Performance of Supply Airflow Entrainment for Particles in an Underfloor Air Distribution System

Can Li[1,2]
Doctoral Candidate
Associate Professor

Nianping Li[1]
Professor

1College of Civil Engineering, Hunan University, Changsha P.R.China, 410082
2 Hunan University of Technology, Zhuzhou P.R.China, 412000
Lc19992@126.com

Abstract: Investigation and engineering projects indicate that the underfloor air distribution (UFAD) system is superior to conventional HVAC systems in some aspects. For example, the former is more adaptive in rearranging the office space to improve indoor comfort conditions and energy conservation. However, the supply air outlet of UFAD system is set on the floor, such that the supply airflow may entrain the dust particles settled on the floor or suspended near the floor. This creates problems that need to be studied, because airflow can bring theses particle to the human breathing zone.

In this paper, the performance of the airflow field in a UFAD system and the characteristics of supply air entrainment for particles are studied. A physical and numerical model is established. Comparison of the numerical calculation results of the airflow field with the measured data is given. Results show that the model is valid and can predict the performance of airflow entrainment for dust particles in different air supply conditions.

Key words: under floor air distribution system, particle, entrainment, simulation

1. INREODCCTION

In many modern office buildings, raised floor were build to create floor voids for flexible and convenient cable management. with the installation of the raised floor, under floor air distribution (UFAD) system, which make use of the floor void to distribute conditioned air, has been gaining concern among the industry and the number of applications have been growing since it was first introduced in the 1950s. With UFAD system, the air diffuser convenient be arranged or operated. Besides this, many researchers show that well-designed UFAD systems can provide such benefits as:

- performed better than the traditional ceiling-based systems both in the energy aspect and in the indoor air quality aspect[1];
- reduced life cycle building coats[2];
- improved thermal comfort[3].

However, the UFAD systems conduct different air distribution pattern from traditional over-head air conditioned systems. For UFAD systems, the particle deposited under floor or suspended on the air near floor may be entrained by air flow then be carried into the human's inhaled zone have to be considered.

Experiments are usually conducted in model ventilation room to measurement airflow filed and concentration of particle due to these parameter in real room were complicated and influenced by many factors [4,5]. Chao et al. measured airflow and air temperature distribution in the occupied region to an under floor ventilation system [6]. Lu et al. studied the model and measurement of airflow and aerosol particle distribution in a ventilated chamber which is divided into two zones by a partition with a large opening [7].

With the application of computational Fluid Dynamics (CFD) in building environment has undergone extensive development in recent years, many reports have been found in which air flows or aerosol particle suspend in ventilated room are simulated .Shimada et al [8] and Cheong et al [9] have employed the commercial code FLUENT investigate the transport and dispersion of contaminant particle in a ventilated room or turbulent flow behavior. They have both conducted satisfactory agreement results with the experimental data.

In this paper, the re-normalization group(RNG) k- ε turbulence model of FLUENT is employed simulate the airflow velocity profiles in a UFAD system model room. The numerical calculate results are compared with measurements. And the particle transport behavior with the air movement is investigated by employing Particle Discrete Model (DPM) simulating.

2. COMPUTAL MODLE

2.1 Air Flow Model

In this study, just isothermal air flow is investigated. The air is regarded as a continuous fluid, so the air movement in the room is governed by the Eulerian conservation equation

$$\frac{\partial}{\partial t}(\rho\phi)+div(\rho v\phi)-div[\Gamma_{\phi} grad\phi]=S_{\phi} \qquad (1)$$

Continuity equation (ϕ=1)

$$\Gamma_{\phi}=0, S_{\phi}=0 \qquad (2)$$

Momentum equation ($\phi=u_i$ u_i represents each of the three velocity components u v w)

$$\Gamma_{\phi}=\mu_{eff}, S_{\phi}=-\frac{\partial p}{\partial x_j} \qquad (3)$$

Turbulent kinetic energy equation ($\phi=k$)

$$\Gamma_{\phi}=\mu_{eff}/\sigma_k, S_{\phi}=P_k-\rho\varepsilon \qquad (4)$$

Dissipation equation ($\phi=\varepsilon$)

$$\Gamma_{\phi}=\mu_{eff}/\sigma_k, S_{\phi}=\frac{\varepsilon}{k}(C_{\varepsilon_1}P_k-C_{\varepsilon_2}\rho\varepsilon)-R \qquad (5)$$

In RNG k- ε model equations, μ_{eff} denotes effective viscosity that is calculated by

$$d\left(\frac{\rho^2 k}{\sqrt{\varepsilon\mu}}\right)=1.72\frac{\hat{\upsilon}}{\sqrt{\hat{\upsilon}^3-1+C_{\upsilon}}}d\hat{\upsilon}$$

Where $\hat{\upsilon}=\mu_{eff}/\mu$ and $C_{\upsilon}=100$, μ denotes gas phase turbulent viscosity, defined as $\mu=\rho C_{\mu}(k^2/\varepsilon)$. P_k is the generation of turbulence kinetic energy due to the mean velocity gradient defined by $P_k=\mu_j S_{ij}$.

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_i}{\partial x_i}\right)$$

is the rate-of-strain tensor. More details of turbulence parameters of RNG k- ε model can be found in Ref.[10].

The boundary conditions which are specified for gas phase in this study are defined as follows:

- the inlet is defined as an opening with a uniform velocity;
- Neumann boundary conditions are applied at the outlet .That is, mass flow boundaries are specified to ensure the mass flow rate out of the domain the same as the mass flow rate into the flow domain;
- A non-slip condition at the solid wall is applied for velocities. “Wall functions” are applied to describe the turbulent flow properties in the near wall region.

The equations are discretized into algebraic equations by the Finite volume method (FVM). The discretization method is a second-order upwind scheme and SIMPLE algorithm is adopted. The Boussinesq model is employed to consider the buoyancy effect.

2.2 Particle Phase Model

In FLUENT, a discrete trajectory approach (Lagrangian method) is employed in the Discrete Phase Model (DPM). The Lagrangian approach splits the particle phase into a representative set of individual particles and tracks these particles separately through the flow domain by solving the equations of particle movement. The following assumptions are used:

•all particles are inert particles in spherical solid shape;

• heat and mass transfer between air and particles are neglected;

• no particles coagulation in the particle deposition process;

• all particles on solid surface are entrapped by them, such as walls, floors and ceiling.

This force balance equates the particle inertia with the forces acting on the particle, and can be written (for the x direction in Cartesian coordinates) as

$$\frac{du_{pi}}{dt} = F_D(u_i - u_{pi}) + g_i(\rho_p - \rho)/\rho_p + F_i \quad (6)$$

where $F_D(u_i - u_{pi})$ is the drag force per unit particle mass, and F_D is given by

$$F_D = \frac{18\mu}{\rho_p D_p^2} \frac{C_D \mathrm{Re}}{24} \quad (7)$$

u_p is the particle velocity, ρ_p is the density of the particle.

The drag coefficient, C_D, is evaluated from experimental-fitted expression of Morsi and Alexxander.

$$C_D = a_1 + \frac{a_2}{R_e} + \frac{a_3}{{R_e}^2} \quad (8)$$

Where a_1 a_2 a_3 are constants that apply to smooth spherical particles over several ranges of Re. The second term on the right-hand side of equation (6) is the gravitational force per unit particle mass. And the third term, F_i, represents the additional acceleration force exerted on particles depending on the flow condition and particle properties. These forces include pressure gradient force, Basset force and virtual mass force caused by unsteady flow, Brownian force, lift force due to shear (Saffman's lift force),and Thermophoretic Force caused by temperature gradient. The particle properties (such as particle size and density) and the air flow field influence the magnitude of these forces. Considering the particle size and density in indoor environments, some of the forces are small enough to be negligible [11].Ref. [11] analyzed each of the force then found the Brownian force and Saffman's lift force may be relatively large for the fine particles and flow field to be studied. Therefore, only the Brownian force and Saffman's lift force are taken into account for this study since they may play an important role in sub-micron particles' motion near the walls, where the velocity gradient is large enough to make Saffman's lift force dominant. The Saffman's lift force is from Li and Ahmadi and is a generalization of the expression provided by Saffman [12]:

$$F_{si} = \frac{2K\nu^{0.5}\rho d_{ij}}{\rho_p D_p (d_{lk} d_{kl})^{0.25}} (u_i - u_{ip}) \quad (9)$$

Where K = 2.594 and d_{ij} is the deformation tensor.

The initial conditions for particle tracking include the starting positions and initial velocities of particles which have the density of the oil smoke particle, i.e.

$$\rho_p = 865.0 kg/m^3$$

The starting positions of particle is assumed in the underground and uniformly distributed in that area with 0 initial velocities.

3. EXPERIMENT PROCEDUCES

Experimental work is carried out in a UFAD model room. The room size used in the experiment is L(X)×W(Y)×H(Z)=3 m×3 m×2.64 m., The height of floor plenum is 180mm. Air is entrained the floor plenum from three rectangular air duct and is supplied to room through a circular opening（φ100）on the floor. In the ceiling of room, 4 rectangular air return grill are arranged regularly, which size is L(X)

×W(Y) =0.2m×0.2m. As be seen in Fig.1. The positions of the velocity measurement points are as

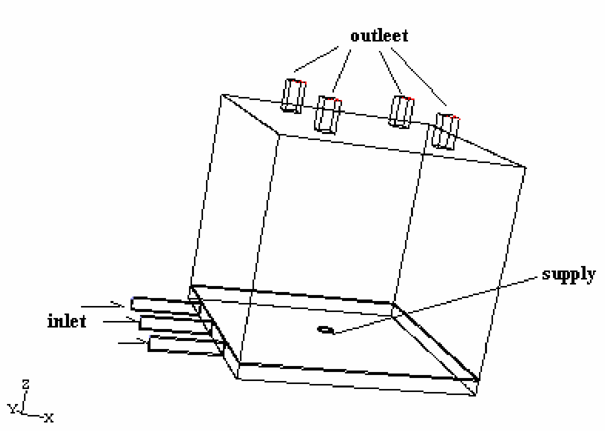
$$\begin{cases} x = -1000, -500, 0, 500, 1000 \\ y = -1000, -500, 0, 500, 1000 \\ z = 1100 \end{cases}$$

and

$$\begin{cases} x = 0 \\ y = 0 \\ z = 100, 500, 1100, 1500, 1800 \end{cases}$$

The origin of coordinates is set in the center of the floor diffuser, meantime, the central axis of the floor air diffuser is the central axis of the room.

Experiment performances in two conditions as inlet air velocity is 1.1m/s and 1.23m/s respectively. Velocity of each measured spots is measured in both inlet velocity conditions. In fact, the value of the velocity is turbulent, it always fluctuate within in a range. The range is varied for different spot. So, to conveniently compare with numerical simulations, which are fixed, the average value of measured data within 30 seconds is extracted when processing data.



• **Fig.1 The structure of model room**

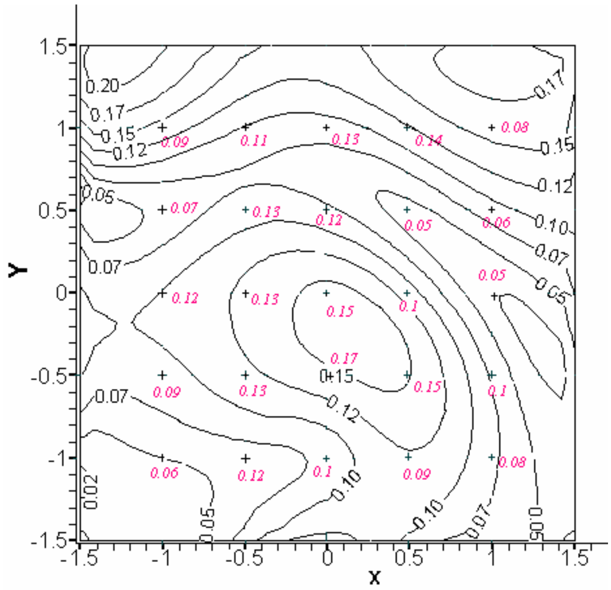
4. RESULTS AND ANALYSIS

4.1Gas Phase Calculation

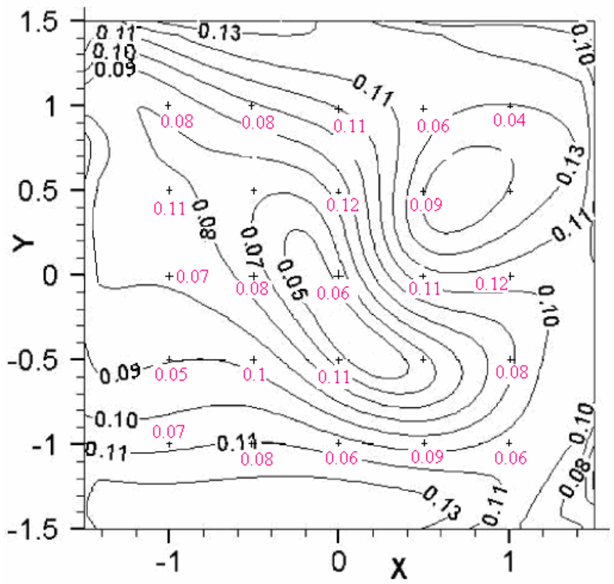
The room considered here is divided into 150×150×66 cells (i.e. control volumes). The inlet section is divided into 3× 4×30 cells, while 10× 10×10cells are set up for the exhaust section

To compare numerical calculation with experimental data, the similar conditions are employed in the calculation. That's, the main flow is defined as entering the inlet with a uniform velocity, i.e. (a) $u_x = 1.1 m/s, u_y = u_z = 0$ (for ACH 8h^{-1}), and (b) $u_x = 1.23 m/s, u_y = u_z = 0$ (for ACH 9h^{-1}).

The quantities k and E at the inlet are normally based on the mean flow characteristics at the inlet.



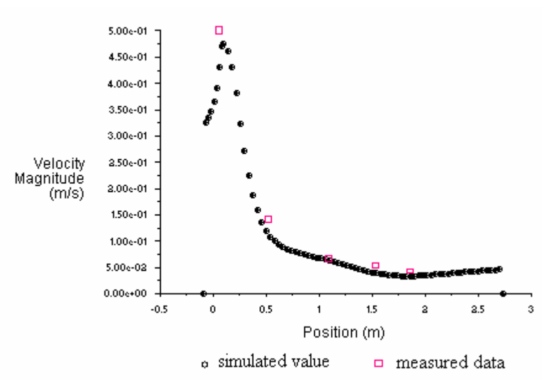
(a) Inlet velocity is 1.1m/s



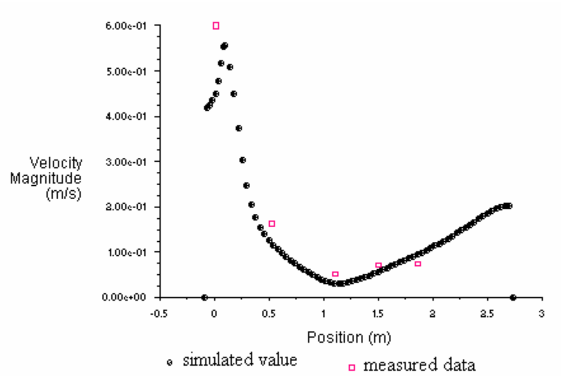
(b) Inlet velocity is 1.23m/s

• **Fig.2 Air velocity contour as Z=1.1m**

Fig2 (a) and (b) show the calculated velocity contour in these two conditions as Z=1.1m. The corresponding measured values of velocity of each test point are marked in the figure yet. Comparisons of air velocity along the central axis of measured data with simulated results in two conditions are showed in Fig.3 (a) and (b). From fig.2, it can be found that in both inlet conditions reasonable and acceptable of air velocity field room horizon plan is been simulated. Especially the calculated velocity values for the points some distant away from walls is good agreement with the measured values. However, the simulated results for less point which set on the wall vicinity is disagreement with the measured data. The main reason may be the model can not treat the room wall perfect that can be seen in Ref.[8]. Fig.3 indicates that along the central axis of room the simulated calculation results of test points are good agreement with the measured data in both inlet velocity conditions, although the calculated velocity of the point near by the air supply(X=Y=Z=0) is smaller than the measured data.



(a) Inlet velocity is 1.1m/s



- **Fig.3 Comparisons of air velocity along the central axis of measured data with simulated results**

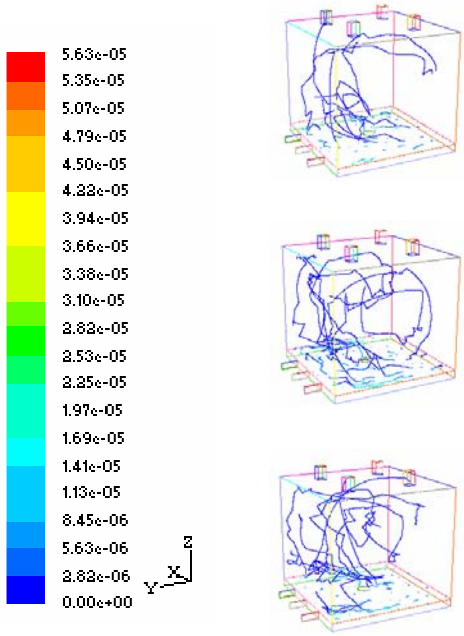
(b) Inlet velocity is 1.23m/s

4.2 Case of Particle Phase Calculation

Many of simulations of contaminant particle dispersion and distribution within room using the Lagrangin particle-tracking model have reported. The similitude method is employed in this paper, which has been gained result good agreement with measured data [11] [13]. Based on the right airflow field calculated result, cases of movement of the particle are simulated in this study. Consider the particle which size under 10μm is interest for indoor air quality since they are known as inhaled particle, the particles of the case are divided into three groups with 1,5 and 10μm diameter. To compare the particle movement and distribution, particles are assumed to be uniformly carried by each size group. The particle mass carried by each sample group is assumed to be 2250μg. The particles are assumed to initially be on the floor, therefore the initial position of particles are set up in the whole surface on the floor.

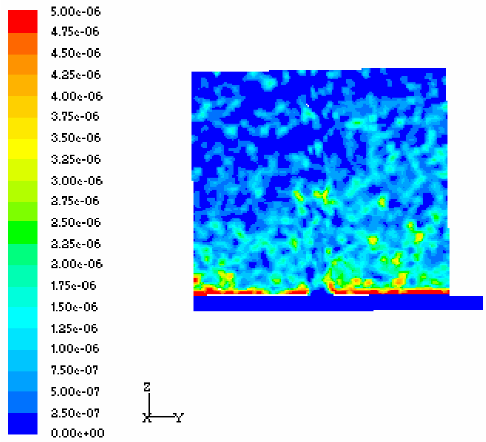
'Trap' or 'escape' type boundary conditions are set for wall boundary conditions of the particle transport model, i.e. there is no particle rebound when it reaches the wall.

Fig.4 gives the comparison of the tracks between three group particles (PM1/PM5/PM10) when inlet air velocity is 1.23m/s. The calculation results report and Fig.4 illustrate that the bigger particles (PM10) spend few time suspended in ventilation space, while the smaller particles (PM1/PM5) spend few time suspended in there. At the other inlet velocity, the similar result is conduct. This means the smaller particles contributed more to the indoor pollutant concentration.

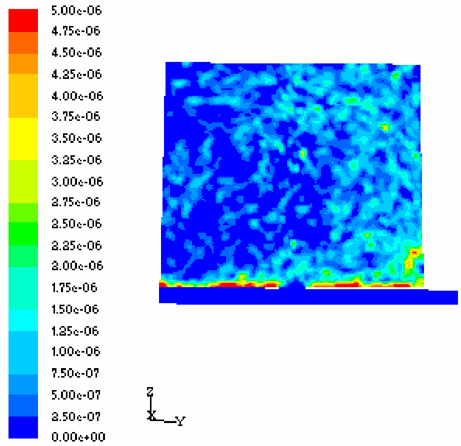


Colored by DPM concentraction(kg/m^3)

• **Fig.4. Tracks of particles (the top one is trace of the particle of PM10, the middle one is tracks of the particle of PM5, the bottom one is tracks of the particle of PM1)**



(a) **Inlet velocity is 1.1m/s**



(b) **Inlet velocity is 1.23m/s**

• **Fig.5. Contours of DPM concentration of surface x=0**

The comparison of contours of DPM concentration of surface x=0 between different inlet velocity condition can be seen in Fig.5. It can be seen that the DPM concentration of the occupied region decrease with the ventilation rate growth in current study.

5. CONCLUTIONS

A CFD analysis of airflow patterns and aerosol particle movement performance in a UFAD system is been presented in this paper. Comparisons of air flow velocity field of computations with experiments have also been given. The calculated velocities of almost testing point are accordance with the measured data. The following conclusions emerge.

• the model of UFAD system presented in this study is valid to both experimental ventilation conditions and the boundary conditions being set is proper.

• the model of DPM model employed in current study can predict the performance of the particle entrained with the airflow

• The ventilation rate influences the particle moving performance in current study, the concentration of particle decrease with the growth of ventilation rates

• the smaller particles spend more time suspended in ventilated spaces and so have significant contribution to the pollutant concentration of indoor air

• Further work is required in this area, e.g. the study of airflow distribution in room with non-thermal jet in UFAD SYSTEM, and the particle movement under buoyancy influence.

ACKNOWLEDGMENTS

The financial supported provided by the Specialized Research Fund for the Doctoral Program of High Education (SRFDP) ID20050532022 is grateful acknowledged.

REFERENCES

[1] K.J. Loudermilk, Underfloor air distribution solutions for open office applications, ASHRAE Transactions [J] 105 (1999) 605–613.

[2] F. Bauman, T. Webster, Outlook for underfloor air distribution, ASHRAE [J] 43 (6) (2001) 18–27.

[3] T. Webster, F. Bauman, J. Reese, Underfloor air distribution: thermal stratification, ASHRAE [J] 44 (5) (2002) 28–36.

[4] Thatcher TL, Lai Alvin CK, Moreno-Jackson R, Sextro RG, Nazaroff WW. Effects of room furnishings and air speed on particle deposition rates indoors. Atmospheric Environment [J] 2002; 36: 1811–9.

[5] Koponen IK, Asmi A, Keronen P, Puhto K, Kulmala M. Indoor air measurement campaign in Helsinki, Finland 1999—the effect of outdoor air pollution on indoor air. Atmospheric Environment [J].2001;35: 1465–77.

[6] C.Y.H. Chao, M.P. Wan. Airflow and air temperature distribution in the occupied region of an under!oor ventilation system. Building and Environment[J] 2004 ;39: 749 – 762

[7] Weizhen LU, Howarth AT, Adam N, Riffat S. Modeling and measurement of airflow and aerosol particle distribution in a ventilated two-zone chamber. Building and Environment [J] 1996;31:417–23.

[8] Shimada M,Okuyama K,Okazaki S,Asai T,Matsukura M, Ishizu Y. Numerical simulation and experiment on the transport of fine particles in a ventilated room. Aerosol Science and Technology [J] 1996;25:242–55.

[9] Cheong KWD,Djunaed y E,Poh TK,Tham KW, Sekhar SC, Wong NH,Ulah MB. Measurements and computations of contaminant's distribution in an office environment. Building and Environment [J] 2003;38:135–45.

[10] FLUENT user's guide. Version 6.2 FLUENT incorporated, Lebanon,NH, 2003.

[11] Bin Zhao; Ying Zhang, Xianting Li et al. Comparison of indoor aerosol particle concentration and deposition in different ventilated rooms by numerical method. Building and Environment [J] 2004;39 1 – 8.

[12] Saffman PG. The lift on a small sphere in a slow shear flow. Journal of Fluid Mechanics [J]1965; 22:385–400.

[13] Z.F. TIAN, J.Y. TU,G.H. YEOH, R.K.K. YUEN. On the numerical study of contaminant particle concentration in indoor airflow. Building and Environment [J]; 2005;40 321 – 330.