Building and Environment 106 (2016) 340–351

Contents lists available at ScienceDirect

Building and Environment

journal homepage: www.elsevier.com/locate/buildenv

Influence of human breathing modes on airborne cross infection risk

J.M. Villafruela [a, *], I. Olmedo [b], J.F. San José [a]

[a] Department of Energy and Fluid Mechanics, University of Valladolid, 47011 Valladolid, Spain
[b] Department of Chemical Physics and Applied Thermodynamics, Cordoba University, 14014 Córdoba, Spain

ARTICLE INFO

Article history:
Received 27 April 2016
Received in revised form
8 July 2016
Accepted 9 July 2016
Available online 12 July 2016

Keywords:
Airborne cross infection risk
Human exhalation flow
CFD
Transient
Displacement ventilation
Human microenvironment

ABSTRACT

CFD simulation is an accurate and reliable method to predict the risk of airborne cross-infection in a room. This paper focuses on the validation of a 3-D transient CFD model used to predict personal exposure to airborne pathogens and infection risk in a displacement ventilated room. The model provides spatial and temporal solutions of the airflow pattern in the room (temperature, velocity and turbulence), as well as contaminant concentration in a room where two thermal manikins simulate two standing people, one of whom exhales a tracer gas N_2O simulating airborne contaminants. Numerical results are validated with experimental data and the model shows a high accuracy when predicting the transient cases studied. Once the model is validated, the CFD model is used to simulate different airborne cross-infection risk scenarios. Four different combinations of the manikins' breathing modes and four different separation distances between the two manikins are studied. The results show that exhaling through the nose or mouth disperses exhaled contaminants in a completely different way and also means that exhaled contaminants are received differently. For short separation distances between breathing sources the interaction between breaths is a key factor in the airborne cross-infection for all the breathing mode combinations studied. However, for long distances the general airflow conditions in the room prove to be more important.

1. Introduction

CFD has been used to obtain spatial and temporal solutions of the characteristics of indoor airflow patterns over the last 40 years. The first numerical results of CFD in indoor environments were based on predicting velocity fields and air distribution in rooms [1–4]. Later, analysis of different air diffusers and airflow patterns generated in ventilated spaces also became an important issue in CFD studies [5–9].

As the capacity of computers and numerical models grew, CFD simulations of indoor environments became more complex. Airflow patterns, temperature fields and the contaminant distribution of occupied rooms using different heating, cooling and ventilation systems were numerically analyzed for many years. These simulations included thermal loads, such as people and airborne contaminants (VOCs, particles or airborne pathogens) [10–18].

People were first considered as thermal loads, and later a person's breathing was also taken into account as a source of contaminants. A person infected by any virus or bacteria, such as measles, flu or tuberculosis, may exhale small droplet nuclei that are basically carried by the airflow [19,20]. These particles may be rebreathed by other people in the room causing a risk of infection. Realistic cases of airborne cross-infection situations have been studied by means of CFD using, in many cases, experimental measurements to validate models and their results. Particularly after the SARS outbreak in 2003, the need to obtain fast and reliable answers to the origin and mechanisms of airborne infection transmission linked to CFD began to take on a key role in this research field [21–25]. In recent years, CFD simulations have been used to analyze and understand the diffusion of exhaled contaminants in different indoor environments, such as operating theatres and isolation rooms [26–31], hospital wards [32,33] offices [34,35] or rail and aircraft cabins [36–38]. All this research evidences that CFD is an important tool vis-à-vis obtaining answers to the complex airborne infection route. Particular effort has been made to study the human breathing processes involved in the airborne transmission routes of diseases, such as coughing [39–41], sneezing [42,43] or breathing exhalation and inhalation [44–49].

However, transient analysis of breathing processes and the

---

[*] Corresponding author.
E-mail address: manolo@eii.uva.es (J.M. Villafruela).

http://dx.doi.org/10.1016/j.buildenv.2016.07.005

dispersion of exhaled contaminants remains limited. Human respiration processes, such as breathing, are transient and change their characteristics over a short period of time. This makes it difficult for experimental techniques to take measurements at the same pace as breathing conditions change [40,50,51]. Most CFD simulations consider the breathing process as steady, and assume a constant exhalation velocity value. This simplifies the problem studied. However, the dynamic characteristics of the breathing process are not studied and, therefore, accuracy when predicting the dispersion of exhaled contaminants may be reduced.

This paper proposes a transient study of the dispersion of exhaled contaminants and of the interaction between the respiration flows of two people in a displacement-ventilated room. In order to achieve this objective, a CFD model reproduces with a high degree of accuracy the experimental tests carried out in a full-scale test room with two breathing thermal manikins [52]. The exhalation of the source manikin (SM) contains a tracer gas simulating fine droplets (<5 μm in diameter) which may contain biological contaminants. This work focuses only on the spread of these small exhaled droplets that follow the air stream because of their small size [19]. The second manikin, considered the target (TM), is placed in front of the SM and is susceptible to inhaling the contaminants exhaled by the source and so become infected by certain pathogens. CFD simulations allow a realistic analysis of the dispersion of exhaled contaminants over time and a study of how they influence the TM's microenvironment. The amount of contaminants inhaled by the TM at different instants is also studied. The target manikin's risk of airborne cross-infection is studied considering different separation distances between the manikins (0.35 m, 0.50 m, 0.80 m and 1.10 m). The experimental results [52] are used to validate the numerical simulations.

The second objective of this paper is to analyze the same risk situation between the two manikins but modifying their breathing modes in order to predict how this parameter impacts on the dispersion of exhaled contaminants and therefore on personal exposure caused to the TM. Four combinations of exhalation modes are studied (see Table 1). The two manikins always inhale through the nose.

Each test is studied for four different separation distances between the two manikins (0.35 m, 0.50 m, 0.80 m and 1.10 m), thereby obtaining 16 different simulated situations.

2. Experimental method

The numerical results obtained with CFD are validated with results of experimental tests carried out in a test room at Aalborg University, the room measuring 4.1 m in length, 3.2 m in width and 2.7 m in height (Fig. 1). In the middle of the left-hand wall a semi-cylindrical displacement diffuser is mounted. This measures 0.6 m in height and has a 0.1 m radius, and provides a cold air supply at 16 °C with an air exchange rate of 5.6 h. Two rectangular exhaust openings measuring 0.3 m × 0.1 m each are placed in the two upper corners of the same wall as the diffuser. A radiator measuring 0.55 m × 0.40 m × 0.05 m is also placed in the middle plane of the test room. The heat load of the radiator is maintained at 300 W.

Two thermal breathing manikins are placed in the room facing each other (Fig. 1). The manikins are placed along the central plane of the room. The source manikin (SM) is considered to be an infected person and exhales contaminants simulated using a tracer gas, N_2O. This contaminant source provokes a risk situation of contagion to the susceptible person located in the same room. The target manikin (TM) simulates the susceptible person.

The SM was always 0.80 m from the radiator. Four different distances (d) between the manikins (0.35 m, 0.5 m, 0.8 m and 1.1 m) are used to study its influence on the cross-infection risk. The manikins are 1.68 m tall, average-size women. Each manikin is responsible for 94 W of heat load. The breathing functions of the manikins were performed by artificial lungs, presenting a sinusoidal airflow shape.

In the experiments, both manikins exhale through the mouth and inhale through the nose. The nostrils have an angle with the horizontal plane of about 45° and an angle with the intervening angle of 30° between the vertical planes. The mouths have a 123 mm^2 opening and a semi-ellipsoid form. The respiratory minute volume is 11.34 l/min for SM and 9.90 l/min for TM. Breathing frequency is 19.9 min^{-1} and 15.0 min^{-1} for SM and TM, respectively. This kind of manikin has been used in many previous studies. A detailed study of the similarities and differences in the breathing dynamics process between these manikins and human subjects has recently been published [53]. For more details concerning the experimental setup and data acquisition see Ref. [54].

Table 1
Exhalation modes for the two manikins. The manikins always inhale through the nose.

Test	Target manikin (TM)	Source manikin (SM)
MM	Mouth	Mouth
NM	Nose	Mouth
MN	Mouth	Nose
NN	Nose	Nose

3. Computational model

This section gives details of the numerical model used to accurately predict the cross-infection risk between two people located in a displacement ventilated room as well as all the phenomena involved, such as temperature and velocity gradients or contaminant dispersion.

3.1. Equations

The computational model solves the chemical species, continuity, momentum, energy and turbulence conservation equations given that the problem is three-dimensional, transient and non-isothermal and involves two species. The effect of radiation is included using the surface-to-surface radiation model. The RNGk-ε model that accounts for low-Reynolds-number effects in conjunction with enhanced wall treatment which combines a two-layer model with enhanced wall functions is used in these simulations. The pressure-velocity coupling was resolved using the PISO scheme. A second-order implicit transient formulation is chosen which is unconditionally stable with respect to time-step size. A second-order upwind discretization scheme is used for all equations [55].

3.2. Geometry and grid

The computational geometry replicates in detail the experiments carried out in the full-scale chamber (Fig. 1). Most of the chamber geometry is created with a hexahedron. Only near the manikins and the diffuser has a tetrahedral mesh been used. A conformal mesh joins both blocks of cells. Due to their geometry complexity and the expected high velocity and temperature gradients, mesh refinement was performed around the two manikins, the radiator, the two exhausts, and the diffuser. The mesh is significantly refined at the manikins' faces and in the exhalation zone in order to accurately simulate the interaction between the two breathing flows. Coarse mesh with hexahedral cells was considered for the rest of the room, totaling over one and half