Journal of Southeast University (English Edition) Vol. 26, No. 2, pp. 283 - 286 June 2010 ISSN 1003—7985

Application of jet main region specification model in CFD simulation for room air movement analysis

Wang Zhigang[1,2] Zhang Yufeng[1] Sun Yuexia[3]

([1] School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China)
([2] Tianjin Municipal Engineering Design and Research Institute, Tianjin 300051, China)
([3] Texas Institute of Allergy, Indoor Environment and Energy, University of Texas at Tyler, Tyler 75799, TX, USA)

Abstract: This paper analyzes the applications of four air terminal device (ATD) models (i. e., the basic model, the box model, the *N*-point momentum model, the jet main region specification model) in computational fluid dynamics (CFD) simulation and their performance in case study. A full-scale experiment is performed in an environment chamber, and the measured air velocity and temperature fields are compared with the simulation results by using four ATD models. The velocity and temperature fields are measured by an omni-directional thermo-anemometer system. It demonstrates that the basic model and the box model are not applicable to complicated air terminal devices. At the occupant area, the relative errors between simulated and measured air velocities are less than 20% based on the *N*-point momentum model and the jet main region specification model. Around the ATD zone, the relative error between the numerical and measured air velocity based on the jet main region specification model is less than 15%. The jet main region specification model is proved to be an applicable approach and a more accurate way to study the airflow pattern around the ATD with complicated geometry.

Key words: computational fluid dynamics; air terminal device model; jet main region specification model; indoor air movement

During the last two decades there has been great interest in developing CFD programs for predicting the air movement in a ventilated room[1-3]. In a ventilated room, the supply air condition and the type of diffusers are two essential parameters affecting the air flow pattern in the room. The conventional method considers the supply diffuser as a simple free opening. However, this method cannot handle the diffuser whose geometry is complicated. Nielsen[4] proposed a box model (a box located around the diffuser), in which the description of the diffuser boundary conditions was transferred to the description of the box boundary conditions. The box model requires measurements of parts of the boundary conditions. Therefore, this method is not very practical. Chen and Moser[5] proposed a new momentum method in which the velocity vector was calculated based on the effective area, not on the opening area, in order to obtain correct velocity descriptions. But most of the commercial CFD software does not support the separate description of boundary conditions for continuity and momentum equations. This paper introduces the jet main region specification model, and puts forward equations to calculate the boundary conditions around air terminal devices (ATD) in the model. The performances of the basic model, the box model, the *N*-point momentum model and the jet main region specification model are compared.

Received 2009-11-20.
Biographies: Wang Zhigang (1978—), male, graduate; Zhang Yufeng (corresponding author), male, doctor, professor, yufengfa@tju.edu.cn.
Citation: Wang Zhigang, Zhang Yufeng, Sun Yuexia. Application of jet main region specification model in CFD simulation for room air movement analysis[J]. Journal of Southeast University (English Edition), 2010, 26 (2): 283 - 286.

1 Methodology

1.1 Jet main region specification model

In the jet main region specification model, the ATD boundary conditions are transferred to the specification of a rectangular block in front of the device. The boundaries along the airflow are set at the jet main region and their conditions are calculated by characteristic equations, while the parameters perpendicular to the airflow are determined by $\partial\phi/\partial y = 0$[6]. For the non-isothermal free jet, which is popular in engineering applications, the characteristic equations to calculate air velocity and temperature are shown as

$$\frac{u_x}{u_0} = \frac{m_1\sqrt{F_0}}{x} \tag{1}$$

$$\frac{\Delta T_x}{\Delta T_0} = 0.73\frac{u_x}{u_0} = \frac{n_1\sqrt{F_0}}{x} \tag{2}$$

$$\Delta T_x = T_x - T_0, \quad \Delta T_0 = T_0 - T_n$$

where u_x is the jet axial velocity at a distance of x to the discharge, m/s; u_0 is the average velocity at discharge, m/s; x is the distance to the discharge flow, m; F_0 is the area of discharge, m^2; T_x is the jet axial temperature at a distance of x to discharge, K; T_n is the temperature of the surrounding air, K; T_0 is the average temperature at discharge, K; m_1 and n_1 are velocity and temperature decay coefficients, respectively, which are determined by air opening patterns.

The *Y*-axis deviation from the horizontal axis of the jet distance can be expressed as

$$\frac{y}{d_0} = \frac{x}{d_0}\tan\beta + A_r\left(\frac{x}{d_0\cos\beta}\right)^2\left(0.51\frac{ax}{d_0\cos\beta} + 0.35\right) \tag{3}$$

$$A_r = \frac{gd_0(T_0 - T_n)}{T_n u_0^2}$$

where y is the vertical distance of the curving trail of the jet flow at x, m; d_0 is the diameter of air opening, m; β is the angle between horizontal and jet flows, (°); a is the turbulence coefficient, which is determined by air opening patterns; A_r is the Archimedes number; g is the acceleration of gravity, m/s^2.

1.2 Set up of air terminal device model

Software Airpak 2.1 is used to set up the four air terminal device models, i. e., the basic model, the box model, the N-point momentum model and the jet main region specification model. These models are used in the air movement simulations in an office, respectively. The office dimensions are 5.4 m × 4.7 m × 2.6 m. The layout of the office is shown in Fig. 1.

Supply air terminal device with a height of 1.49 m is located on the ground at point A (see Fig. 1). The dimension of its cross-section is shown in Fig. 2. The exhaust air terminal device is installed in the ceiling at point B (see Fig. 1).

The supply air flow is 0.141 m^3/s. The supply air temperature is 21.5 ℃ and the exhaust air temperature is 28.0 ℃. The two sides of the supply air terminal device are against the wall, as shown in Fig. 3. Thus $N = 3$ for the N-point momentum model.

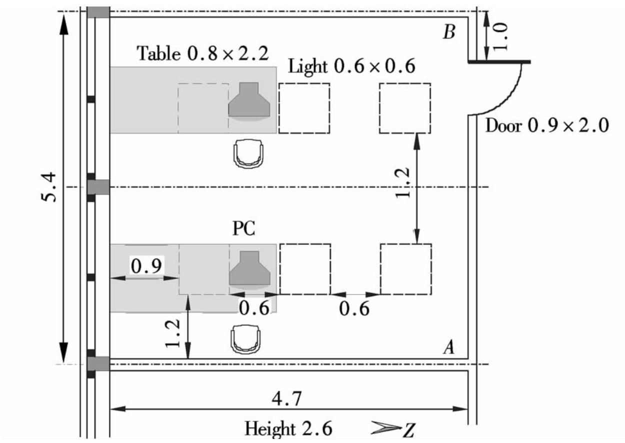

Fig. 1 Layout and dimension of the office room (unit: m)

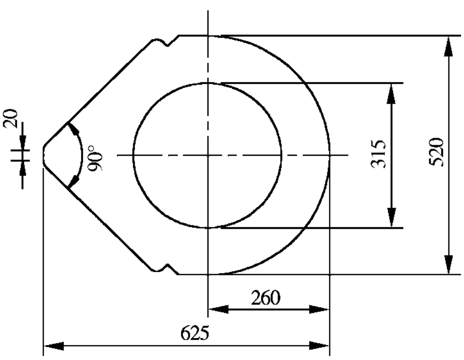

Fig. 2 Cross sectional view of ATD used in the displacement ventilation system in the office (unit: mm)

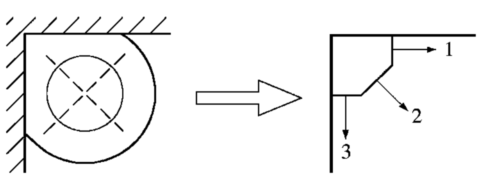

Fig. 3 Sketch map of the N-point momentum model for ATD ($N = 3$)

In the jet main region specification model, the boundary conditions are defined in nine surfaces perpendicular to directions of 1, 2 and 3. In each direction, there are three surfaces and they are denoted as S_{i1}, S_{i2}, S_{i3} ($i = 1, 2, 3$) from down to up. According to Eqs. (1) to (3), the air velocity and the temperature of discharge are calculated and shown in Tab. 1.

Tab. 1 Boundary conditions of nine surfaces in the main jet region specification model

Surfaces	Height/m	Air temperature/℃	Air velocity/($m \cdot s^{-1}$)
S_{11}	0.43	18.90	0.32
S_{12}	0.40	17.87	0.31
S_{13}	0.38	17.38	0.30
S_{21}	0.21	19.50	0.12
S_{22}	0.09	18.88	0.08
S_{23}	0.03	18.41	0.08
S_{31}	0.43	18.90	0.32
S_{32}	0.40	17.87	0.31
S_{33}	0.38	17.38	0.30

1.3 Full-scale experiment

A full-scale experiment is performed in order to make a comparison with the CFD simulation. The test room has a modular structure based on a load-bearing steel construction with a module of 1.2 m. Its size is 5.4 m × 4.7 m × 2.6 m. The walls surrounding the perimeter are made of insulated chipboard. One of the walls has a glass wall with a height of 2.4 m. The floor is made of a 0.6 m × 0.6 m chipboard that is raised 0.5 m above a structural concrete slab.

The measurement system consists of 16 omni-directional thermal anemometers. The instantaneous values of velocity and temperature are measured simultaneously. The anemometers are calibrated before measurement. They are grouped in two sets of 8 (modules 1 and 2). Each set is mounted on a mobile stand. The velocity and temperature fields near the occupant area are measured. Fig. 4 shows the position of the mobile stand, i. e., point 1 ($x = 3.5$ m, $y = 1.1$ m) and point 2 ($x = 3.5$ m, $y = 1.6$ m). The heights of the anemometers on the mobile stand are 0.05, 0.1, 0.3, 0.6, 1.1, 1.4, 1.7 and 2.2 m above the floor, respectively. In order to compare the performances of different models in simulating the air flow around ATD, the air velocities at the points shown in Fig. 4(b) are measured. Coordinates are (4.6, 1.4, 0.1) for point 3, (4.1, 0.7, 0.1) for point 4 and (3.4, 0.2, 0.1) for point 5.

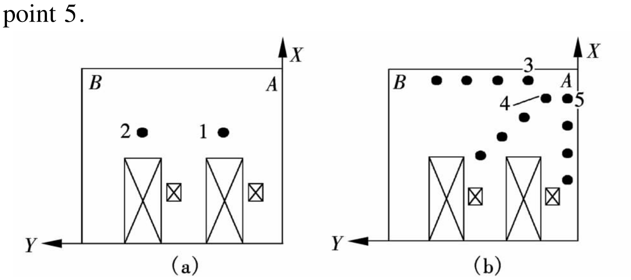

Fig. 4 Positions of the points measured in the full-scale experiment. (a) Measuring points at occupant area; (b) Measuring points around ATD

2 Results and Discussion

In order to demonstrate the advantages and the applicability of the jet main region specification model in modeling the

ATD with complicated geometries, other conventional methods, i. e. , the basic model, the box model, the N-point momentum model, are also used in this paper to describe the ATD. The comparisons of the simulated air velocities and the measured values at points 1 and 2 are shown in Fig. 5. The Y axis is the ratio of measured velocity to supply air velocity. The X axis is the ratio of height at the measured point to the height of the space.

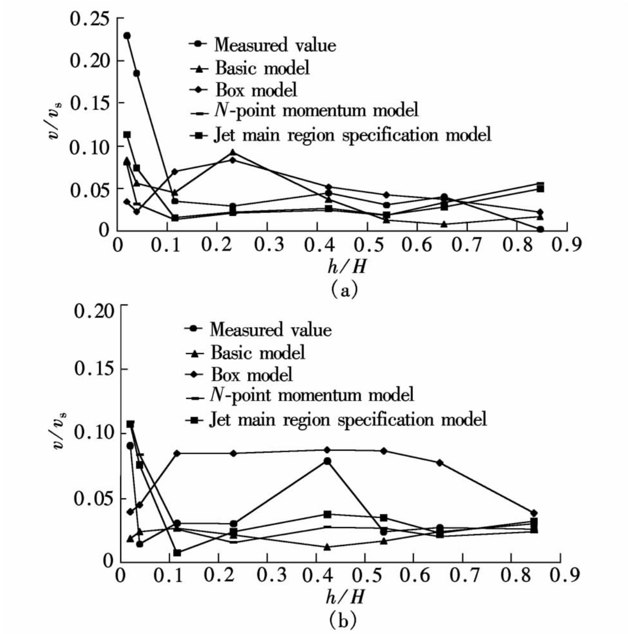

Fig. 5 Comparison between measured and simulated air velocities at occupant area based on different air supply opening models. (a) At point 1; (b) At point 2

Fig. 5 shows that the basic model is completely invalid in simulating the ATD in this study. The velocity increases unexpectedly at the point that $h/H = 0.23$. This is due to the fact that the effective area in the basic model is difficult to define for the ATD with complicated and unregulated geometry. Although the box model can predict the air movement trends in the tested chamber, the relative error between the simulated and measured values is up to 85% at the point that $h/H = 0.019$. This is because the box model cannot model the jet flow at a certain angle, as in the case of the present case study. The N-point momentum model and the jet main region specification model both have good agreement with the measured values with a relative error less than 20%. But the simulated air temperature in the jet main region specification model does not fit the measurement values well (data are not shown). This indicates that the equation to evaluate air temperature (see Eq. (2)) needs to be improved in the future study.

The comparison of simulated and measured velocities around the ATD is shown in Fig. 6. The Y axis is the ratio of measured velocity to supply air velocity. The X axes are the ratios of the coordinates at the measured point to the length (Fig. 6(a)), width (Fig. 6(b)) and radius (Fig. 6(c)) of the space. The simulated air velocity by using the jet main region specification model is closer to the measured value, compared with the N-point momentum model.

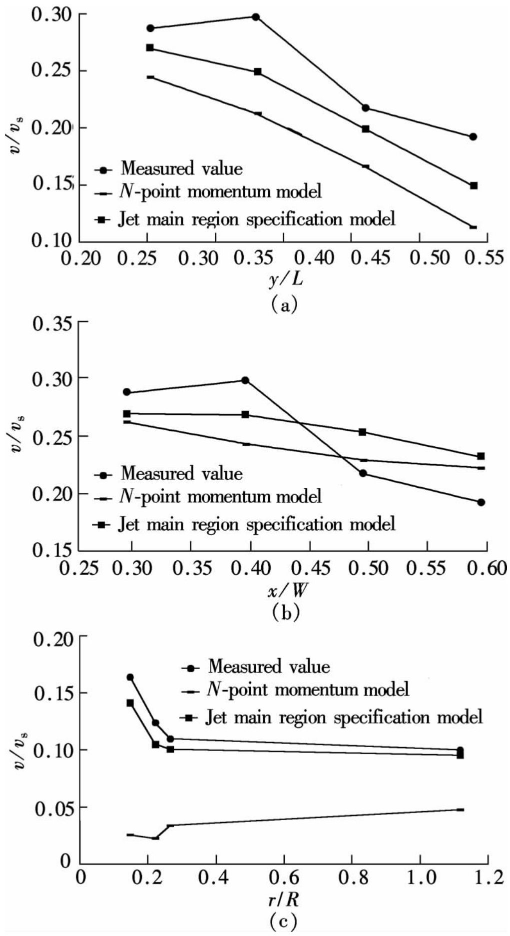

Fig. 6 Comparison between measured and simulated air velocities around ATD based on the N-point momentum model and the jet main region specification model. (a) Along Y axis; (b) Along X axis; (c) Along radius

In the jet main region specification model, the boundary conditions of the volume are calculated using the diffuser jet characteristic equations. For three-dimensional diffusers, the modeling process is that, part of the volume surfaces needs to be located at the beginning of the jet main region and the other part of the volume surfaces should be selected to make the volume as small as possible. This would reduce the inaccuracy of the simplification. The advantage of this method is that users can avoid describing the complicated diffuser geometry by using data from manufacturers' catalogs. The jet main region specification model is applicable for different diffusers, thus providing an accurate prediction of the air movement patterns in a ventilated room.

It is very important to find the suitable diffuser characteristic equations for the jet main region specification model. In this paper, the equations for air velocity and temperature calculation are provided. Huo et al. [7] compared the efficiency and effectiveness of models in simulating the ATD, i. e. , the basic model, the box model, the N-point momentum model and the jet main region specification method. However, their applications and validations were not provided. In the present case study, the jet main region specification model is proved to be an applicable approach and a more accurate way to study the airflow pattern around the ATD with complicated geometry.

3 Conclusion

The jet main region specification model can describe well ATD with complex structure and predict the air velocity field of jet flow with a certain jet angle. The jet main region specification model is easy to implement and has high prediction accuracy, which benefits its application in engineering. However, the equation for temperature description in modeling ATD needs improvements in future study.

References

[1] Posner J D, Buchanan C R, Dunn-Rankin D. Measurement and prediction of indoor air flow in a model room [J]. *Energy and Buildings*, 2003, **35**(5): 515 − 526.

[2] Zhao Bin, Li Xianting, Chen Qingyan. A simplified system for indoor airflow simulation [J]. *Building and Environment*, 2003, **38**(4): 543 − 552.

[3] Bartak M, Beausoleil-Morrison I, Clarke J A, et al. Integrating CFD and building simulation [J]. *Building and Environment*, 2002, **37**(8/9): 865 − 871.

[4] Nielsen P V. The box method—a practical procedure for introduction of an ATD in CFD calculation [D]. Aalborg, Denmark: Aalborg University, 1997.

[5] Chen Q, Moser A. Simulation of a multiple nozzle diffuser [C]//*Proceedings of the* 12*th AIVC Conference*. Ottawa, Canada, 1991: 1 − 14.

[6] Huo Y, Zhang J S, Shawe C, et al. A new method to specify the diffuser boundary conditions in CFD simulation [C]// *Proceedings of the* 5*th International Conference on Air Distribution in Rooms*. Yokohama, Japan, 1996: 232 − 240.

[7] Huo Y, Haghighat F, Zhang J S, et al. A systematic approach to describe the ATD in CFD simulation for room air distribution analysis [J]. *Building and Environment*, 2000, **35**(6): 563 − 576.

主流区风口模型在室内气流 CFD 模拟中的应用

王志刚[1,2]　张于峰[1]　孙越霞[3]

([1]天津大学环境科学与工程学院,天津 300072)

([2]天津市市政工程设计研究院,天津 300051)

([3]Texas Institute of Allergy, Indoor Environment and Energy, University of Texas at Tyler, Tyler 75799, TX, USA)

摘要:分析了 4 类风口模型(即基本模型、盒子模型、*N* 点动量模型、主流区风口模型)在计算流体动力学中的应用以及在实际工程案例中的运行情况. 在环境舱内进行了足尺实验,利用全方位热风速表测量了环境舱内的气流速度和温度场,并与 4 类风口模型的模拟结果进行比较. 结果表明:基本模型与盒子模型并不适用于结构复杂的送风口模拟;基于 *N* 点动量模型和主流区风口模型的流体速度在工作区的模拟值与实测值的相对误差小于 20%;在主流区风口模型中,送风口附近的气体速度模拟值与实测值的最大相对误差小于 15%. 证明主流区风口模型相对于其他 3 类风口模型而言,对于结构复杂的送风口模拟更准确,应用性更强.

关键词:计算流体动力学;风口模型;主流区风口模型;室内气流

中图分类号:TU83