 and 330 degrees. These simulations are performed with an inlet wind velocity of 15 m/s in 10 meter height outside the building.

To verify that the flow pattern inside the building remains independent of the wind velocity, there is also performed a range of simulations with inlet wind velocity varying according to table 1. In these simulations, the permeability of the wall and the wind direction is kept constant.

The inlet wind profile follows the logarithmic expression

$$u(z) = \frac{u_*}{\kappa} \ln \left(\frac{z}{z_0} \right) \quad (eq.1)$$

where $u(z)$ is the wind speed in height z , z_0 is the roughness of the surface, set to 0.1m, u_* is the friction velocity set to according to table 1 and κ is the von Karman constant, equal to 0.4. The turbulence intensity at the inlet boundary is set to 5%. The outlet boundary is continuative, meaning that the normal derivatives of all quantities are set to zero. On the wall surface boundaries the boundary conditions are set to “no-slip” i.e. the velocity at the surface is zero. The lateral and top boundaries of the simulation domain have symmetry conditions.

Table 1. Friction velocities used in equation 1.

$u(10)$	5	10	15	25
u_*	0.43	0.87	1.30	2.17

2.2 Permeable wall

To avoid excessive use of grid cells, the slotted walls were modelled as a permeable wall with a pressure drop. The pressure drop over slotted walls of different permeability was found simulating the air flow trough the wall, using the CFD solver. The fraction of the wall open to flow and the air velocity was varied to produce the velocity dependent pressure loss curves. Miguel (1998) used a similar procedure on physical measurements of the pressure loss over insect screens.

2.3 Full-scale measurements

Full-scale measurements of the air velocity inside the building were performed with the help of a 1.5 meter radial fan which produced the wind pressure on the wall. The pressure was applied on a 3.4 meter section of the building.

3 RESULTS AND DISCUSSION

3.1 Air flow around and inside of the building

It is assumed that the air inside the building is well mixed and that the inside temperature equals the outside temperature. The air velocity inside the building has a spatial variability. It is therefore defined a residential zone where the climatic conditions should be better than outlined in

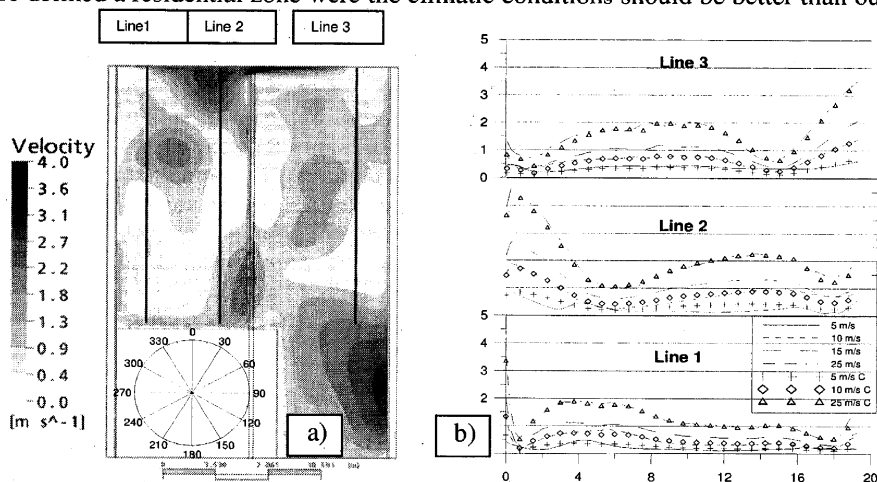


Figure 1 a) position of the residential zone in the building b) Air velocity in the residential zone

table 1. The residential zone is where the cattle rest, and is indicated in figure 1a with tree lines positioned 0.7 meters above the floor. Figure 1b shows the air velocity along these lines for different wind velocities and a 50 % permeability of the wall section. Figure 1b also shows the predicted velocity along the lines when 15 m/s is used as a reference velocity, marked with a C. The correlation between the simulated velocity and the predicted velocity with 15 m/s as basis is very good for 25 m/s and 10 m/s and somewhat poorer for 5 m/s. This means that the flow pattern is independent of the wind velocity and that it is adequate to use only the simulation with a velocity of 15 m/s in 10 meter height to assess also other wind velocities.

Figure 2 shows the maximum allowed wind velocity in 10 meter height outside of the building, above which the air velocity in the residential zone exceeds 0.3 m/s. A similar graph is produced for the 3.9 m/s threshold given in table 1. Combining the two graphs produces figure 3, which shows the allowed combinations of wind velocity (u_{10}), permeability of the wall and outdoor temperature (T) for wind direction 270°. A regression analysis of the data presented in figure 3 produces an algorithm for automatic control of the permeability shown in equation 2.

$$permeability(u_{10}, T) = 141.5 - 10.93u_{10} + 0.2853u_{10}^2 - 0.002486u_{10}^3 + 10.75T \quad (eq.2)$$

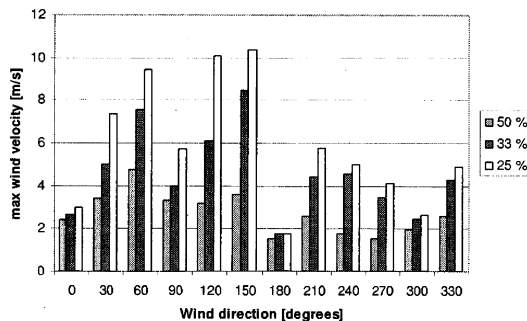


Figure 2 Maximum the maximum allowed wind velocity in 10 meter height

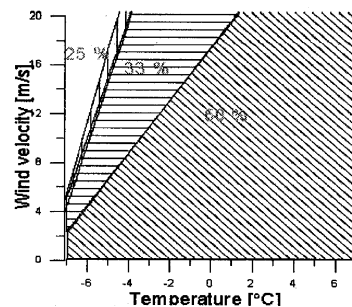


Figure 3. Allowed combinations of wind velocity, permeability of the wall and outdoor temperature for wind direction 270°

3.2 Full-scale measurements

Figure 6 shows the air velocity measured longitudinally from the ventilation openings in 2.3 meter height and the simulated air velocity in the same height.

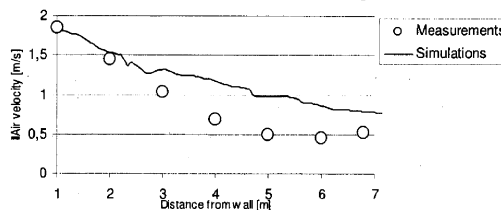


Figure 4. Measurements and simulations of the longitudinal air velocity profile in 2.3 meter height

REFERENCE

- Gregory, N. G. (1995). "The role of shelterbelts in protecting livestock: a review." *New Zealand Journal of Agricultural Research* 38(4): 423-450.
- Holmes, C.W., Sykes, A. R. (1984) "Shelter and climatic effects on livestock". Pp. 19-35. in Shelter research needs in relation to primary production, Sturrock, J.W.. *Water and Soil Misc. publ. no. 59*
- Miguel, A. F. (1998). "Airflow through porous screens: from theory to practical considerations." *Energy and Buildings* 28(1): 63-69.

Experimental Study of Airflow in Naturally Ventilated Double Skin Facade

Authors: Yuan Yuan^a, Xiaofeng Li, Yingxin Zhu, Borong Lin

^a*Department of Building Science, Tsinghua University, Beijing, 100084, China*

ABSTRACT: Along with the worldwide use of Double Skin Facade (DSF), the detailed research of the airflow in ventilation cavity, especially for naturally ventilated DSF, becomes extremely significant. However, until now, there is a lack of field measurements and validate simple ventilation models.

In this paper, field measurements are conducted to analyze the temperature distribution and air flow rates in naturally ventilated DSF. Different field measuring techniques are analyzed. Tracer gases are recommended to determine the air flow rate. On the base of measured data, a simple ventilation model is developed to predict the air flow rate in DSF cavity.

KEYWORDS: Double Skin Facade, Air flow rate, Field measurement, Tracer gases, Natural ventilation, Ventilation model

1 INTRODUCTION

Cavity ventilation system is used to remove the cavity heat in Double Skin Facade (DSF), and has a great contribution on the improvement on the DSF thermal performance. Former researchers have conducted some experiments on DSF natural ventilation system. Dirk² measured the pressure differences through inlet grids to determine the air flow rates, and the experimental results are compared with a room ventilation model. R.Letan⁶ simulated the building ventilation through DSF using CFD, and conducted laboratory experiments to verify the simulation results. Park¹ also conducted a field measurement of DSF and carried out real time control of DSF.

However, there are two deficiencies in the former studies. First, there is a lack of discussion on measuring methods, such as surface temperature and air flow rate. Second, the room ventilation model should be developed to suit the DSF. Consequently, this paper analyzed the different measuring techniques used for DSF, and takes the first step to develop a DSF ventilation model.

2 EXPERIMENTS

2.1 Measuring Techniques

2.1.1 Temperature

All contact sensors on glazing surface, such as thermocouple, show non-negligible temperature increase because of solar absorption by the sensor and electric wires. In this paper, a comparison experiment is conducted to determine the measuring accuracy. Two 2mm T type thermocouples are fixed in the centre of one glazing; one is explored under the solar intensity of 350 W/m², the other is not. The results show that the explored sensor showed 0.5°C higher than the shaded one. So it is concluded that for 0.2mm T type thermocouple, the measuring error of glazing surface temperature is less than 0.5°C, which is acceptable for a field measurement.

2.1.2 Air flow rate

In literatures, three techniques are described to measure airflow rates of naturally ventilated DSF: (1) the pressure difference across inlet/outlet (Dirk Saelens²); (2) wind speed in cavity by

placing anemometers (Park¹); (3) tracer gas techniques. The first two methods can only be used in some DSF with particular structure. While, tracer gas is a more common way, which is mostly used for the building ventilation, but seldom used in DSF. In this paper, tracer gas in a constant emission way is used to determine the air flow rate in naturally ventilated cavity. One thermal bulb-type anemometer is also placed in the cavity to give a referenced air velocity.

2.2 Location and Equipments

The Experimental Double Skin facade unit is located in the Low Energy Demo Building of Tsinghua University, China. The experimental unit is on the third floor, in the middle of the east facade. This unit is consisted by a single pane exterior glazing, an aluminium blind and a double panes interior glazing. Two top-swing window openings in the facade are electrically controlled (fig 1 and fig 2).

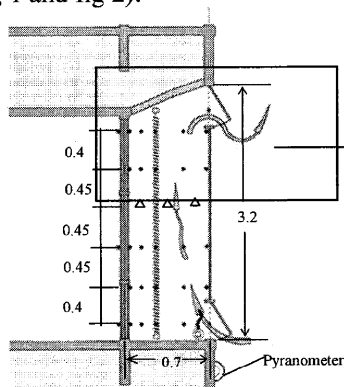


Figure 1. Location of sensors

• presents the thermocouples. There are about 30 temperature points. $\odot \mid$ presents the tracer gas inlet, which was a plastic tube with a diameter about 5cm. This tube is as long as the cavity width. There are more than 2000 small holes on the tube, which ensure a steady and linear release of tracer gases. There are 6 sampling points (Δ) in the middle height of the cavity.

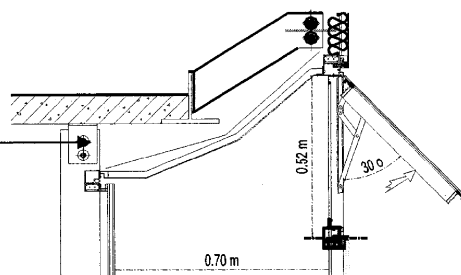
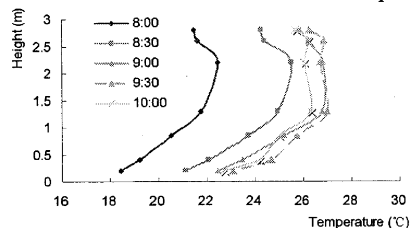
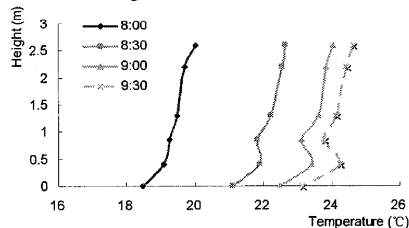


Figure 2. Dimension of openings

2.3 Results

The experiments are conducted during 8:00am~10:00am in each day from 6th Sep to 15th Oct. 2005. Data are used when the outdoor wind velocity is lower than 2.5m/s in the height of 10m with a direction of north, and the solar intensity on east facade is 200-600W/m². The outdoor temperature rises from 16°C to 30°C.

In the cavity, the blind shading divides the cavity into two spaces. The temperature distributions in both exterior and interior space are showed in fig 3.



(a) Time: 8:00~10:00 am Exterior space

(b) Time: 8:00~9:30 am Interior space

Figure 3. air temperature vertical distribution with a maximum solar intensity of 330W/m²

The above diagrams show that in exterior space, the temperature is stratified vertically. Where the height is lower than 1.4m (half of total height), the temperature rises with height linearly. While, in the height of over 1.4m, the temperature keeps constant and small fluctuates around the outlet. However, in interior space, the temperature shows little vertical difference, less than 2°C.

Figure 4 shows the cavity air flow rates as a function of inlet and outlet temperature differences. Least-squares procedure indicates that there is a proportional relationship between air flow rates and the square root of the temperature difference. One more point should be paid attention is that when the air temperature difference is smaller than 2°C, the air flow rates are no longer reduced. It can be explained by the wind effect. In small temperature difference cases, the wind effect should be taken in to account.

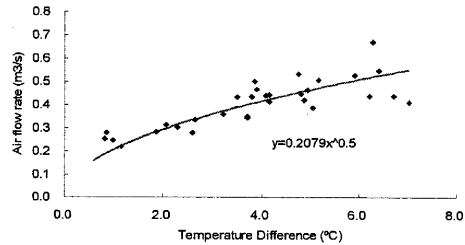


Figure 4. Air flow rate as a function of temperature differences

3 SIMULATION MODEL

3.1 Ventilation Model

The airflow in the cavity is caused by two driving forces: buoyancy effect and wind pressure. The ASHARE Handbook Fundamentals⁵ indicates that when the wind velocity in the height of 10m is lower than 2.5m/s, and the wind direction is north, wind pressure difference between two openings is about 0.02 *pa*, which is much more less than stack effect, so can be ignored.

Karl T Andersen⁴ proposed air flow rate caused by buoyancy effect as follow.

$$Q = A \sqrt{\frac{2(\Delta p_1 + \Delta p_2)}{\rho_o (\xi_1 + \xi_2)}} \quad (1)$$

Where

A = the area of the openings; $\Delta p_1, \Delta p_2$ = the pressure differences across the openings

ξ_1, ξ_2 = the form resistance of openings; Q = Air flow rate

Experiment results (fig 3) shows that the air temperature vertically stratifies. There is a critical height (H_r). Where the height is lower than H_r , the temperature rises with height linearly. In the height of over H_r , the temperature keeps constant. Consequently, a parameter $\eta = H_r/H$ is defined to describe the temperature distribution. When $\eta = 0$, the temperature is uniform, and when $\eta = 1$, the temperature linearly rises. η value has a strong correlation with the structure of DSF. Generally, for one-story DSF with a relative wide cavity, η is about 0.5, while for multi-story DSF with a narrow cavity, η trends to be 1.

According to the relationship between density and temperature of ideal air, one can get:

$$Q = A \sqrt{\left(1 - \frac{\eta}{2}\right) \frac{2(T_r - T_o)gH}{T_r(\xi_1 + \xi_2)}} \quad (2)$$

3.2 Comparison between model and experiment results

Table 1 and figure 5 give the results of model and experiments, where the form resistances are $\xi_1 = 12.3$; $\xi_2 = 15.1$, determined by CFD simulation. Table 1 state that the buoyancy model with the η of 0.5 gives the best predicted values, especially in the range of 3 to 5 °C. However, in the low range (2-3 °C) and the high range (above 5 °C), the predicted values are about 10% higher than measured values. Considering the measuring errors, this deviation is acceptable.

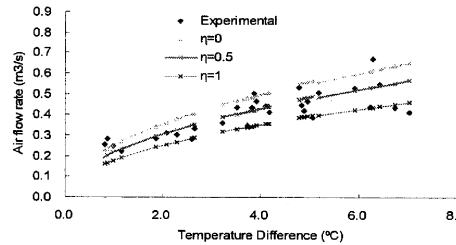


Fig 5. Air flow rate as a function of temperature difference, outdoor wind speed < 2m/s

Table 1 the average values of predicted and measured air flow rate

Temperature Difference	Air flow rate (m³/s)						
	Measurement	$\eta=0$	Dev.	$\eta=0.5$	Dev.	$\eta=1$	Dev.
2-3 °C	0.31	0.39	26%	0.34	9%	0.29	-6%
3-4 °C	0.42	0.48	16%	0.42	1%	0.35	-17%
4-5 °C	0.45	0.53	18%	0.46	2%	0.38	-15%
>5 °C	0.49	0.62	26%	0.54	9%	0.44	-10%

4 CONCLUSIONS

In this paper, the airflow in naturally ventilated DSF has been experimentally analyzed. The accuracy and applicability of different measuring techniques are analyzed. As a result, the 0.2mm T type thermocouples for surface temperature measuring and tracer gases for airflow rate are recommendable.

According to the experimental data, the air temperature is stratified vertically in the cavity. Air flow rate shows a proportional relationship with the square root of the inlet and outlet temperature difference.

A simple ventilation model is developed, which predicts best when the temperature differences are 3-5 °C.

However, the effect of cavity blind shading is ignored in this simple model. A detailed model considering the influence of cavity shading and all kinds of opening forms is being developed.

REFERENCES

- 1 Park, Cheol-Soo; Augenbroe, Godfried; Messadi, Tahar; Thitisawat, Mate; Sadegh, Nader, Calibration of a lumped simulation model for double-skin facade systems, energy and buildings, v36, n 11, November, 2004, p1117-1130
- 2 Dirk Saelens, Hugo Hens, Experimental evaluation of airflow in naturally ventilated active envelopes, Journal of thermal env. & bldg. sci., vol.25-October 2001
- 3 C.S. Park, Occupant responsive optimal control of smart facade systems, Ph.D. thesis, Georgia Institute of Technology, 2003.
- 4 Andersen, K.T., 1995, Theoretical considerations on natural ventilation by thermal buoyancy, ASHRAE Transactions, 101(2): 1103 1117.
- 5 Chapter 16, ASHARE Handbook Fundamentals, ASHRAE, 2001.
- 6 R.Letan etc. Passive ventilation and heating by natural convection in a multi-storey building, Building and Environment, 2002

Aerodynamic Effects of Different Ventilation Methods on Buses

Marcus Thomas, Rajnish N Sharma*, Michael Kilduff

*Department of Mechanical Engineering, The University of Auckland, Private Bag 92019,
Auckland, New Zealand*

ABSTRACT: This paper describes a study of the aerodynamic effects of open windows and mechanical ventilation units on large vehicles. Predominantly it considers a MAN city bus, and the aerodynamic effects resulting (including Helmholtz Resonance excitation) from window openings and presence of air conditioning units. Analysis was conducted using both wind tunnel testing and Computational Fluid Dynamics modelling. The results of both analyses, CFD and Wind tunnel, show an increase in drag of up to 2% for single openings, and up to 5% for multiple openings. The analysis also revealed for Strouhal numbers between 0.27 and 0.4 the Helmholtz resonance present was significantly large enough to represent an increased breathing mode. Analysis of air conditioned buses shows an increase in drag of up to 3% (for standard positions on the roof), similar to the increase in drag expected of multiple openings, but a decrease in drag of up to 12% when the unit is shifted forward.

KEYWORDS: Bus aerodynamics, Window openings, CFD, Drag

1 INTRODUCTION

As the price of petrol increases, the demands on public transport will increase. Good business sense shows that to improve the profitability of a transport business, one must maximize the capacity while maximizing the fuel economy. Buses and heavy transport are largely restricted by a need to maximize cargo capacity; the aerodynamics of such vehicles becomes a secondary feature [1]. Previous work at the University of Auckland [2, 3] has investigated methods of reducing drag by changing the external shape of the bus. The results of these investigations were quantified using wind tunnel testing, and CFD analysis was used primarily to visualize flow patterns. As air-conditioning units count for a small amount of the fuel usage in their running alone, this project looks to assess the aerodynamic effects of using natural ventilation compared to air-conditioned ventilation. The results of this study are quantified by CFD analysis and wind tunnel testing, with CFD also providing flow visualization.

2 METHODOLOGY

This study compares the drag of three variants of the MAN city bus (see Figure 1): (1) a bus with a roof-mounted air conditioning unit; (2) a bus with open windows; and (3) a control bus with no windows or air conditioning. Three variants of the air conditioned bus were tested, with the air conditioning unit in two standard places (over the front and rear wheels), and at the leading edge. Eight standard variants of the windowed bus as shown in Figure 2 were individually analysed for drag and lift, as well as internal pressure variation in the bus cavity due to the excitation of the Helmholtz resonance.



Figure 1. MAN Buses with narrow openeable windows

2.1 Experimental Methodology

Experiments were conducted in the University of Auckland de-bray wind tunnel. A 1/20th scale model of the MAN 10.160 series bus was specifically developed, taking care to ensure the internal cavity representation, opening (window) and bus dimensions were preserved. For wind tunnel testing of blunt bodies, accepted practices adopted included neglecting moving wheels and ground in the raw results, with actual results including correction factors. A false ground was erected in the tunnel (see Figure 3) to model the flow over the ground expected in a real life situation, and to minimize the wall/ground boundary layer effects present at the bottom surface of the wind tunnel. A three component force balance was used to measure the lift and drag forces, while a Setra differential pressure transducer was mounted inside the vehicle cavity for measuring internal pressure. The main errors factored into analysis were blockage error and drag on the struts [4]. Blockage of the scale model was 3.5%, and appropriate corrections were made to the data.

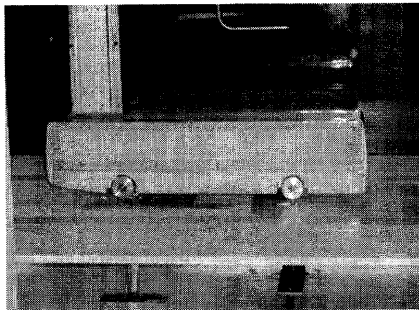


Figure 3. Bus model in the wind tunnel

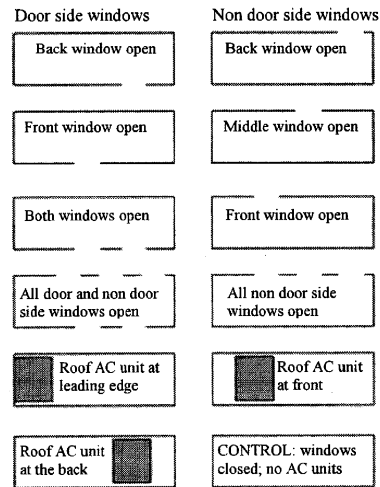


Figure 2. Window openings and AC unit locations

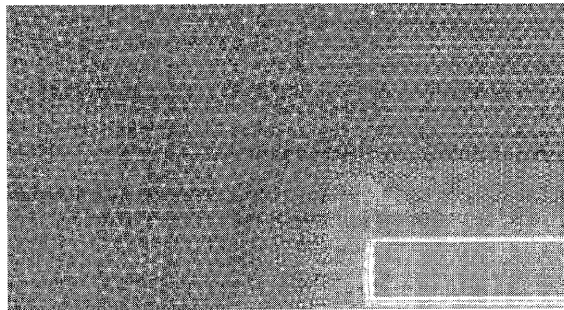


Figure 4. CFD Mesh around the bus

2.2 CFD Methodology

Analysis of the flow over the bus was conducted in Ansys Workbench / CFX 5.7.1. The dimensions of the computational domain were determined through standard practices for blunt bodies, including extending the domain length ten model heights to the rear [5, 6] and five model heights to the front, and at least ten model heights vertically. The domain was enlarged to five obstruction heights by five obstruction widths, to allow for flow disturbance [7, 8]. CFX employs an unstructured 3-D mesh, with user defined mesh settings. For bus simulations, standard near surface mesh resolution was attained, in particular around the body edges and openings (see Figure 4). A 50km/hr wind speed was used, and the K-Epsilon turbulence model was employed due to its ability to accurately predict near-wall flows. Each simulation was converged to the target RMS residual of 1.0×10^{-3} , considered appropriate for engineering applications. Simulations were used

for flow visualization, for example as shown in Figure 5, however upon investigation quantifiable results were also obtained.

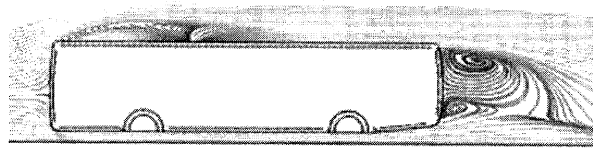


Figure 5. Streamlines from CFD for a standard bus

3 RESULTS AND DISCUSSION

Table 1. Drag coefficients from CFD and wind tunnel testing for buses with open windows

below shows drag coefficients from CFD and wind tunnel testing of windowed buses. As can be seen, the CFD data and the wind tunnel testing results demonstrate good agreement, showing trends of an increase in drag with opening of windows in the bus. This amount of drag can be low enough (measured to be 1-2%) with only one open window to be insignificant. However when multiple windows are open this increases the drag significantly by up to 4%. The differences between CFD and wind tunnel test results is consistent across the simulations, and is attributed to the K-epsilon model and scale errors.

Table 1. Drag coefficients from CFD and wind tunnel testing for buses with open windows

	CFD	% Diff [#]	Wind Tunnel	% Diff [#]
Control Bus	0.837		0.866	
Front non-door side window open	0.844	0.8	0.874	1.0
Back non-door side window open	0.839	0.2	0.856	-1.1
Middle non-door side window open	0.843	0.7	0.872	0.7
Front door side window open	0.842	0.6	0.875	1.1
Back door side window open	0.843	0.7	0.884	2.1
All non-door side windows open	0.861	2.8	0.878	1.5
All door side windows open	0.842	0.5	0.865	-0.1
All windows open	0.865	3.3	0.896	3.4

Note: # %Diff = $100 \times (C_D - C_{D(\text{control})}) / C_{D(\text{control})}$

Table 2 shows drag coefficients from CFD and wind tunnel testing of air conditioned buses. Once again CFD analysis and wind tunnel testing are in good agreement, which shows when considering an air-conditioning unit the placement has to be taken into account. The same trends were revealed in both the CFD and wind tunnel testing that shifting the air-conditioning unit forward into the area usually occupied by the separation bubble reduces the drag experienced by the body. This is quantified to be 4-12% depending on the location, with the greatest benefit with a air conditioning unit located at the leading edge. As expected, the drag increases on the bus if the air-conditioning unit is placed at the rear, by 2.3%. This is in the same range as that of a multiple window bus, but is a significant increase compared to a single open window.

Table 2. Drag coefficients from CFD and wind tunnel testing for buses with air conditioning units

	CFD	% Diff C_D	% Diff F_D	Wind Tunnel	% Diff C_D	% Diff F_D
Control Bus	0.837			0.866		
Front AC-Unit	0.744	-11.2	-4.1	0.730	-14.5	-7.5
Leading Edge AC-Unit	0.682	-18.6	-12.1	0.699	-18.1	-11.3
Back AC-Unit	0.793	-5.3	2.3	0.808	-5.2	2.3

A sample of the type of internal pressure fluctuations obtained with an open window is shown in Figure 6 that demonstrates the presence of intense fluctuations believed to be due to Helmholtz resonance. Since the resonance frequency is changing, it is clear that grazing flow excitation is involved. This may be beneficial from the point of view of natural ventilation, which should be investigated further.

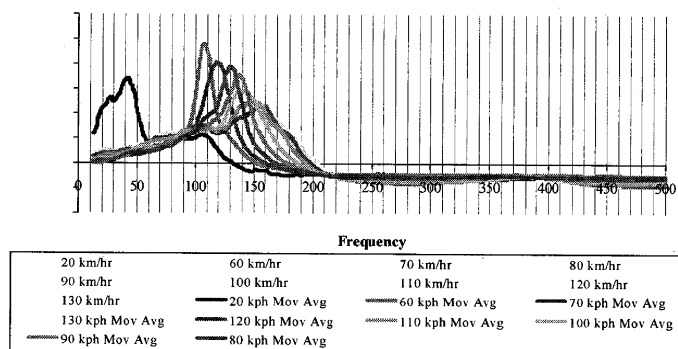


Figure 6. Internal pressure spectra for non door side back open window

4 CONCLUSIONS

From the testing performed on buses with open windows, the following conclusions are made:

- Wind tunnel and CFD analysis agree, showing a trend of increase in the drag coefficient, with the number of open windows. With a single window opening a 2% increase was obtained while multiple openings produced increases of up to 4%.
- Spectral analysis shows that large fluctuations in internal pressure result from grazing flow excitation of Helmholtz resonance.

From the testing performed on Air-conditioned buses, the following conclusions were made:

- In the standard position at the back, the air-conditioning unit increases drag force by 2%.
- Shifting the air-conditioning unit to the front of the bus reduces drag by over 11%.

5 REFERENCES

1. Cooper, K.R. (2004) Commercial Vehicle Aerodynamic Drag Reduction: Historical Perspective as a Guide, *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*. National Research Council of Canada, Ottawa, Canada.
2. Lee, C.J., and Oram, W. (2003) Aerodynamic Drag Reduction of Buses. Department of Mechanical Engineering, Auckland, New Zealand.
3. Haines, J.G., and Chadwick, D. (2004) Aerodynamic Drag Reduction of Intercity Buses. Department of Mechanical Engineering, Auckland, New Zealand.
4. Filippone, A. (2003) *Advanced Topics in Aerodynamics*, [online] Available from: <http://www.aerodyn.org> last viewed on 14th July 2005.
5. Cooper, K., Gerhardt, H.J., Whitbread, R., Garry, K.P., and Carr, G.W. (1986) A Comparison of Aerodynamic Drag Measurements on Model Trucks in Closed-Jet and Open-Jet Wind Tunnels. *Journal of Wind Engineering and Industrial Aerodynamics*, **22**, 299-316.
6. Ortega, J.M., Dunn, T., McCallen, R., and Salari, K. (2004) Computational Simulation of a Heavy Vehicle Trailer Wake, *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*. Lawrence Livermore National Library.
7. Hinterberger, C., García-Villalba, M., and Rodi, W (2004) Large Eddy Simulation of Flow Around the Ahmed Body, *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*. Institute for Hydrodynamics, University of Karlsruhe, Germany.
8. Bayraktar, I., Bayraktar, T., and Baysal, O. (2004) Computational Parametric Study on External Aerodynamics of Heavy Trucks, *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*. Old Dominion University, Norfolk, VA 23529.

Comparison between different methods for urban ventilation study: the case of the city of Belo Horizonte, Brazil

Daniele Gomes Ferreira^a, Eleonora Sad de Assis^b

^a*The Federal University of Minas Gerais, 116 Três de Maio Street, Belo Horizonte, MG, Brazil*

^b*The Federal University of Minas Gerais, 697 Paraíba Street, Belo Horizonte, MG, Brazil*

ABSTRACT: The accelerated growth of big cities has caused changes in urban climate. The density and configuration of urban areas are responsible for phenomena as heat islands and reduction of wind speed. The city of Belo Horizonte in Brazil is an example of these kinds of changes. In the city these phenomena are accentuated because of rough topography and its tropical climate. This paper presents results of a research about natural air flow in this city comparing two methods: computational interpolation of data from local weather stations and physical simulation of a scaled model in a wind tunnel. Results indicated the effects of topography in the air flow and validate the hypothesis produced by computational simulation.

KEYWORDS: urban ventilation, urban planning, environmental comfort

1 INTRODUCTION

Urban ventilation is one of the main climate variables affected by the growth of cities in tropical areas. The density and configuration of urban areas are responsible for changes in direction and wind speed, often causing situations of thermal discomfort and bad quality of air. Ventilation in urban areas contributes with evacuation of heat produced by buildings and human activities in addition to spread the air pollution.

The distribution and characteristics of the wind over a region are determined by several global and local factors as the seasonal global distribution of air pressure, the rotation of the earth, daily variations of temperature and the topography of the given region and its surroundings (Givoni, 1976). So, the winds condition study has to approach these scales to show global effects until local differences in air flow.

In local scale [1], topography is an important element that changes mainly the predominant wind paths in specific areas. In general, the major roughness of the urban area cause speed reduction and aerodynamic effects restricts to specific regions due to higher wind friction with construction surfaces. Consequently, the thermal conversion becomes prejudiced and temperatures in urban spaces can increase (Oke, 1981; silva, 1999). Most of Brazilian cities are situated in tropical latitudes where the effects of high excess temperatures in urban areas become greater due to irregular occupation and increased emission of pollutants. In the city of Belo Horizonte, high level of temperature and local wind patterns are accentuated by illegal occupation of valleys and hillsides, rivers drainage systems and poor distribution of green spaces.

In Belo Horizonte city rough relief is a feature of the landscape. Therefore a study had been made to show the topographical influence on regional mean wind conditions since the relief suffers few changes during the years and its effects can be previewed and applied in urban planning decisions. This paper presents a comparison between two different methods applied to investigate natural air flow in the city. Studies on vulnerable areas with low ventilation conditions are scarce in the city of Belo Horizonte so this research is important for planning purposes.

2 BELO HORIZONTE CITY: THE STUDY AREA

Belo Horizonte is located at 19°55' South latitude, 43°56' West longitude and at a mean altitude of 875 meters (Fig. 1). The regional climate is considered as tropical of altitude however climatic classifications indicate a local changing to a warmer and drier condition (Aw, according to Köppen classification). This indicates a possible urbanization influence on the local climate (Assis, 1990). The annual average temperature is 21.1°C and the maximum and minimum averages are 27.1°C and 16.7°C respectively (Brasil, 1992).

The city major altitudes are located on the Southern and Southeast of the town because of the Serra do Curral (a local mountain). The maximum altitude is approximately 1.480 meters. The Western side also presents high altitudes that reach 970 meters. The North portion of the town presents smooth topography and altitudes are between 675 and 850 meters.

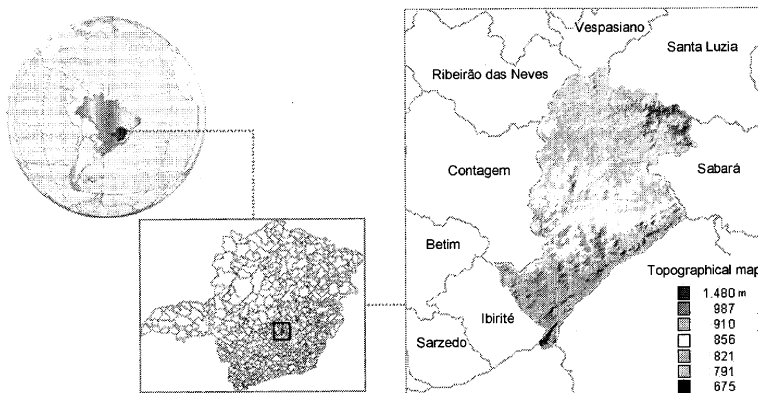


Figure 1: Topographical map of Belo Horizonte city. (After Ferreira, 2004)

3 METHODOLOGY

The methodology used in this work contains two main phases: computational interpolation of data from local meteorological stations and physical simulation of a scaled model. The first phase included database of wind speed and direction from thirteen fixed meteorological stations located in Belo Horizonte and in its surroundings – four stations are placed in the city (Fig. 2). It was important to feature surrounding areas due to physical barriers that influence winds conditions, direction and speed. These are from 1998 and 2001 and were used to calculate monthly averages of four years. The meteorological stations have different ways to register data: some of them use the hourly series and others register information by daily or monthly means. To equalize the datum collection, it was applied the treatment method used by the standard stations in Brazil, which gives the climatological data based on World Meteorological Organization (WMO) technical regulations (Brasil, 1992). They were statistically analyzed to represent an annual mean wind variation and plotted on a topographical map of the city using a surface mapping software.

The second phase was a qualitative physical simulation of a topographical city model in a wind tunnel. The model was made with a scale of 1:40.000. Topography was represented by cork blades over a plane wood support measuring 90x70x1 cm. The model was tested in a boundary layer wind tunnel using the sand erosion method representing the dominant winds condition. In different moments of the experiment were made photographic registers to analyze the effects of the wind speed variation. The results showed the influence of local topography on air flow and

indicate bad ventilated areas in the city. Physical model tests validated the computational simulation hypothesis.

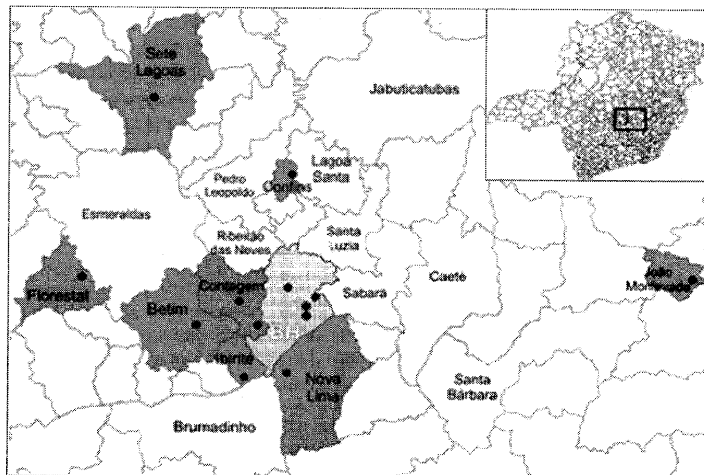


Figure 2: Fixed Meteorological Stations located in Belo Horizonte and surroundings. (After Ferreira, 2004)

4 RESULTS AND CONCLUSIONS

Results show a generic view of mean wind conditions in Belo Horizonte city and the areas with bad natural ventilation. In the first phase of study, vector map showed predominant wind direction and speed variation in the analyzed area (Fig. 3). The dominant direction expressed in the results was the East. Major speeds were sited in Northern and Central areas which had mean winds of even 2,8 m/s. In these areas, the relief is smoothness than in the Southern. this means that wind can flow easier than in mountainous regions. The Southern and Southeastern regions of the town exhibited the lowest wind speed which reach less than 1,5 m/s. Probably, the mountain range (Serra do Curral) acts mainly as a barrier to change wind speeds and direction. The direction of wind in the extreme Southern of the city was also modified to the Northeast.

Physical simulation in a boundary layer wind tunnel pointed out the air flow trends validating computational hypothesis. Besides, it showed areas which have bad natural ventilation conditions. They are on the lee side of mountainous obstructions and shows the topographical influence in wind conditions. The higher was an obstruction, the bigger was the bad ventilated wind area. Southern region was the place which had the most areas in bad ventilation conditions due to irregular relief. In the computational simulation the less wind speed occurred in this area.

To translate the results acquired in physical simulation to a useful map that could be understood by urban planners and designers, a cartographic representation was done using the photo took after the end of the wind tunnel experiment (Fig. 4). First, the picture was treated in an image-editing software to make a layer of areas which gathered sand during the test. Then this image was overlayed on a map of the city neighborhoods. Based on these results, it is possible to study specific places which might be not properly ventilated.

As a conclusion, it is important to attest the study relevance. The methods used in this study are complementary. Computational simulation gives support to physical simulation since it demonstrates the dominant wind conditions which are an essential data to test the scaled model. In the other hand, physical simulation validates computational method.

In this study, the physical model had a problem with its scale in correspondence with the wind tunnel parameters. So it was not possible to extract numerical data and then wind tunnel tests were only qualitative. However the results could be applied in urban planning indicating areas which should be protected by urban legislation against a strong urban development. In the same way, computational simulation provides aid to urban planning showing the main paths and directions of wind. These results point out places which should be preserved due to their importance as wind approach in the city.

This kind of research may not only contributes to a better understanding of wind paths in the city site but mainly it may be used to improve the land use laws having in sight to preserve good ventilation conditions. Acknowledging areas well and bad ventilated in urban spaces could assist on streets orientation design, building orientation establishment and distribution of green areas, tools which contributes to a better quality of life in cities.

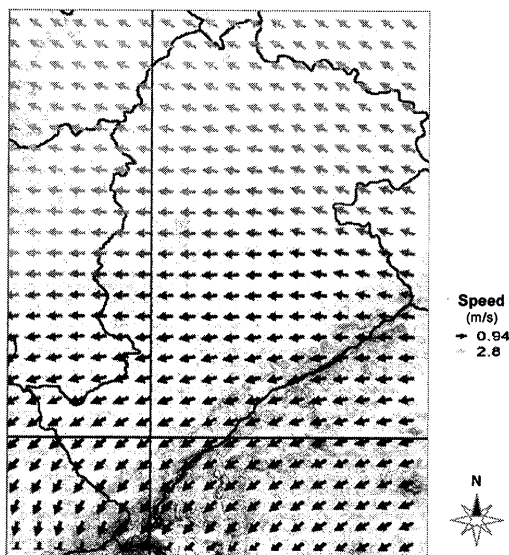


Figure 3: Mean wind conditions – period from 1998 to 2001. (After Ferreira, 2004)

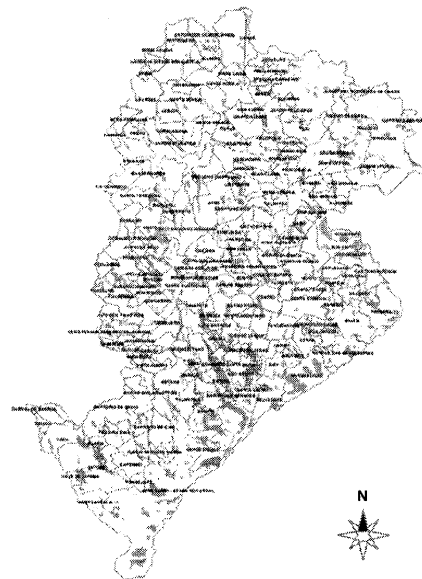


Figure 4: Neighborhood map with bad ventilated areas. (After Ferreira, 2004)

[1] Climate scale classification defined by Monteiro (1976), according to Cailleux and Tricart.

5 REFERENCES

- 1 E. S. Assis, Mecanismos de desenho urbano apropriados à atenuação da ilha de calor: análise de desempenho de áreas verdes urbanas em clima tropical, Rio de Janeiro, Faculdade de Arquitetura e Urbanismo da Universidade Federal do Rio de Janeiro, 1990. 165 p. (Master Degree Research in Architecture).
- 2 Brasil (Ministério da Agricultura e Reforma Agrária – Departamento Nacional de Meteorologia – DNMET), 1992. Normais Climatológicas (1961-1990). Brasília, DNMET, Brazil.
- 3 D. G. Ferreira, Estudo da Ventilação Urbana na Cidade de Belo Horizonte, MG, Belo Horizonte, Escola de Arquitetura da Universidade Federal de Minas Gerais, 2004 (Undergraduate Program on Scientific Research, Department of Technology of Architecture and Urban Planning, School of Architecture).
- 4 B. Givoni, Man, Climate and Architecture. 2ª ed. Applied Science Publishers Ltd, Londres, 1976.
- 5 T. R. Oke, Boundary Layer Climates, New York, NY: Methuen & Co., 1981.
- 6 C. A. F. Monteiro, Teoria e clima urbano, São Paulo, Faculdade de Filosofia, Letras e Ciências Humanas da Universidade de São Paulo, 1976. 181 p. (Thesis, Geography Department).