Building and Environment 71 (2014) 1–6

Contents lists available at ScienceDirect

Building and Environment

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

Studying passive ultrafine particle dispersion in a room with a heat source

Siamak R. Ardkapan [a,*], Peter V. Nielsen [b], Alireza Afshari [a]

[a] Department of Energy and Environment, Danish Building Research Institute, Aalborg University, DK-2450 Copenhagen, Denmark
[b] Department of Civil Engineering, Aalborg University, DK-9000 Aalborg, Denmark

ARTICLE INFO

Article history:
Received 25 April 2013
Received in revised form
5 September 2013
Accepted 7 September 2013

Keywords:
Particle dynamics
Particles
Computational fluid dynamics
Turbulence models
Particle dispersion
Passive particle source

ABSTRACT

The distribution of particles in a room is of great interest because of the effect of particles on human health. Using computational fluid dynamics, it is possible to study the behaviour of particles in a room. In this study, the commercial software STAR-CCM+ was used to simulate the dispersion of passive particles in a room with displacement ventilation system. In addition, some experiments are performed to verify the accuracy of the simulation results. According to the comparison of the experiment and the simulations in front of supply opening, the $k-\varepsilon$ model seems to give better results than the $k-\omega$ model. It is shown that, in order to have an accurate result, the simulation of radiation effect is essential. Furthermore, the results of particle simulations show that when the passive particle source is in the height of 1.5 m and the heater is at the height of 0.5 m, the average concentration in the room is the lowest. Depending on the particle source height and heater location, the average particle concentration profile will change. In remote areas, the concentration profile does not show any significant difference between upper zone and lower zone.

1. Introduction

Ultrafine particles have been studied from different aspects including health effects. Because of their small size (less than 100 nm), it has been difficult to study their behaviours but they can enter deep into the lung and cause respiratory problems [1].

Air distribution in the room is divided mainly into two types, mixing ventilation and displacement ventilation. This paper focuses on displacement ventilation. The main driving force of displacement ventilation is the buoyancy force, which causes stratification in the room. The buoyancy effect in the room causes movement from down to top of a room. Studying the plumes caused by a heat source in the room has revealed the stratification height that divides the room into bottom zone and top zone. The stratification height is the location where the amount of plume flow over the heat source plus the cold down draught is equal to the supplied airflow rate.

Air movements in a room have been studied by means of microscopic and macroscopic scale methods. One of the microscopic scale methods is computational fluid dynamics (CFD) which has developed over the last decades regarding the improvements in the speed of calculation [2].

In order to test CFD for a specific problem, experimental verification is recommended by experts. Detailed information regarding selection of the governing equations and the methods of evaluating results can be found in the literature sources for a specific area of research. In the simulations of a ventilation system, information regarding boundary condition estimations, turbulence model selection and mesh quality assurance can be found in literature resources [3]. Nowadays commercial CFD software can be used to initialise the problem, solve it according to the governing equations in the fluid dynamics and visualise the results.

There are some empirical equations for predicting the airflow rate of the plume over a heat source. For the point heat source in a bounded surrounding which has temperature stratification, the equation for the airflow rate of the plume is as follows [4]:

$$q = 0.005P_c^{1/3}z^{5/3} \quad (1)$$

Where z is the height above a point heat source and P_c is the convective component of the heat source, which is calculated according to the following equation:

$$P_c = KP_t \quad (2)$$

* Corresponding author. Tel: +45 23605619; fax: +45 4586 7535.
E-mail address: sra@sbi.aau.dk (S.R. Ardkapan).

0360-1323/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.buildenv.2013.09.012

Where K is the convective heat factor and is an empirical value and P_t is the total power of heat source.

Knowing the supplied airflow rate, it is possible to find the location of the stratification height. The temperature stratification in a room has an effect on the particle concentration in the room. Studies have shown that contaminant concentration in the top zone is higher than the concentration in the bottom zone for a room with stratified flow, when the heat source is also a contaminant source. However, the concentration distribution depends on other factors such as the type of particle source; whether it is passive or thermal [5].

To compare temperature profiles of different cases, a non-dimensional temperature has been used with the following equation.

$$\theta = \frac{(T - T_S)}{(T_E - T_S)} \tag{3}$$

Where T is the temperature of an arbitrary place, T_S is the temperature of supplied air to the room and T_E is the temperature of air in exhaust.

In addition, to compare the concentration profile of pollutants in a room, a contamination ratio is used that is the contamination concentration of a location divided by the concentration of the contamination at exhaust.

The particle dispersion in the air is considered as a multiphase flow of a particle phase and air phase. The particle phase is considered as a dispersed phase in the air if the interaction of particle-fluid is the main driving force for the overall transportation of the particles. The simulation of a dispersed flow is performed using some specific models such as Eulerian and Lagrangian approaches. In the Lagrangian approach, the particle variables are calculated along the trajectory of individual particles. For large numbers of particles, it is practical to determine parcels where they are representative of a group of particles.

In the Eulerian approach, the properties are calculated for a computational volume of the flow [6]. The two approaches have pros and cons. The Lagrangian approach is more physically robust if, the size distribution, particle-wall interaction and diffusion of particles are of great importance [7]. On the other hand, the Lagrangian approach needs more memory and time than the Eulerian approach.

In the Lagrangian approach, the properties of a particle are calculated using simple equations of classical mechanics. The equations of motions for the particle phase are expressed as below.

$$\ddot{x} = \frac{f_b + f_s}{m} + g \tag{4}$$

Here x is the position of the particle, f_s is the representative of surface forces, f_b is the fluid force exerts on the particle i.e. body forces, m is the mass of the particle and g is the gravity acceleration vector. f_b includes drag forces, pressure gradient force and virtual mass force. $\ddot{x}$ is the acceleration of the particle.

The forces exerted on the particles include drag force, gravitational force, pressure gradient force and virtual mass force. In the indoor air conditions, the effects of the virtual mass force can be neglected as the virtual mass force depends on the ratio of the density of the air to the density of the particle which is about10^{-3} [8].

For sub-micro particles, in the boundary layers of the walls of a room, the importance of the diffusion is higher than thermophoresis. Since the thermophoresis depends on the size of the particles, in addition to the temperature difference between wall and the air [9,10]. It has been shown that for the Nano-size particles, thermophoresis force is very small and consequently it has very small effect on the particle distribution [11].

The aim of this study is to evaluate particle distribution in a room with displacement flow pattern for the case where the particle source is a candle which is usually applied in the ultrafine particle experiments. The aim also is to see the impact of the location of heat source on the distribution of particles in different heights. The focus of the paper is on ultrafine particles.

2. Methods

In this study, a room was selected for simulation by a commercial CFD software, STAR-CCM+. The real room dimensions were 5 m × 2 m × 2.9 m and the inlet was a slot inlet with the dimensions of 4 cm × 53 cm and aspect ratio of 13. The inlet velocity was set 0.3 m/s. The air movement from the inlet of the room was measured and visualised by smoke in the room to be used later for the assessment of the results of the turbulence models. Fig. 1 shows the locations of inlet, outlet, heater, particle source

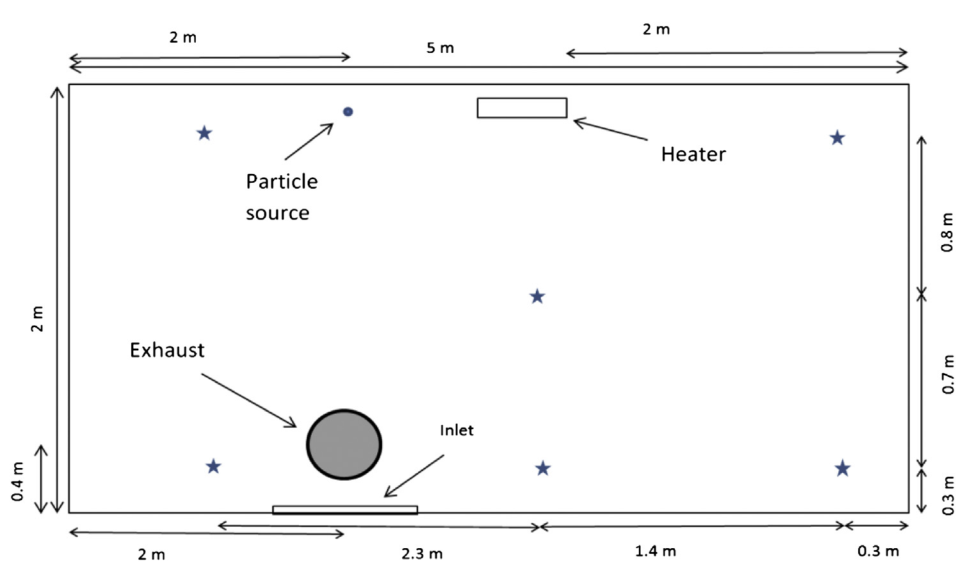

Fig. 1. Outline of the room including heater location, particle source and sampling points (stars) in the CFD simulation.