A simplified system for indoor airflow simulation

Article *in* Building and Environment · April 2003

DOI: 10.1016/S0360-1323(02)00182-8

Bin Zhao
Tsinghua University

Xianting Li
Tsinghua University

PERGAMON

Building and Environment 38 (2003) 543–552

BUILDING AND ENVIRONMENT

www.elsevier.com/locate/buildenv

A simplified system for indoor airflow simulation

Bin Zhao[a], Xianting Li[b,*], Qisen Yan[b]

[a]*Department of Engineering Mechanics, Tsinghua University, Beijing, China*
[b]*Department of Building Science, School of Architecture, Tsinghua University, Beijing, China*

Received 25 September 2001; received in revised form 11 June 2002; accepted 12 September 2002

Abstract

This paper proposes a simplified system based on a new air supply opening model and a numerical method of solving the discrete algebraic equations to accelerate and simplify the convergence procedure of predicting air distribution in ventilated rooms. The so-called "*N*-point air supply opening model" is applied to describe boundary conditions of air terminal devices in computational fluid dynamics calculation. It directly and accurately describes inlet boundary conditions and models inlet mass, momentum and buoyancy flows. The new numerical method is called error pre-treatment method, which solves the algebraic equations first on coarse meshes to pre-treat low-frequency error before solving on finer meshes. It is simpler than multi-grid method and effective for SIMPLE algorithm that is commonly used for indoor airflow simulation. A zero-equation turbulence model is adopted to further simplify the simulation. These models and methods constitute the simplified system of indoor airflow simulation. The airflow in a room ventilated by displacement diffuser, square diffuser, and grille diffuser is calculated by the simplified system, respectively. Comparing calculated results to measured data, it is evident that the simplified methodology can predict indoor airflow quickly with satisfactory results for engineering applications.

Keywords: CFD; Indoor airflow; Boundary condition; Air supply opening model; Turbulence model

1. Introduction

Recent advances in computational fluid dynamics (CFD) and computer power make it possible to accurately predict some features of airflow within ventilated spaces. The CFD method has been successfully applied for airflow analysis in relatively complicated conditions, such as non-isothermal, three-dimensional and with furniture inside the room. However, it usually requires lots of meshes and consumes much time to get convergence for engineering problems. It is important to develop simple models and methods to simulate indoor airflow so that heating, ventilating and air-conditioning (HVAC) designers or architects can finish the predictions in short time to meet the engineering demand on a personal computer. The main factors affecting the simulation speed are the method of describing inlet boundary conditions [1], turbulence model used [2], and the numerical method of solving the discrete equations [3].

The velocity level in a room ventilated by air jet is strongly influenced by the air supply conditions. The momentum flow in supply jets controls air movement in the room. Therefore, it is very important to describe the momentum flow correctly. The momentum flow from air supply openings depends on small details in the design. This means that a numerical prediction method should be able to handle details from the order of a few millimeters for openings to a few meters for a room. This wide range of geometry necessitates the use of many grid points and therefore demands a large computer. However, many HVAC engineers and architects do not have access to a large computer. It is important for them to predict the indoor airflow quickly with acceptable accuracy. Besides, HVAC designers usually are not able to use commercial CFD codes to model the complex geometry of the air terminal device in a room. An applied method for describing air inlet boundary conditions, also called air supply opening model (ASOM), should be developed to simplify the predicting procedure.

On the other hand, some researchers pointed out that the k–ε turbulence model is most proper for simulating

* Corresponding author. Tel.: +86-10-627-85860; fax: +86-10-6277-3461.
E-mail address: xtingli@tsinghua.edu.cn (X. Li).

0360-1323/03/$ - see front matter
PII: S0360-1323(02)00182-8

the turbulent airflow in ventilated rooms [4,5]. Nevertheless, for three-dimensional, non-isothermal conditions, it still consumes too much time to get convergence, especially for those with complicated diffusers [6,7]. Furthermore, indoor airflow is always mixed convective flow including thermal plumes, wall jets and flows with stratified temperature field. Nielsen [8] pointed out that these flows require different turbulence models. It means even k–ε turbulence model may not get reasonable results.

In addition, solving the algebraic equations is the main time consuming element of a simulation procedure. Although multi-grid method has been developed to accelerate the speed of convergence for solving the non-linear algebraic equations, it is neither easy to apply nor effective for SIMPLE algorithm, which is the most commonly used for indoor airflow simulation [3]. Therefore, it is necessary to develop a simple method to accelerate the convergence for SIMPLE algorithm.

The following sections will present an ASOM called N-point ASOM and a new numerical method of solving algebraic equations called error pre-treatment method. Together with an existing zero-equation turbulence model, the simplified simulation system can quickly predict indoor airflow with acceptable accuracy for engineering applications.

2. The simplified simulation system

2.1. The N-point ASOM

2.1.1. Brief review of ASOM

Since the 1970s, some models describing the inlet boundary conditions for indoor airflow simulation have been developed [9]. Generally, the existing ASOM may be divided into two types: models describing boundary conditions in the vicinity of supply openings, and models converting the description of boundary conditions to the description of a box boundary conditions around the supply opening. The former directly describes the mass and momentum boundary conditions while the latter describes those conditions on the surfaces of the box, which is an indirect one.

Indirect ASOM includes box method [9,10], prescribed velocity method [11] and main region specification method [12]. All the methods transfer the description of the diffuser boundary conditions to the box boundary conditions. And the boundary conditions on the box surface perpendicular to the supply opening surface are considered as $\partial\phi/\partial n = 0$, where n is the tangent direction of the studied surface, and Φ is velocity, kinetic energy and the dissipation rate of kinetic energy, etc. The difference between the box method and the main region specification method is that the former adopts measured data as boundary conditions at the box surface parallel to the supply opening surface, while the latter adopts the jet formulas since one volume surface is located inside the main region of the diffuser jet. For prescribed velocity method, the inlet boundary conditions are specified by conventional method; nevertheless, one component of the velocity is measured inside the assumed box area. The velocity component values inside the box are prescribed as extra boundary conditions to correct the predicted velocity around the inlet area. All these three methods demand either detailed measurements for each supply opening, or adopt the jet formulas. However, detailed measurements are time consuming and may not be practical for engineering applications, and the jet formulas may not be able to predict the velocity around the box correctly due to the complex conditions in a real room. Some examples shown in Ref. [13] indicate the uncertainty of jet formulas. Furthermore, it may cause much error for those cases where the diffuser jet is strongly confined, as the jet is not upstream influence.

However, models directly describing inlet boundary conditions need neither measurements nor empirical formulas. There are mainly two direct ASOMs: basic model and momentum model. Basic model uses a simple opening with the same effective area as the complex diffuser [14] while momentum model describes velocity vector based on diffuser effective area [15]. The former is a coarse method, even for those with small effective area. In actual applications, there are many, these kinds of air supply openings, such as square ceiling diffusers and displacement ventilation diffusers. The example of Ref. [16] shows that the basic model may cause obvious error for a diffuser with small effective area. To keep the appropriate supply air mass flow and to introduce the same amount of air into the room, the boundary conditions for continuity equation and momentum equations have to be described separately in the momentum method [15]. It is not convenient to use this method as most commercial CFD software does not support separate description of boundary conditions for continuity and momentum equations. Besides, for cases that air jet has various angles from the diffuser, momentum method does not tell how to describe the diffusers in detail. For example, how many simple elements should be used to simulate a ceiling diffuser by momentum method?

In principle, direct ASOM can be applied for any case even when the diffuser jet is confined or non-isothermal. Thus, it should be the first choice for engineering application. Momentum method can ensure adequate modeling of inlet mass and momentum flow simultaneously. The following describes a method called "N-point ASOM" that combines the positive features of both direct ASOM and momentum method.

Table 1
The control equations with zero-equation turbulence model

φ	Γ_φ	S_φ
1	0	0
u	μ_{eff}	$-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu_{\mathrm{eff}}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{\mathrm{eff}}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial z}\left(\mu_{\mathrm{eff}}\frac{\partial w}{\partial x}\right)+g_x(\rho-\rho_{\mathrm{ref}})$
v	μ_{eff}	$-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu_{\mathrm{eff}}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu_{\mathrm{eff}}\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{\mathrm{eff}}\frac{\partial w}{\partial y}\right)+g_y(\rho-\rho_{\mathrm{ref}})$
w	μ_{eff}	$-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left(\mu_{\mathrm{eff}}\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial y}\left(\mu_{\mathrm{eff}}\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu_{\mathrm{eff}}\frac{\partial w}{\partial z}\right)+g_z(\rho-\rho_{\mathrm{ref}})$
h	$\frac{\mu_{\mathrm{eff}}}{\sigma_{\mathrm{h}}}$	$S_{\mathrm{h}},\ \sigma_{\mathrm{h}}=1.0$
		$\mu_{\mathrm{eff}}=\mu_{\mathrm{l}}+\mu_{\mathrm{t}}, \mu_{\mathrm{t}}=0.03874\rho Vl$

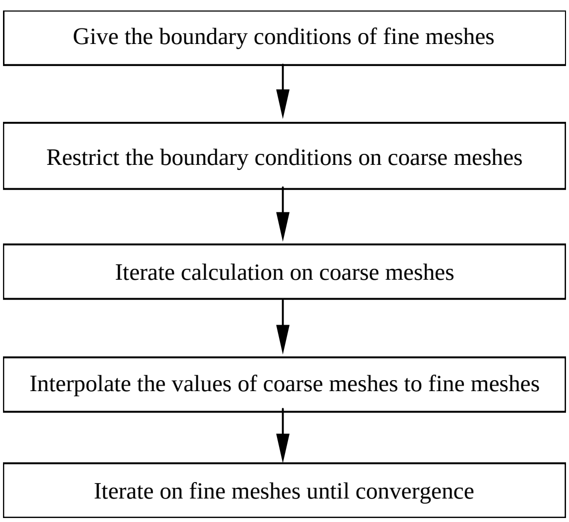

Fig. 1. The flowchart of error pre-treatment method.

2.1.2. The N-point ASOM

The essential of N-point ASOM is to replace the real diffuser by several simple openings so as to reduce the number of grids for numerical calculation. To ensure satisfactory results, the model should be able to simulate the major characteristics influencing the diffuser air jet. The nature of the supply opening (the geometry, outlet turbulence characteristic, etc.), supply mass and momentum flows are known as the key factors influencing the air jet characteristic [17–19]. For non-isothermal air jet, buoyancy flow is also a main factor [19]. N-point ASOM replaces the geometry of a real supply opening by simple openings while maintaining the inlet momentum and mass flows.

For a supply opening with one discharge direction, the actual inlet momentum flow is

$$J_{\mathrm{in}}=mV_{\mathrm{r}}=m\frac{L}{A_{\mathrm{e}}}, \tag{1}$$

where J_{in} is the actual inlet momentum flux (kg m/s^2), m is actual inlet mass flow rate (kg/s), V_{r} is actual inlet velocity, m/s, L is inlet volume flow rate, m^3/s, A_{e} is effective area of the supply opening, m^2.

To describe the right momentum flow while maintaining the same mass (volume) flow rate using replaced simple openings, the effective area coefficient for calculating the inlet momentum source term in the CFD code is used. That is:

$$J_{\mathrm{in}}=m\frac{L}{R_{\mathrm{fa}}A}, \tag{2}$$

where A is the gross area of the supply opening, m^2, R_{fa} is coefficient of effective area, which is the ratio of effective area to gross area of the supply opening, $R_{\mathrm{fa}}\leqslant 1$.

By Eq. (2), inlet momentum flow can be correctly defined and the right inlet mass flow is also given at the same time. It is not necessary to describe boundary conditions for momentum and continuity equations separately. The theory above is the main point of momentum method [15]. Because momentum is a vector, a supply opening may need to be replaced by several simple openings to describe different inlet momentum directions. Zhao and Li [20] studied several types of the commonly used diffusers in ventilated rooms numerically and gave the suggested number of simple openings (N) for different diffusers [20].

Buoyancy flow is another key factor affecting indoor airflow for non-isothermal air jet. The buoyancy inflow (B) of the supply opening is

$$B=g\,\Delta\rho L, \tag{3}$$

where g is the acceleration of gravity, m/s^2, $\Delta\rho$ is difference of supply and indoor air density, kg/m^3, L is supply volume flow rate, m^3/s.

For indoor air, we have

$$\Delta\rho=-\Delta T\beta\rho, \tag{4}$$

$$\beta=\frac{1}{T_0}, \tag{5}$$

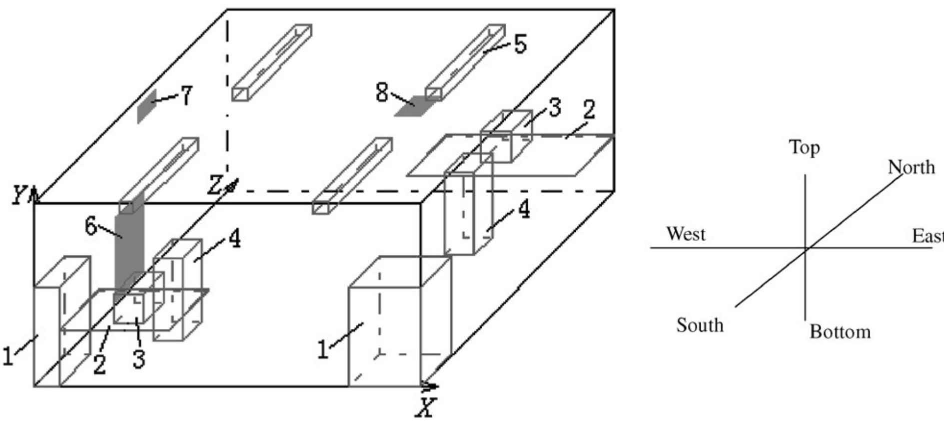

Fig. 2. Sketch of the test chamber and positions of air supply diffusers: (1—cabinet, 2—table, 3—computer, 4—person, 5—lamp, 6—displacement diffuser, 7—grille, 8—square ceiling diffuser).

Table 2
Total heat transfer for different cases

Items	Quantities W
Human simulators	75 × 2
Computers	108+173 close to window
Lamps	34 × 4
External heat transfer east wall and window	135[a] 161[b] 158[c]
Gross	710[a] 728[b] 725[c]

[a]Displacement ventilation.
[b]Grille ventilation.
[c]Square ceiling ventilation.

where ΔT is the difference of supply and indoor air temperature (K), β is volumetric coefficient of expansion $(1/K)$, T_0 is average temperature of indoor air (K).

From Eqs. (3)–(5), the buoyancy flow is

$$B = -g\,\frac{\Delta T}{T_0}\,\rho L. \tag{6}$$

The N-point ASOM can ensure B if L is correctly defined. Therefore, it can be applied for both isothermal and non-isothermal air jets.

Some validation cases of N-point ASOM can be found in Ref. [20]. This paper will further study non-isothermal, three-dimensional room flows with furniture inside using N-point ASOM combined with a zero-equation turbulence model and an error pre-treatment method.

2.2. The zero-equation turbulence model

To simulate room airflow quickly, Chen and Xu [21] developed a zero-equation turbulence model by directly numerical simulation (DNS) data. The model uses a single algebraic equation to express the turbulent viscosity

$$\mu_t = 0.03874\rho V l, \tag{7}$$

where l is the distance to the nearest closure and V is the local mean velocity.

Based on Eq. (7), the Reynolds averaged Navier–Stokes (RANS) equations are closed. The governing equations of mass, momentum and energy for indoor airflow can be written in a general form as follows:

$$\frac{\partial}{\partial t}(\rho\varphi) + \mathrm{div}(\rho\vec{u}\varphi - \Gamma_\varphi\,\mathrm{grad}\,\varphi) = S_\varphi. \tag{8}$$

The details of φ, Γ_φ and S_φ are given in Table 1.

Where u, v, w are the three velocity components in x, y, z directions, respectively. μ_l, μ_t, μ_{eff} are laminar, turbulence and effective viscosity, respectively, p is air pressure and h is air enthalpy.

Table 3
Flow boundary conditions for different diffusers

Diffusers	Air exchange rate (ACH)	Supply air temperature T_{in}(°C)	Exhaust air temperature T_{out}(°C)	Gross area of diffuser A (m^2)	Coefficient of effective area, R_{fa}
Displacement	5.0	13	22.2	1.1 × 0.53	0.4
Grille	5.0	15.1	24.5	0.28 × 0.18	0.67
Square ceiling	4.9	14.5	24.1	0.3 × 0.3	0.177

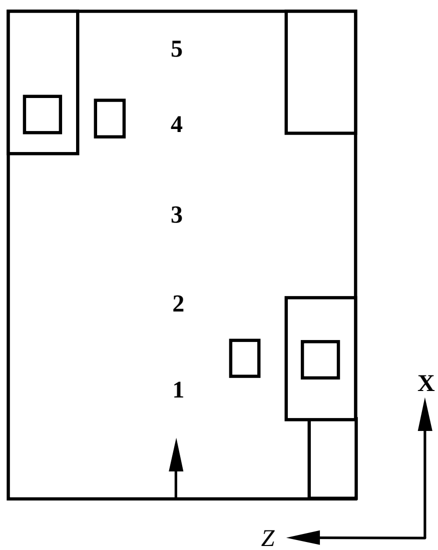

Fig. 3. Measuring positions in the test chamber.

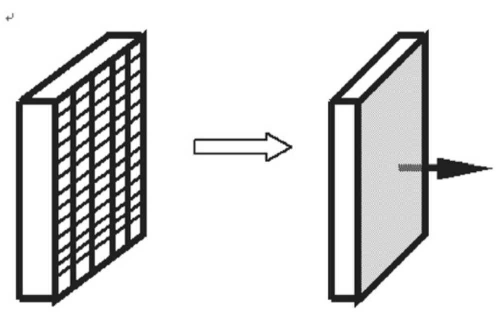

Fig. 4. The N-point air supply opening model for the displacement diffuser ($N = 1$).

The above zero-equation turbulence model has been used to simulate indoor airflow for natural, forced, and mixed convection cases [2,21,22]. Reasonable agreement between numerical results was achieved in shorter time than by k–ε turbulence model or more complicated ones.

2.3. The error pre-treatment method

According to the error theory of multi-grid method that iteration error can be low frequency on coarse meshes and high frequency on finer ones, an "error pre-treatment method" which is simpler than the multi-grid method for solving the algebraic equations has been developed [23]. Fig. 1 shows the flowchart of the method. The method is divided into two steps. First, the low-frequency error is cleared up on coarse meshes. When convergence is achieved, the second step is started on fine meshes, where the results of first step are adopted as initial data of iterative calculation for fine grids.

Examples of isothermal and non-isothermal indoor airflow simulations show that the method can reduce the convergence time by about $1/3 - 1/2$ compared to conventional method [23,24].

2.4. Simulation tool

A CFD program, STACH-3 [25], is modified to fulfill the simplified simulation system. The software adopts finite volume method and SIMPLE algorithm, where the momentum equations are solved on staggered grids. The differential scheme is hybrid scheme. Both the false time step and linear relaxation are used to ensure convergence.

3. Validation of the simplified simulation system

This section demonstrates the simplified simulation system by applying it to predict indoor airflow in ventilated rooms. The air supply openings used in the simulation include displacement diffuser, grille and square ceiling diffusers, respectively.

3.1. Validation conditions

The experimental data used to compare with simulation results are from Ref. [26]. The airflow pattern and the distributions of air velocity and temperature were measured in a full-scale test chamber shown in Fig. 2. The test chamber has a neighboring climate chamber used to simulate the outdoor climate. Table 2 presents values of the internal heat sources and heat transfer from the external wall and window. The internal heat sources, including persons, computers and lamps were the same for the three cases. The variable part of the cooling load was heat transfer through the external wall and window between the test chamber and climate chamber. The flow boundary conditions for supply and return diffusers are given in Table 3. The flow rates were in the range usually applied for mechanical ventilation of small office spaces.

The measuring positions were located at five poles in the chamber (see Fig. 3), and vertical distributions of air velocity and temperature were measured.

3.2. Validation cases

3.2.1. Displacement diffuser

The displacement diffuser has a low momentum from the air supply. In this case, the inflow air directions are parallel and form one "beam" [26]. According to Ref. [20], N is equal to 1 for the N-point ASOM, see Fig. 4.

The simulated results compared with the experimental data are shown in Fig. 5, θ is dimensionless temperature: $\theta = T - T_{in}/(T_{out} - T_{in})$. The agreement for

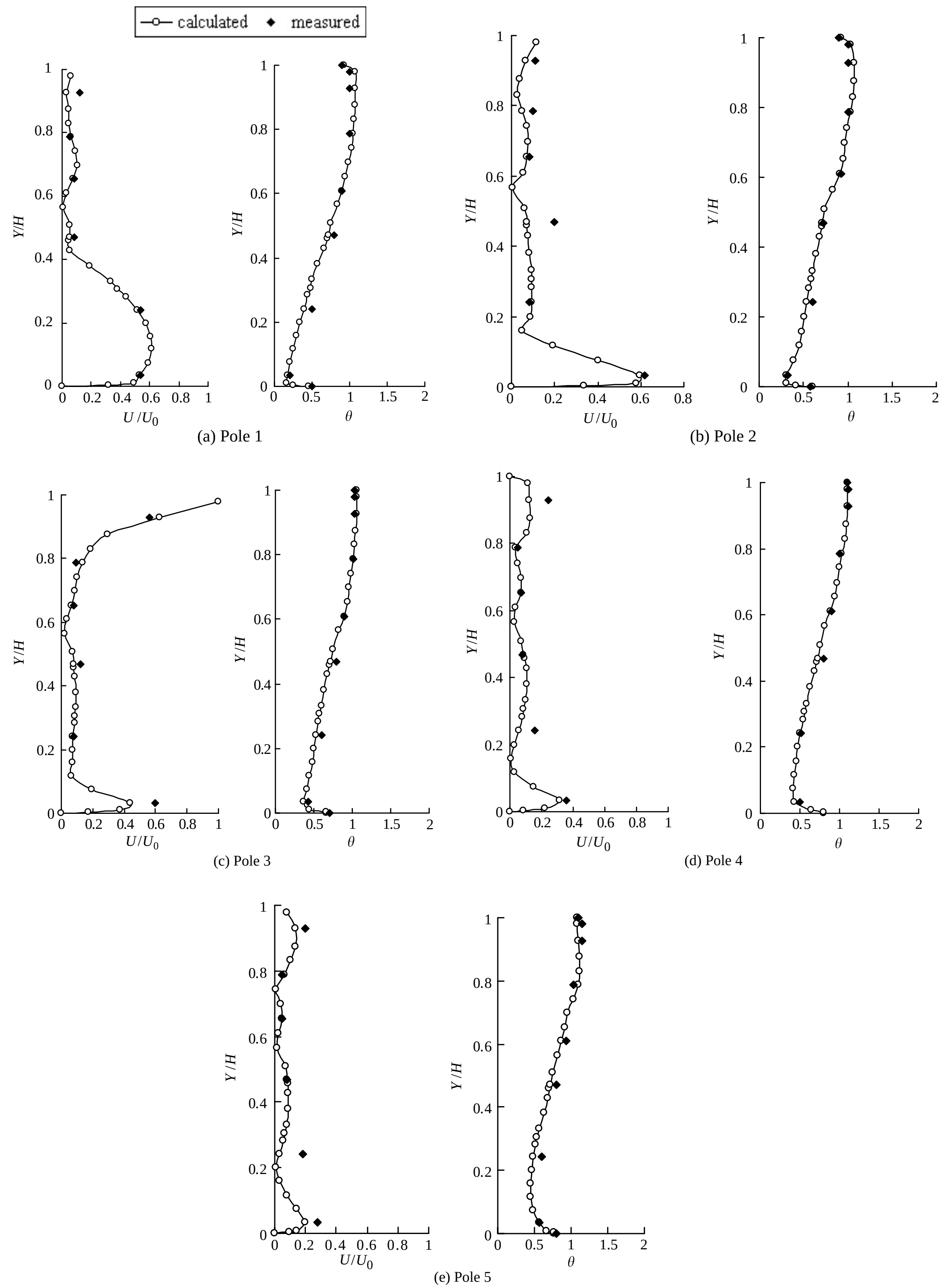

Fig. 5. Comparison of calculated and measured velocity and temperature profiles for displacement ventilation case at five positions in the room $\theta = T - T_{in}/(T_{out} - T_{in})$.

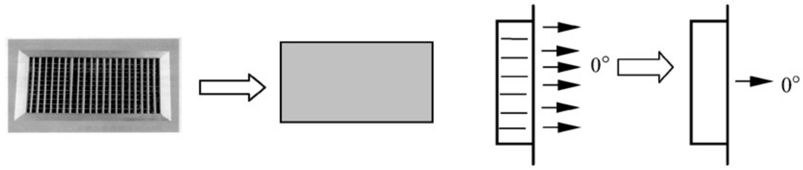

Fig. 6. The N-point air supply opening model for the grille diffuser ($N = 1$).

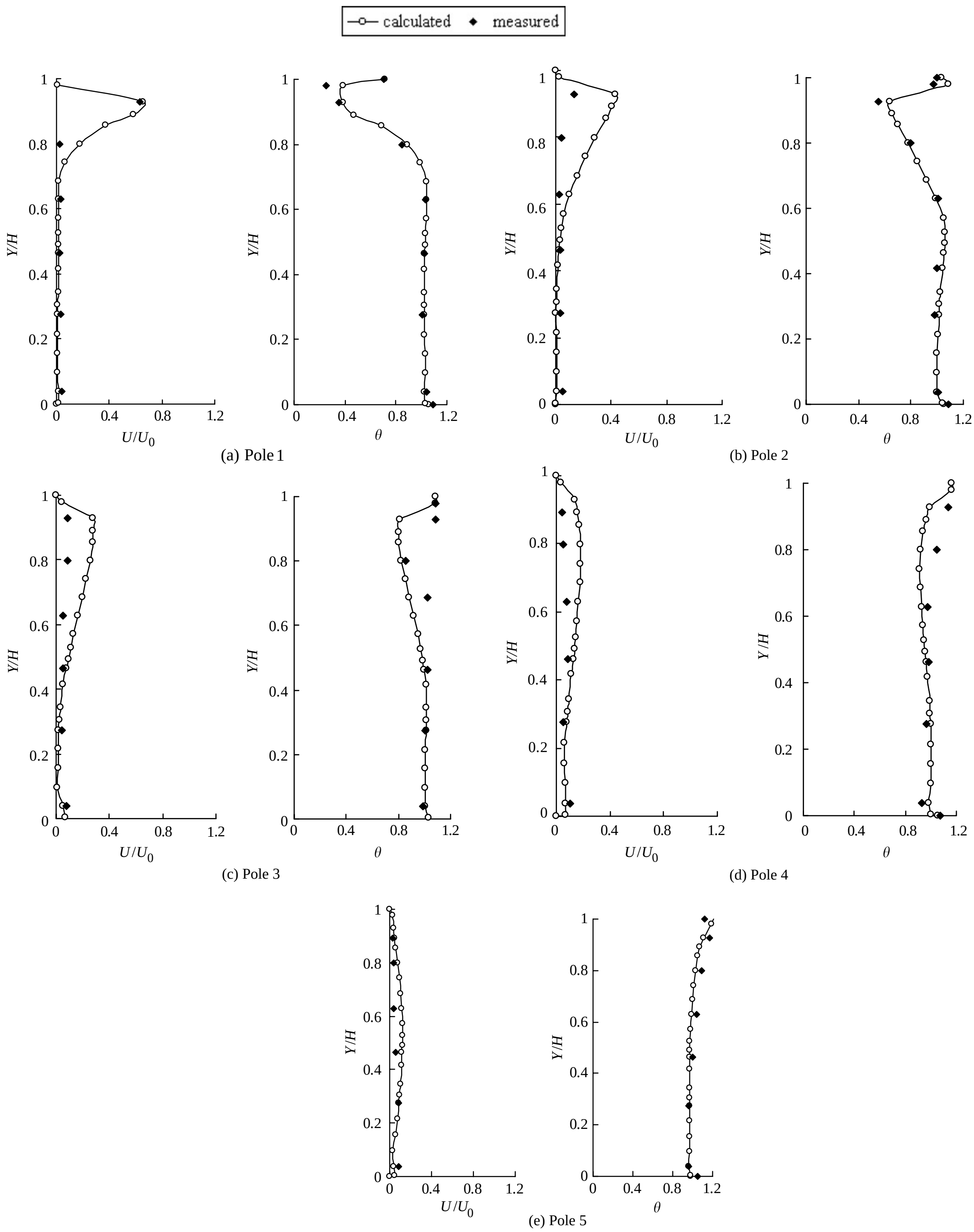

Fig. 7. Comparison of calculated and measured velocity and temperature profiles for grille diffuser case at five positions in the room.

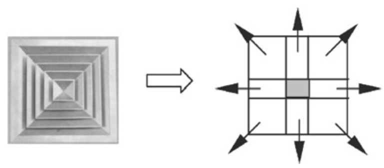

Fig. 8. The N-point supply opening model for the square ceiling diffuser ($N = 9$).

velocity is acceptable considering the relatively low velocity in this case. The simulated temperature distributions agree very well with the measured temperatures, and the room air temperature stratification is correctly predicted. The grids calculated are $48 \times 38 \times 21$. It took about 4 h to get convergence on a Pentium III 933 MHz PC, which is much faster than using

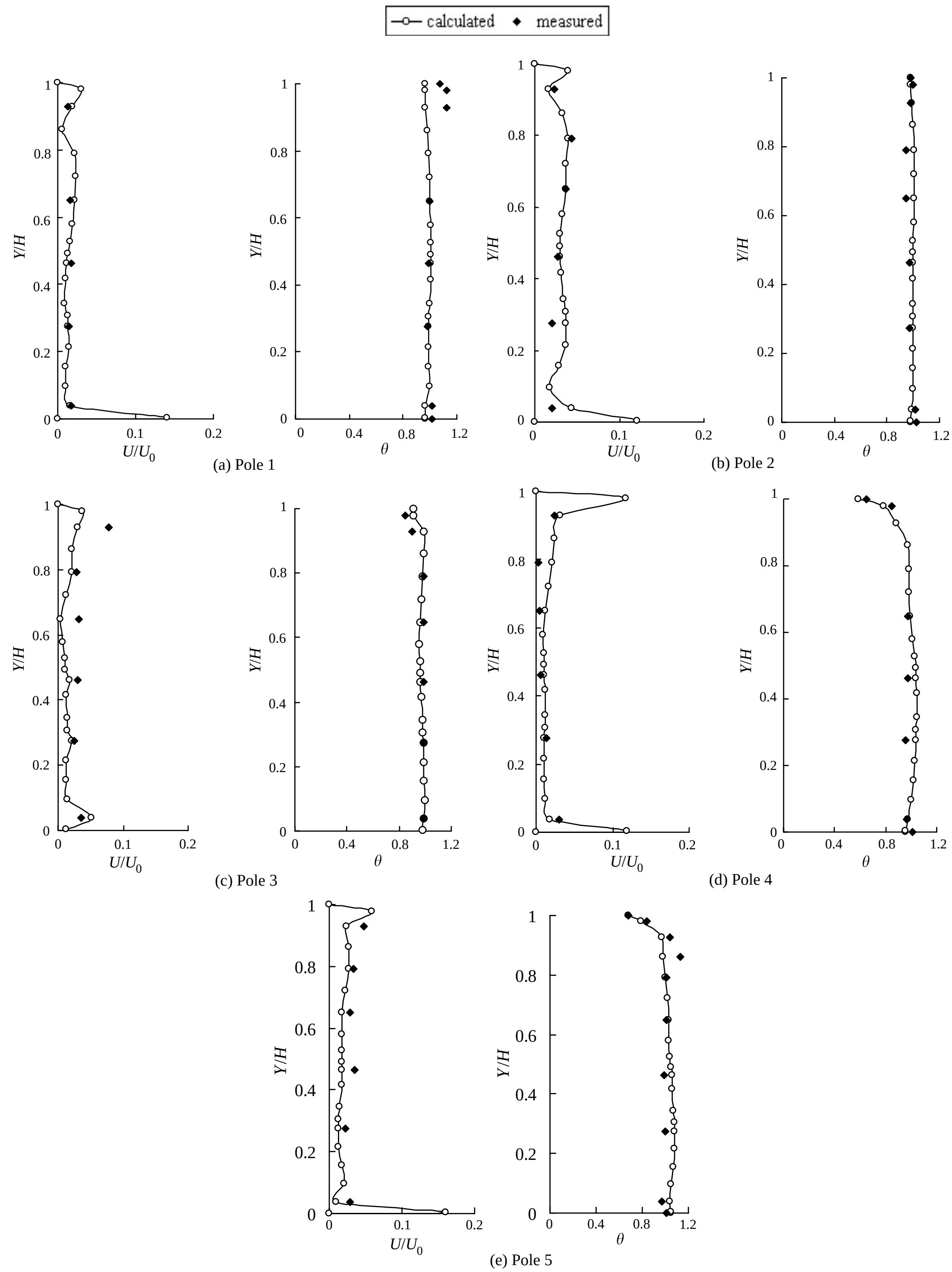

Fig. 9. Comparison of calculated and measured velocity and temperature profiles for square ceiling diffuser case at five positions in the room.

k–ε turbulence model and conventional method of solving algebraic equations. The similar case will cost about 24 h on the same computer by k–ε turbulence model and conventional algebraic equations solving method [25].

3.2.2. Grille diffuser

In this case, the supply air "beam" is one so $N = 1$ according to Ref. [20] (see Fig. 6). Fig. 7 presents the calculated and measured velocity and temperature distributions. The velocity in jet region (pole 1) is well predicted, while the distributions in other poles are also acceptable. The simulated temperatures near the wall region do not agree well with the measured data. Nevertheless, the uniform temperature distribution in lower part of the room (working zone) is correctly predicted. The computing time for this case on the computer mentioned above is about 3 h (for a mesh of $35 \times 23 \times 30$).

3.2.3. Square ceiling diffuser

For this case, the supply air spreads in all directions so it is more complicated than the two cases above. According to the outlet characteristic, N is set to nine as shown in Fig. 8 [20].

Fig. 9 gives the measured and calculated air velocities and temperatures at five positions in the room. The air temperature in lower part of the room is correctly predicted. This is not surprised since the room airflow is well mixed and the energy balance ensures a correct mean air temperature. The calculated temperatures near the ceiling do not agree so well with the measured data, possibly because of the limitation of the zero-equation turbulence model used for near-wall turbulence modeling. Nevertheless, the simulated results in working zone are reasonable for directing engineering design. For a mesh of $32 \times 21 \times 29$, about 3.5 h are needed to get convergence on the same computer.

4. Conclusions

A simplified system for indoor airflow simulation is developed based on the N-point ASOM, a zero-equation turbulence model and an error pre-treatment method. Three cases using different kinds of air supply diffusers are simulated to demonstrate the application of the simplified method. The following conclusions may be drawn from this study:

(1) The simplified system can correctly simulate air velocity and temperature distributions in the room except in a few positions.
(2) Using the simplified simulation system costs much shorter computing time than using conventional CFD methods, making it possible for engineers to apply in design procedure.

Acknowledgements

Dr. Qingyan Chen of Massachusetts Institute of Technology, and Dr. Jelena Srebric of Pennsylvania State University kindly provided the measured data. Dr. Chen also provided key references of air supply opening models. Their support is highly appreciated.

References

[1] Nielsen PV. Description of supply openings in numerical models for room air distribution. ASHRAE Transactions 1992;98(1):963–71.
[2] Xu Weiran. New turbulence models for indoor airflow simulation. Ph.D. thesis, Massachusetts Institute of Technology, USA, 1998.
[3] Liu Chaoqun. Multigrid method and application in computational fluid dynamics. Beijing: Tsinghua University Press, 1995 [in Chinese].
[4] Chen Qingyan. CFD for HVAC: success and failure. ASHRAE Transaction 1996;102(1):178–86.
[5] Jones PJ, Whittle GE. Computational fluid dynamics for building airflow prediction—current status and capabilities. Building and Environment 1992;27(4):321–8.
[6] Srebric J. Simplified methodology for indoor environment design. Ph.D. thesis, Massachusetts Institute of Technology, USA, 2000.
[7] Chen Qingyan, Moser A, Huber A. Prediction of buoyant, turbulent flow by a low-Reynolds-number k–ε model. ASHRAE Transaction 1990;96(1):564–73.
[8] Nielsen PV. The selection of turbulence models for prediction of room airflow. ASHRAE Transactions 1998;104(2):1119–27.
[9] Nielsen PV, Restivo A, Whitelaw JH. The velocity characteristics of ventilated rooms. Transactions of ASME Journal of Fluids Engineering 1978;100:291–8.
[10] Nielsen PV. The box method—a practical procedure for introduction of an air terminal device in CFD calculation. Department of Building Technology and Structural Engineering, Aalborg University, Denmark, 1997.
[11] Nielsen PV. The prescribed velocity method—a practical procedure for introduction of an air terminal device in CFD calculation. Department of Building Technology and Structural Engineering, Aalborg University, Denmark, 1998.
[12] Huo Y, Haghighat F, Zhang JS, Shaw CY. A systematic approach to describe the air terminal device in CFD simulation for room air distribution analysis. Building and Environment 2000;35(6):563–76.
[13] Srebric J, Liu J, Chen Q. Experimental validation of jet formulae for air supply diffusers. Proceedings of Seventh International Conference on Air Distribution in Rooms (ROOMVENT'2000). Reading University, United Kingdom, July 9–12, 2000.
[14] Heikkinen J. Modeling of a supply air terminal for room air flow simulation. Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within buildings, vol. 3. Ottawa, Canada, 1991. p. 213–30.
[15] Chen Q, Moser A. Simulation of a multiple-nozzle diffuser. Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within buildings, vol. 2. Ottawa, Canada, 1991. p. 1–13.
[16] Emvin P, Davidson L. A numerical comparison of three inlet approximations of the diffuser in case E1 Annex20. Proceedings of ROOMVENT'96, vol. 1. 1996. p. 219–26.
[17] Chen Q, Suter P, Moser A. Influence of air supply parameters on indoor air diffusion. Building and Environment 1991; 26(5)(4): 417–31.
[18] Li ZH, Zhang JS, Zhivov AM, Christianson LL. Characteristics of diffuser air jets and airflow in the occupied regions of mechanically ventilated rooms—a literature review. ASHRAE Transaction 1993;99(1):1119–26.

[19] Yu Changzhao. Turbulence jets. Beijing: Senior Education Press, 1993 [in Chinese].
[20] Zhao Bin, Li Xianting. The N-point air supply opening model for indoor airflow simulation. Department of Building Science, School of Architecture, Tsinghua University, China, 2000.
[21] Chen Qingyan, Xu Weiran. A zero-equation turbulence model for indoor air flow simulation. Energy and Building 1998;28(2):137–44.
[22] Zhao Bin, Li Xianting, Yan Qisen. Simulation of indoor air flow in ventilated room by zero-equation turbulence model. Journal of Tsinghua University (Science & Technology) [in Chinese], 2001;41(10):109–113.
[23] Zhao Bin, Li Dongning, Li Xianting, Yan Qisen. Error pre-treatment method for numerical simulation of indoor air flow. Journal of Tsinghua University (Science & Technology) [in Chinese], 2001;41(10):114–117.
[24] Li Dongning, Zhao Bin, Li Xianting. The engineering application of error pre-treat method. Proceedings of International Conference on Energy Conversion and Application (ICECA'2001). Wuhan, China, June 17–20, 2001. p. 188–91.
[25] Zhao Bin, Li Xianting. The STACH-3 manual. Department of Building Science, School of Architecture, Tsinghua University, China, 2000.
[26] Chen Q, Srebric J. Simplified diffuser boundary conditions for numerical room airflow models. Final report for ASHRAE RP-1009, MIT, Cambridge, 2000.