Building and Environment 85 (2015) 144–152

Contents lists available at ScienceDirect

Building and Environment

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

Simulating buoyancy-driven airflow in buildings by coarse-grid fast fluid dynamics

Mingang Jin [a], Wei Liu [a, b], Qingyan Chen [b, a, *]

[a] School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA
[b] Tianjin Key Lab of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China

ARTICLE INFO

Article history:
Received 10 April 2014
Received in revised form
6 November 2014
Accepted 13 November 2014
Available online 4 December 2014

Keywords:
Heat sources
Plume model
Computational fluid dynamics
Experimental validation
Analytical model

ABSTRACT

Fast fluid dynamics (FFD) is an intermediate model between multi-zone models and computational fluid dynamics (CFD) models for indoor airflow simulations. The use of coarse grids is preferred with FFD in order to increase computing speed. However, by using a very large mesh cell to represent a heat source that could have a much smaller physical size than the cell, coarse-grid FFD would under-predict the thermal plume and thermal stratification. This investigation integrated a thermal plume model into coarse-grid FFD. The integration first used the plume model to calculate a source for the momentum equations and then corrected the temperature at the plume cell. The integration enabled coarse-grid FFD to correctly predict the plumes. When applied to displacement ventilation, coarse-grid FFD with the plume model can accurately predict the mean air temperature stratification in rooms as compared with experimental data from the literature. The improved model has also been used to calculate the ventilation rate for buoyancy-driven natural ventilation. The calculated ventilation rates agree well with the experimental data or predictions by CFD and analytical models. Coarse-grid FFD with the plume model used only a small fraction of the computing time required by fine-grid FFD, while the associated errors for the two grid sizes were comparable.

1. Introduction

Whole-building airflow simulations are required in applications such as natural ventilation design, coupled building airflow and energy simulation, smoke control, and air quality diagnosis in a building. These simulations generally use multi-zone models [1]. However, the models can provide only very limited airflow information because of the assumption that a room within a building can be treated as a single homogeneous node. Computational Fluid Dynamics (CFD) models, on the other hand, can perform detailed airflow simulations, but the use of CFD for whole-building airflow simulations is too computationally expensive [2]. Between CFD models and multi-zone models, researchers have also developed intermediate models for whole-building airflow simulations. Zonal models [3] are typical intermediate models that can achieve a balance between reduced computing costs and the level of detail required in airflow simulations. Additionally, by using very coarse grids, coarse-grid CFD models [4] can provide more detailed airflow simulations at a competitive computing speed with respect to zonal models, and they are expected to replace zonal models in the future [2]. Fast Fluid dynamics (FFD), a recently developed intermediate model that can provide reliable simulations of indoor airflows at a speed that is about 15 times faster than CFD models, currently has great potential for performing whole-building airflow simulations [5,6]. Because FFD is also a grid-based model, reducing the grid number can further enhance the computing speed of FFD simulations. Coarse-grid FFD would be an ideal tool for performing whole-building airflow simulations at a greatly reduced computing cost.

Although a coarse grid could significantly reduce the computing time of FFD simulations, it may cause problems in the representation of the boundary conditions encountered in building airflow simulations. For example, many heat sources in buildings are of small physical size, such as computers, desk lamps, occupants, etc. Using a very large mesh cell to represent a small-sized heat source would result in the prediction of lower energy intensity in the cell and smaller buoyancy forces from the heat source. Coarse-grid FFD

* Corresponding author. Tianjin Key Lab of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin 30072, China. Tel.: +1 (765) 496 7562; fax: +1 (765) 494 0539.
E-mail addresses: jin56@purdue.edu (M. Jin), yanchen@purdue.edu (Q. Chen).

http://dx.doi.org/10.1016/j.buildenv.2014.11.028
0360-1323/

would thus tend to under-predict the plume flow generated by the heat source and would not accurately predict buoyancy-driven ventilation and room air temperature distribution. Because buoyancy-driven ventilation is a major feature of high-performance building systems, such as displacement ventilation and buoyancy-driven natural ventilation systems, correct prediction of buoyancy-driven ventilation and room air temperature distribution is essential with coarse-grid FFD.

Thus it is necessary to improve the representation of small heat sources with large cells. In CFD models, simulations are normally performed on fine grids, which allows CFD models to avoid the aforementioned problem. Instead, to reduce the complexities of representing heat sources in simulations, it is still necessary for CFD models to apply simple heat source geometries or use replacement boundary conditions [7]. As intermediate models, zonal models also have the same problem of representing heat sources as coarse-grid FFD does. Thus the approaches applied in zonal models to model thermal plumes could also be a potential solution for coarse-grid FFD.

Extensive research has been conducted into the characteristics of thermal plumes [8]. Morton et al. [9] proposed a theoretical model to describe the physics of thermal plumes, and this model has been adapted for studying a wide variety of thermal plumes. Kofoed [10] experimentally studied thermal plumes generated by indoor heat sources in ventilated rooms and proposed a model coefficient to account for the influence of enclosing walls. Trzeciakiewicz [11] experimentally investigated the characteristics of thermal plumes in response to objects of varying shape, such as computers, desk lamps, and light bulbs. The investigation revealed that the experimentally determined model of a plume above a point heat source could be used to characterize the thermal plumes in displacement ventilation. Zukowska et al. [12] investigated the characteristics of the thermal plume generated by a sitting person using four different geometries and found that a rectangular box could correctly simulate the enthalpy flux and buoyancy flux generated by the person. Craven and Settles [13] performed a computational and experimental investigation to characterize the thermal plume from a person and concluded that the room temperature stratification had a significant effect on plume behaviour. The aforementioned research into the characteristics of indoor thermal plumes has provided a wealth of information.

On the basis of these studies of thermal plume physics and analytical plume models, simple models have been developed to quantify ventilation and temperature distributions in buildings [14,15]. In addition, plume models have been used to improve the performance of other models for simulating buoyancy-driven airflows in buildings. Inard et al. [16] integrated a wall thermal plume model into a zonal model to improve the latter's performance in simulating the temperature distribution in a room. Musy et al. [17] integrated a plume model with a zonal model to obtain a better simulation of natural convection in a room with a radiative-convective heater. Stewart and Ren [18] used a plume model to improve the simulation accuracy of a zonal model for studying the airflow rising from a cooking plate. It has been shown that plume models effectively enhance the performance of zonal models in simulating buoyancy-driven airflows in buildings.

The integration of plume models with zonal models suggests that plume models could also be integrated with FFD for improving the performance of coarse-grid FFD simulations for room airflows driven by heat sources. This study therefore developed a method of implementing a plume model in FFD when the mesh cell is much larger than the heat source. The proposed integration method was also tested and evaluated.

2. Research method

2.1. Fast fluid dynamics

FFD simulates an airflow by numerically solving a set of partial differential equations representing the transport phenomena in the airflow, Eqs. (1)–(3), which are derived on the basis of the conservation of mass, momentum (Navier–Stokes equations), and scalar transport quantities (such as energy and species), respectively.

$$\frac{\partial U_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial U_i}{\partial t} + U_j\frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu\frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{1}{\rho}F_i, \tag{2}$$

$$\frac{\partial \phi}{\partial t} + U_j\frac{\partial \phi}{\partial x_j} = \Gamma\frac{\partial^2 \phi}{\partial x_j \partial x_j} + S, \tag{3}$$

where i or $j = 1, 2, 3$; U_i is the ith component of the velocity vector, x_i the ith direction of coordinate, t time, p pressure, ρ density, ν the kinetic viscosity, F_i the ith component of the body forces, ϕ the scalar variables, Γ the transport coefficient for ϕ, and S the source term. In each time step, FFD solves this set of transport equations sequentially. To enhance computational efficiency, a time-splitting scheme [19] was applied to solve the transport equations. For example, FFD splits the scalar transport Eq. (3) into an advection Eq. (4) and a diffusion Eq. (5),

$$\frac{\phi^{(1)} - \phi^n}{\Delta t} = -U_j\frac{\partial \phi^n}{\partial x_j}, \tag{4}$$

$$\frac{\phi^{n+1} - \phi^{(1)}}{\Delta t} = \Gamma\frac{\partial^2 \phi^{n+1}}{\partial x_j \partial x_j} + S, \tag{5}$$

where ϕ^n and ϕ^{n+1} represent the variable at the current and next time steps, respectively, and $\phi^{(1)}$ represents the intermediate variables solved by the advection equation. The advection Eq. (4) is first solved with the conservative semi-Lagrangian scheme [20] to obtain the intermediate value $\phi^{(1)}$, and then FFD is implicitly solved using the diffusion Eq. (5) to update the scalar distribution at the next time step. To effectively resolve the coupling between the momentum equations and the continuity equation, a pressure projection [21] is performed force the velocity field to satisfy continuity.

2.2. Integration with plume model

As a result of natural convection, air surrounded by a heat source in a room can form a buoyancy plume. As the plume rises, it induces the surround air into the plume flow, and flow carries the heat from the heat source to the upper part of the room. To describe the features of buoyancy-driven airflows, the plume flow rate (V_p) and excess temperature (ΔT_p) in the plume region are usually the two most important parameters [22], and they vary with the heat generation rate, heat source geometry, heat source location, etc. When the mesh cell size used in coarse-grid FFD is much larger than the physical size of a heat source, the two parameters predicted by FFD may not be accurate. In order to improve the performance of coarse-grid FFD in predicting the thermal plume, this study used an analytical plume model with empirical coefficients to calculate the plume flow rate and the excess temperature in the