VALIDATION OF A HUMAN EXHALATION FLOW SIMULATION IN A ROOM WITH VERTICAL VENTILATION

I. Olmedo[1], M. Ruiz de Adana[1], P.V. Nielsen[2]
[1]University of Córdoba, Córdoba, Spain.
[2]Aalborg University, Aalborg, Denmark.

Abstract

The human exhalation flow is one of the most important and active sources of biological contaminants in a room. When a person is breathing, tiny particles that may carry pathogens such as viruses or bacteria are exhaled and can follow the airstream. These exhaled contaminants may cause a risk of infection to another person placed in the same room.

The powerful technique of Computational Fluid Dynamics (CFD) is used in this research to predict the dispersion of a contaminated human exhalation flow in a room, which is equipped with vertical ventilation.
An experimental study in a full-scale test chamber: 4.1 m (length), 3.200 m (width) and 2.7 m (height), is carried out to validate the numerical simulations. The incoming air is distributed through a ceiling mounted textile diffuser, which generates both downward and upward flow areas in the room. A breathing thermal manikin with a time dependent breathing function (exhaling through the mouth and inhaling through the nose) is used. A tracer gas, N_2O is used to simulate the gaseous substances, which can be considered as biological contaminants, exhaled by the manikin.

During the experiments five concentration probes and five anemometers are placed along the centreline of the exhalation jet in order to characterize the exhalation and validate the CFD simulation.

The CFD simulation is carried out using the average velocity of the breathing function of the manikin. This boundary condition simplifies the simulation process and decreases simulation time. The results show a quite accurate prediction of the velocity and concentration values along the centreline of the exhalation jet as well as in the rest of the room.

Keywords: human exhalation flow, vertical ventilation, exhaled contaminants.

1 Introduction

In indoor environments the human exhalation flow can be a very significant source of biological contaminants. Some pathogens may be exhaled together with the air and spread in the atmosphere (Nicas et al.(2005), Wan and Chao, 2007, Morawska, 2006).There is an increasing research interest in gaining knowledge about how these contaminants can be disperse in the air and may provoke a risk of cross-infection to other people in the same room. As the source of transmission of diseases, such as measles or pulmonary tuberculosis, may be the human exhalation, an understanding of this exhalation process and its possible prediction with numerical simulation is of significant importance for many engineers and researchers. Computational Fluid Dynamics is being used successfully to predict the dispersion of exhaled contaminants in different indoor environments (Mui et al. (2009)). Different human respiration processes, such as sneezing or coughing, have also been studied in terms of CFD by several authors (Zhao et al. (2005); Gao and Niu, 2006) obtaining good agreements with experimental data.

However, the numerical simulation of a real cyclic breathing function to study the dispersion of exhaled contaminants generated during a breathing process demands a high computational capacity and may be very time consuming. One objective of this paper is to find out how the simplification of this boundary condition, the exhalation flow, may affect the dispersion and distribution of the exhaled contaminants.

In order to analyse this influence, this paper firstly analyses the trajectory of a human exhalation flow in a room with vertical ventilation generated by two textile diffusers. These diffusers generate a low impulse vertical downward flow that is supposed to be very effective in the removal of contaminants in a room. One breathing thermal manikin, which simulates a person, is used during the experiments. Experiments are designed to observe the trajectory of the exhalation flow and measure the velocity and contaminant concentration decays along the exhalation centreline.

Secondly, the same test is simulated numerically but using a constant exhalation velocity instead of the pulsing exhalation used during the experiments. In this way, it is possible to analyse how the boundary condition used to numerically simulate the human exhalation flow in a room may affect the distribution of contaminants in the room.

Finally, the simulated data is compared and validated using the experimental results. The different environmental conditions that may influence the exhalation trajectory are analysed.

2 Methodology

2.1 Experimental set-up

Experiments are conducted in a full-scale test room: 4.1 m (length), 3.2 m (width) and 2.7 m (height) ventilated by two textile diffusers, 1.2 m (length) and 0.6 m (width) each, placed next to each other in the ceiling of the room, 0.42 m from the front wall. The temperature of the air supplied by the diffusers and the air exchange rate during the experiments are set to 16°C ± 0.5°C and 5.6 h^{-1}, respectively. One single circular return opening, 12 cm (diameter), is placed at the height of 2.6 m in the center of the back wall following the recommendations regarding the high location of exhaust openings given by Qian and Li (2010) and Nielsen et al. (2010). One breathing thermal manikin, height 1.68 m, is used to simulate an average person breathing in the room. The manikin is placed in the upward flow area of the test room, close to the back wall and the exhaust opening and facing the right wall. A radiator, 0.55 m x 0.40 m x 0.05 m size is also used in the room during the experiment. The corresponding power heat loads of the manikin and the radiator are 94 W and 390 W, respectively. The arrangement of the test room can be seen in figure 1.

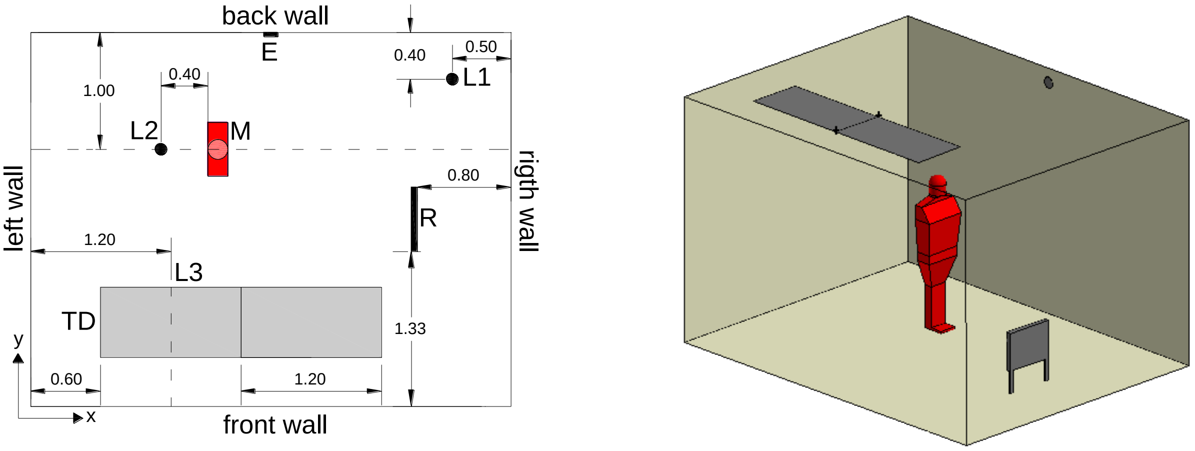

Figure 1. a) Horizontal section of the room with the positions of the manikin (M), radiator (R), textile diffusers (TD), exhaust opening (E) and measuring poles: L1, L2 and L3, b) View of the room with the manikin, the diffusers, the exhalaust opening and the radiator

Tracer gas, N_2O, is added to the exhalation flow of the breathing manikin. This gas is used to simulate the small contaminated droplet nuclei that may carry biological contaminants, (Gao and Niu, 2007; Yin et al., 2011). The manikin exhales 0.75 l of contaminated air per exhalation, with a frequency of 14.6 exhalation/min. The breathing of the manikin is a cycle function that consists of exhalation through the mouth and inhalation through the nose.

A total of nine thermocouples (type k) are used to take measurements in the room at two different poles, L1 and L2, see figure 1(a). Pole 1 (L1) is used to measure the temperature gradient in the room and consists of four thermocouples placed at the height of: 0.1m, 0.6 m, 1.1 m and 1.8 m. Pole L2 and is situated at 40 cm from the back of the manikin and consists of five thermocouples placed at the height of 0.1m, 0.6 m, 1.1 m, 1.8 m and 2.0 m.

The velocity profile of the diffuser is measured at twelve points with hot sphere anemometers along a horizontal line, L3, placed 0.50 m below the center of the left diffuser, see figure 1(a). The acquisition frequency is 100 ms with a precision of ± 5%.

The centreline of the exhalation flow is obtained by observation of the flow with smoke, see figure 2. Five anemometers and five concentration tubes are placed along this centre line in order to measure the velocity and concentration decay, respectively. The results of these measurements are compared with the data obtained with the numerical simulation.

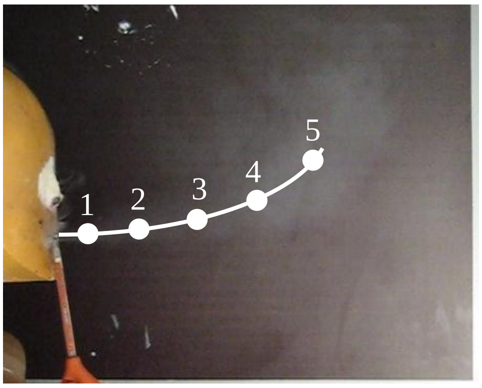

Figure 2. Smoke visualization of the exhalation flow with smoke (Each white dots represent one anemometer and one concentration tube placed along the exhalation centreline during the experiment)

2.2 CFD simulations

The numerical simulation is carried out using the CFD code Fluent (Fluent, 2005) with mesh generation using Gambit 2.6. A transient simulation was performed to study the time evolution of the breathing and contaminant transport. For the simulations, the re-normalization group (RNG) k-epsilon model was used as turbulence model as it produced accurate validated results for indoor airflow, temperature and contaminant distribution in several previous studies (He et al. (2005), Gao and Niu, 2006). The effect of radiation was neglected due to a computational restriction.

The grid generated consists of 638,190 cells.

The heat sources placed in the room, the manikin and the radiator, generate buoyancy flows around them. Therefore, the air density is defined as a linear function of temperature.

The manikin´s geometry was significantly simplified, see figure 1(b). However, the manikin geometry is defined maintaining the same surface area (1.4 m^2) and height of the real manikin (1.68 m^2).

The boundary conditions used for the simulation are shown in table 1.

***Table 1**: Boundary conditions during CFD simulation*

Object	*Boundary type*	*Velocity (m/s)*	*Temperature (°C)*	*Heat generation (W/m^2)*
Inlet	Velocity inlet	0.038	16	
Walls	Adiabatic wall	-	-	
Outlet	Pressure outlet	-	-	
Manikins mouth	Velocity inlet	2.16	34	
Manikin	Wall	-	-	57.7
Radiator	Wall	-	-	694.0

3 Results

During the experiment a significant influence of the front wall on the trajectory of the supply airflow was observe. Figure 3(a) shows the velocity results of the experiment and the numerical simulation along pole L3. It is possible to observe the acceleration of the flow close to the wall due to the Coanda effect generated during the experiment. This effect is also observed with the visualization of the flow with smoke, see figure 3(b). However, the simulation results show less significant flow acceleration close to the wall.

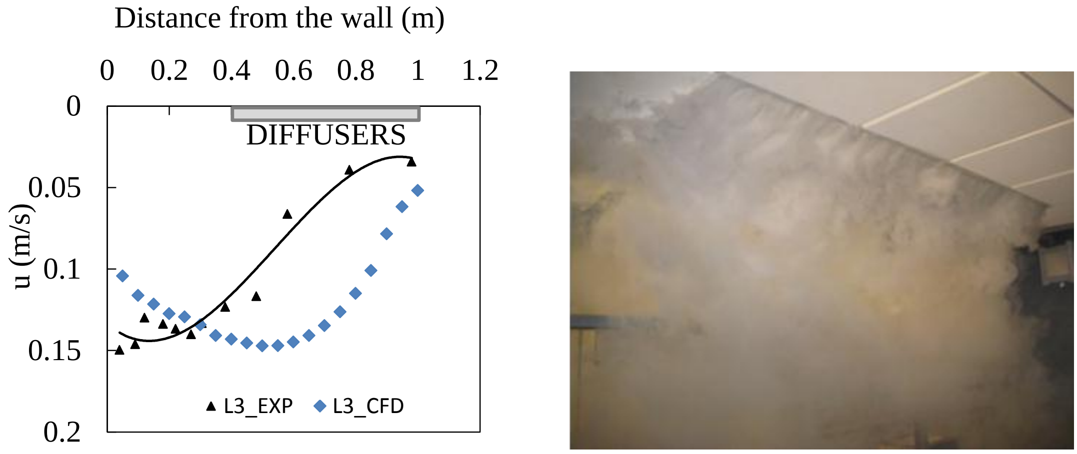

Figure 3. a) Velocity results along pole L3 during the experiments and the numerical simulations, b) Picture of the smoke visualization of the Coanda effect generated by the front wall of the room

The exhalation flows upward due to its high temperature and the influence of the upward flow generated by the ventilation strategy at the position of the manikin in the room, see figure 2. However, the simulation results show a more upward direction of the exhalation flow. Table 2 shows the position of the five points along the exhalation centreline during the experiment and for the numerical simulation.

***Table 2**: Position of the probes along the exhalation centreline during the experiments and during the simulation*

Probe	*Centreline probes position Experimental test*		*Centreline probes position Simulation*	
	H(m)	*x(m)*	*H(m)*	*x(m)*
1	1.52	0.01	1.52	0.01

2	1.53	0.10	1.54	0.10
3	1.54	0.23	1.64	0.25
4	1.56	0.44	1.75	0.40
5	1.58	0.65	1.85	0.45

The high trajectory difference may be due to the average velocity used as boundary condition for the exhalation flow. In consequence, the contaminant distribution is different from that obtained during the experiments. There is a high concentration of contaminants above the manikin between 20 and 30 cm from the manikin's mouth, see figure 4.

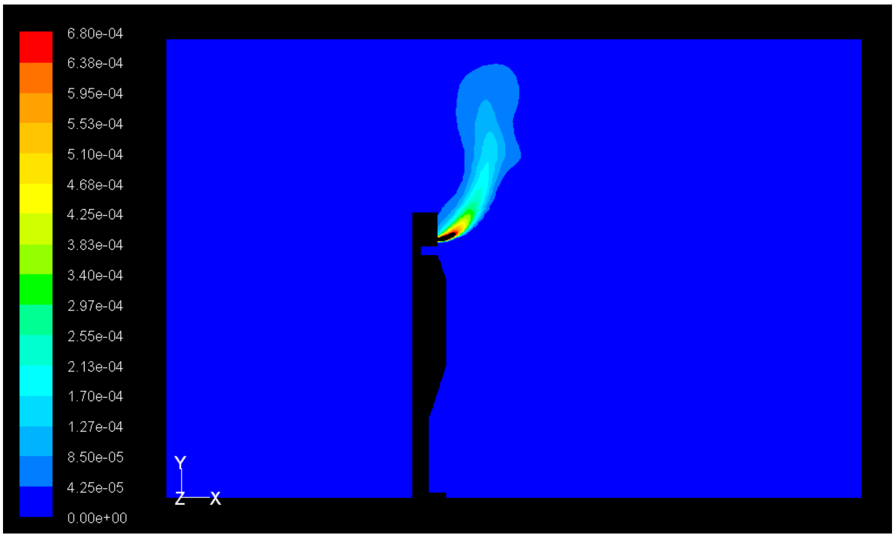

Figure 4. N_2O concentration contour at plane z=1 m (centreline of the manikin´s mouth)

Figure 5 shows the temperature results along L1 and L2. It is possible to observe a significant difference between the experimental and numerical results that may be due to the lack of a radiation model during the simulation due to computational restrictions.

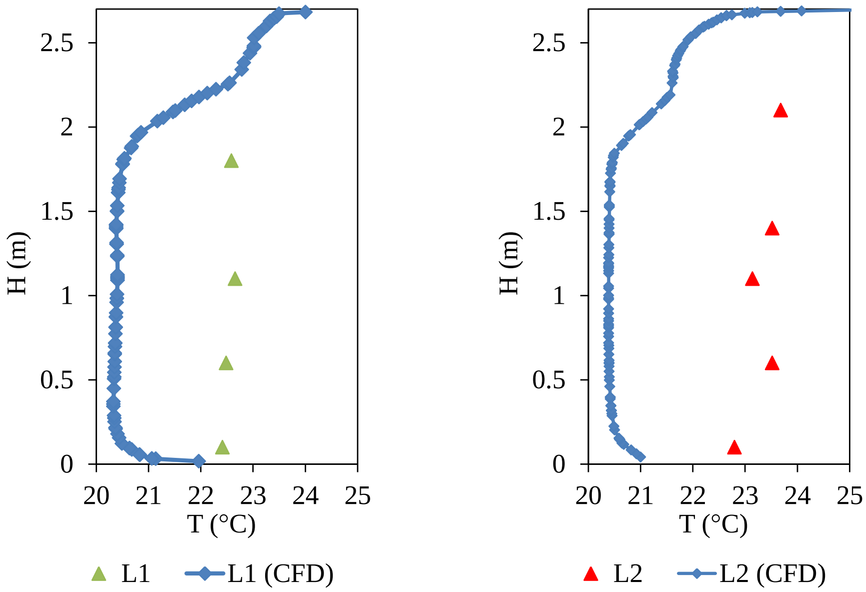

Figure 5. Vertical temperature profiles along poles L1(room) and L2(behind the manikin)

4 Conclusions

Considering the results the following conclusions can be drawn:

- The boundary condition of the average velocity used to simulate a sinusoidal breathing function predicts a more vertical direction of the exhalation flow. Therefore, the dispersion of the contaminants in the air may be completely different from the experimental measurements. This fact will produce a lower contaminant concentration at the height of the breathing of another person in the same room, facing the manikin.
- It has been shown that the lack of radiation model during the numerical simulation produces a poor prediction of the temperature gradients in the room. This fact may also influence the results of the exhalation flow. However, the exhalation velocity is more influential than the temperature distribution, especially at a reduced distance to the manikin.

More simulation work should be done in order to evaluate the influence of the radiation model and other boundary conditions used to simulate the exhalation flow.

5 References

Gao, N. and Niu, J. (2006) *Transient CFD simulation of the respiration process and inter-person exposure assessment*, Building and Environment **41**, 1214-1222.

Gao, N. and Niu, J. (2007) *Modeling particle dispersion and deposition in indoor environments*, Building and Environment **41**, 3862–3876.

He, G., Yang, X. and Srebric, J. (2005) *Removal of contaminants released from room surfaces by displacement and mixing ventilation: modeling and validation*, Indoor Air **15**, 367-380.

Morawska, L. (2006) *Droplet fate in indoor environments, or can we prevent the spread of infection?*, Indoor Air **16**, 335-347.

Mui, K.W., Wonga, L.T., Wu, C.L. and Lai, A.C.K. (2009) *Numerical modeling of exhaled droplet nuclei dispersion and mixing in indoor environments*, Journal of Hazardous Materials **167**, 736-744.

Nicas, M., Nazaroff, W.W. and Hubbard, A. (2005) *Toward understanding the risk of secondary airborne infection: Emission of respirable pathogens*, Journal of Occupational and Environmental Hygiene **2**, 143-154.

Nielsen, P.V., Li, Y., Buus, M. and Winther, F.V. (2010) *Risk of cross-infection in a hospital ward with downward ventilation*, Building and Environment **45**, 2008–2014.

Qian, H. and Li, Y. (2010) *Removal of exhaled particles by ventilation and deposition in a multibed airborne infection isolation room*, Indoor Air, **20**, 284-297.

Wan, M. P. and Chao, C. Y. H. (2007) *Transport characteristics of expiratory droplet nuclei in indoor environments with different ventilation airflow patterns*, Journal of Biomechanical Engineering **129**, 341-353.

Yin, Y., Gupta, J.K., Zhang, X., Liu, J. and Chen Q. (2011) *Distributions of respiratory contaminants from a patient with different postures and exhaling modes in a single-bed inpatient room*, Building and Environment **46** (1), 75-81.

Zhao, B., Zhang, Z. and Li, X (2005) *Numerical study of the transport of droplets or particles generated by respiratory system indoors*, Building and Environment **40**, 1032-1039.