Xu, W. and Chen, Q. 2000. “Simulation of mixed convection flow in a room with a two-layer turbulence model”, *Indoor Air*, 10, 306-314.

Simulation of Mixed Convection Flow in a Room with a Two-Layer Turbulence Model

WEIRAN XU AND QINGYAN CHEN*

Abstract Most indoor airflows are mixed convection. In order to simulate mixed convection accurately and efficiently, this paper uses a two-layer turbulence model. The two-layer model combines a one-equation model for near wall flow together with the standard k-ε model for outer-wall flow. The model has been used to predict the mixed convection by displacement ventilation in an office. The computed results agree well with the corresponding airflow pattern and the distributions of air temperature, air velocity, air velocity fluctuation, and tracer-gas concentration. The model can predict correctly heat transfer from a wall where the standard k-ε model and re-normalization group (RNG) k-ε model with wall functions often fails. The computing cost required by the two-layer model is comparable to that of the standard k-ε model and RNG k-ε model and is significantly less than that by a low-Reynolds number model.

Practical Implications

This paper presents a new two-layer model that can be used to calculate natural, forced, and mixed convection flow in a room. The model can predict correctly room airflow and heat transfer while use similar numerical grid cells as the standard k-ε model. It is particularly useful for rooms with many heated or cooled surfaces. This is because the computing costs are not higher than that with the standard k-ε model, but the computed air flow and heat transfer are more accurate.

Key words CFD; turbulence modeling; validation; displacement ventilation; office rooms.

Nomenclature

Ar_y	local Archimedes number, $g\beta\Delta T y_n / U^2$
C	Contaminant concentration
$C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$	constants used in the ε equation
C_μ	constant used for calculating ν_t

Building Technology Program, Massachusetts Institute of Technology, 77 Mass. Ave.,

Cambridge, MA 02139-4307, U.S.A., Tel: 1-617-253-7714, Fax: 1-617-253-6152.

*Author to whom correspondence should be addressed.

d_k	diffusion of turbulent kinetic energy
$f_{l\mu}, f_{l\varepsilon}, f_{\nu\nu}$	functions used in the one-equation model
g_i	component *i* of the gravitation vector
G_k	gravity production of turbulent kinetic energy, $-\beta g_i \overline{u_i t}$
k	turbulent kinetic energy, $\overline{u_i u_i} / 2$
l, l_μ, l_ε	characteristic lengths
P_k	shear production of the turbulent kinetic energy, $-\overline{u_i u_j}\, \partial U_i / \partial x_j$
Pr_t	turbulent Prandtl number
T	mean temperature
t	fluctuating temperature or time
U_i, U_j	component *i* and j of the mean velocity
U, V, W	mean velocity component in *x, y* and *z* direction
u, υ	fluctuating velocity component in *x, y* direction
x_i, x_j	spatial coordinate in i and j direction
x, y, z	spatial coordinate
y_n	normal distance to the nearest wall
y^*	dimensionless wall distance, $y_n \sqrt{k} / \nu$

Greek Symbols

β	thermal expansion coefficient, $-\frac{1}{\rho}\frac{\partial \rho}{\partial T}$
ε	turbulent energy dissipation
ν	molecular viscosity
ν_t	turbulent viscosity
$\sigma_k, \sigma_\varepsilon$	Prandtl number of *k* and ε

Subscripts

i, j	spatial coordinate indices
in	inlet
t	turbulent quantities

Introduction

Design of an acceptable indoor environment requires detailed information of indoor air distribution, thermal comfort, indoor air quality, etc. The information includes air velocity, temperature, contaminant concentration, turbulence intensity, etc. that can be predicted by computational fluid dynamics (CFD). The CFD application to indoor environment design has enjoyed a great success in the past two decades. However, there are still some uncertainties in the CFD results. This is because most indoor airflows are turbulent. A turbulence model should be used in the CFD to make the flow solvable with the present computer capacity and speed. Unfortunately, the existing turbulence models are either inaccurate (such as the standard k-ε model (Launder and Spalding 1974) or inefficient (such as low-Reynolds number standard k-ε models and Reynolds stress models) to predict indoor airflow.

Most indoor airflows involve three types of convection: forced, natural and mixed convection. For example, the flow from an air-conditioning device is forced convection, and the thermal plumes around the equipment or exterior walls and windows are natural convection. In a typical air-conditioned office with equipment or exterior walls, the airflow is mixed convection. Hence, it is desirable to develop a turbulence model that can predict accurately and effectively these three types of flow. This paper presents a two-layer turbulence model (Xu 1998) that performs two tasks:

1) The model can accurately predict indoor airflows under various conditions, i.e., from purely forced to purely natural convection airflows (Xu 1998). This model allows designers to use one single model to calculate indoor airflows instead of selecting different turbulence models empirically from many available models.
2) The model can be more efficient than available turbulence models, such as low-Reynolds-number k-ε models and Reynolds stress models.

Review of Previous Work

Purely forced and natural convection can be viewed as two extreme cases of mixed convection. Mixed convection is more complicated than forced convection and natural convection since it combines the complexity of both. Turbulent forced convection has been extensively studied and most turbulence models are developed for forced flows. This section reviews the state-of-art studies of turbulent mixed convection and identifies the problems with the existing turbulence models.

Studies on turbulent mixed convection fall into three categories: theoretical analysis, experimental investigation and numerical simulation. Theoretical studies include those by Nakajima *et al.* (1980), Chen *et al.* (1987) and Aicher and Martin (1997). Nakajima *et al.* (1980) and Chen *et al.* (1987) independently proposed damping functions similar to Van Driest's (1956) to enable the mixing-length model to count the buoyancy effects. They also conducted an experiment in a long channel to study basic turbulence and heat transfer characteristics of mixed convection. Aicher and Martin (1997) proposed two heat transfer correlations for the mixed convection in a tube. These studies provided some physical insights to understand the mixed convection. For example, Nakajima and Fukui (1980) found that the heat transfer in the boundary layer of a plate was decreased when the plate was slightly heated. Aicher and Martin (1997) suggested that the decrease is due to the reduction of the turbulent energy in the boundary layer.

Experimental investigations on turbulent mixed convection are very few. The only notable contributions are those by Schwenke (1975) and Blay *et al.* (1992). Schwenke (1975) conducted a series of measurements in a ventilated room with a heated wall. He measured the penetration length, which is the length of how far the air stream from a diffuser can propagate against the buoyancy-induced wall jet. He found that the energy imposed on the heated wall had a significant impact on the airflow pattern. This case can represent the airflow in a typical ventilated room with a hot window. Blay *et al.* (1992) measured the mixed convection flow in a two-dimensional cavity. Their measured quantities include velocities, temperatures and velocity fluctuations at mid-width and mid-height plane of the cavity.

Many researches have contributed to the numerical simulations of mixed convection. Nielsen *et al.* (1979) used the standard k-ε model with wall functions and calculated the flows in a ventilated room with floor heating. This early attempt showed the great potential of numerical simulation and it was considered a milestone in applying the computational fluid dynamics (CFD) technique to indoor airflow simulations.

Many other researchers have applied low-Reynolds-number k-ε models to calculate mixed convection flows in vertical pipes (Jackson *et al.* 1990), vertical channels (Inagaki and Komori, 1995; Fedorov *et al.*, 1997), vertical boundary layers (Patel *et al.* 1996), and cavities (To and Humphrey, 1986; Blay *et al.*, 1992). Their computational results have been compared with corresponding experimental data and the agreement has been reasonably good. However, the computing costs in those applications were very high. For example, To and Humphrey (1986) found that at least five grids are needed in the viscous sublayer and 17 in the buffer layer to produce acceptable results.

Chen (1995) systematically compared the performance of several k-ε models on indoor airflow simulation and found that the performance of various turbulence models was so diverse that no model can be used universally. For example, he found the RNG k-ε model performed best in mixed convection but very poor in forced convection flows. His study included a two-layer model from Rodi (1991). Since the model was developed for forced convection flow, it performed poorly for natural and mixed convection flows in rooms. Therefore, it is important to develop a model that could work for forced, natural, and mixed convection flows normally found indoors.

Very few researchers have attempted to apply the Reynolds-stress models on indoor airflow simulations. Chen (1996) found that the performance of the three Reynolds number model in mixed convection do not perform much better than the RNG k-ε models.

In summary, for turbulent mixed convection, few theoretical and experimental studies exist due to its complexity. Most numerical simulations on the mixed convection have employed various versions of the low-Reynolds-number k-ε models while few others have applied the Reynolds stress models. The computing efforts required by a low-Reynolds-number k-ε models or a Reynolds stress model are much higher than the standard k-ε model and RNG k-ε model (Xu and Chen 1998 and Chen 1996). An accurate and cost-effective turbulence model is not available.

Turbulence Model

In order to predict accurate mixed convection flow in a room with reasonable computing costs, Xu (1998) recently used the data of direct numerical simulation from (Kasagi and Nishimura, 1997) to develop a new two-layer model. In addition to the directional numerical simulation data, this new model is based on the two-layer model for forced convection flow (Rodi 1991) and another for natural convection flow (Xu et al.

1998). Hence, the model should work for forced, natural, and mixed convection flows found indoors. This section briefly introduces the model.

A two-layer model consists of two turbulence models. This paper uses a single k-equation turbulence model for near wall flow and the standard k-ε model for the flow in outer-wall region. The criteria to switch the model from one to the other is the y^* value. If $y^*<y^*_{\text{prescribed}}$ the single-equation model applies; otherwise, the standard k-ε model will be used. $y^*_{\text{prescribed}} = 80$ was found to be the best for room airflows.

In the near-wall region, where $y^*<80$, the new one equation model is used; i.e., the k is solved by Eq. (1), i.e.,

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = d_k + P_k + G_k - \varepsilon \tag{1}$$

The eddy viscosity is calculated by Eq. (2):

$$\nu_t = \sqrt{\overline{\upsilon\upsilon}}\, l_\mu \tag{2}$$

the ε by Eq. (3):

$$\varepsilon = \frac{\sqrt{\overline{\upsilon\upsilon}}\, k}{l_\varepsilon} \tag{3}$$

and l_μ, l_ε, and $\frac{\overline{\upsilon\upsilon}}{k}$ by Eqs. (4), (5) and (6) respectively.

$$l_\mu = \frac{(0.33 + 0.214 f_{l_\mu}) y}{1 + 5.025 \times 10^{-4}\, y_v^{*[1.53 + 0.12 f_{l_\mu}]}} \tag{4}$$

$$l_\varepsilon = \frac{(1.3 + 7.5 f_{l_\varepsilon}) y}{1 + \left(2.12 + 7.88 f_{l_\varepsilon}\right) / y_v^* + (0.028 + 0.0235 f_{l_\varepsilon}) y_v^*} \tag{5}$$

$$\frac{\overline{\upsilon\upsilon}}{k} = 0.4\left[1 - \exp\left(-\frac{y^{*(2 - f_{vv/k})}}{4200(1 - 0.99 f_{vv/k})}\right)\right] \tag{6}$$

where

$$f_{l_\mu} = f_{l_\varepsilon} = \frac{1}{2}\left[1 + \tanh(50 * \left|Ar_y\right| - 4)\right] \tag{7}$$

$$f_{vv/k} = \frac{1}{2}\left[1 + \tanh(120 * \left|Ar_y\right| - 4)\right] \tag{8}$$

2) In the outer-wall region, where $y^*\geq 80$, the standard k-ε model, Eqs. (9) and (10), are used:

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x_i}\left((\nu + \frac{\nu_t}{\sigma_k}) \frac{\partial k}{\partial x_i}\right) + \nu_t (\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}) - \varepsilon + g_i \beta \frac{\nu_t}{\mathrm{Pr}_t} \frac{\partial T}{\partial x_i} \tag{9}$$

where $\sigma_k = 1.0$. The ε solved by a transport equation

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}\left((\nu + \frac{\nu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_i}\right) + \frac{\varepsilon}{k}\left[C_{1\varepsilon} \nu_t (\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}) - C_{2\varepsilon}\varepsilon + C_{3\varepsilon} g_i \beta \frac{\nu_t}{\mathrm{Pr}_t} \frac{\partial T}{\partial x_i}\right] \tag{10}$$

The eddy viscosity is calculated by $\nu_t = C_\mu \frac{k^2}{\varepsilon}$, where C_μ = 0.09. The standard k-ε model does not need length scale prescription.

Simulation of Displacement Ventilation in a Room Using the Two-Layer Turbulence Model

The model used in this paper has been validated against many cases including: natural convection and forced convection in simplified rooms, etc. (Xu 1998). In those simple cases, the velocity and turbulence distributions are measured with anemometers with high precision, such as Laser Doppler Anemometers. However, it is also necessary to examine model performance in a scenario close to a real room, which a design engineer most likely deals with. Altough it is difficult to use high precision anemometers in a real room, a good model must be validated with data obtained from real rooms to evaluate its performance. .

There are many kinds of mixed convection flows in a room. This investigation selects displacement ventilation flow for the validation of the two-layer model. Displacement ventilation has attracted much interest in recent years because of its demonstrated capabilities to improve indoor air quality and reduce energy consumption in buildings (Yuan *et al.* 1998 and Hu *et al.* 1999). Prediction of the airflow in a room with displacement ventilation is more crucial than that with mixing ventilation because the displacement ventilation generates stratified air temperature and contaminant concentrations. The distributions of the air temperature and contaminant concentrations are very important information needed by a designer of the HVAC system. The experimental data of displacement ventilation from Yuan *et al.* (1999) have been used in the current study to verify the performance of the two-layer model. The following section describes the room airflow conditions with the displacement ventilation and computation methods used.

Fig. 1 shows the experimental setup. The experimental rig simulates a 5.16m × 3.65 m × 2.44m office occupied by two *simulated* persons and equipped with two tables, two heated computers, two file cabinets and six overhead lights. A 3.65m×1.04m window was mounted on the wall connecting the office room to an environmental chamber, where the temperature was controlled to simulate outdoor conditions. All other walls were insulated to minimize the heat gain/loss. The displacement ventilation system provided air for the room from a diffuser that was situated on the opposite wall of the window. The air was exhausted through the outlet at the ceiling. Internal heat flows were generated from the computers, occupants and lights. The *simulated* pollutants, SF_6 and CO_2, were discharged at occupants' head level.

The prediction solved the three-dimensional equations for mass, momentum, energy and concentration conservation on a non-uniform grid system: 52 × 48 × 28. This grid appears to be sufficient according to the grid independence study performed in other validation cases (Xu 1998). The boundary conditions and computation details are summarized in Table 1. It should be noted that on the enclosure surfaces, we used the measured wall temperatures as the boundary conditions, not empirical data used by Yuan *et al.* (1999).

Fig. 2 shows that the observed and calculated airflow patterns in the mid-width section via the displacement diffuser. Due to buoyancy, the cold air from the diffuser spreads on the floor level and induced a reverse flow in the layer between 0.5 to 1m. In addition, the hot window induced an upward flow close to it. Our prediction has precisely reproduced all these patterns observed by the smoke-visualization. With the smoke visualization technique, we can measure accurately the low velocity in the room through image analysis.

Fig. 3 shows the temperature profiles at nine locations in the simulated office. The agreement is very good in the region close to the ceiling. In the region close to the floor, some discrepancies occur but the largest difference between the calculated temperatures and measured ones is less than 1°C, about 4% of the measured temperature, which can be considered as a good agreement. The numerical simulation conducted by Yuan *et al.* (1999) used the RNG k-ε model with wall functions to predict the flow and heat transfer. Since wall functions would predict grid dependent heat flux a prescribed convective heat transfer coefficient was used to obtain the correct temperature distributions. The value of the coefficient used in their study was calibrated by the experimental data. This is undesirable in a numerical prediction because the coefficient is generally unknown. The two-layer model does not require prescribing a convective heat transfer coefficient and thus is more favorable for numerical simulations.

Fig. 4 compares the computed and the measured mean velocity profiles. The agreement is acceptable. . The hot-wire anemometers used in the experiment cannot measure accurately when the air velocity is lower than 0.1 to 0.15 m/s. Since the velocity level in this case is low, the uncertainty in the measurements of the velocities is rather high. The predicted velocities are still within the uncertainties of the experimental data.

Similarly, the turbulence quantities measured by the experiment has large uncertainties as well. Therefore, the comparison between experimental data and prediction is not very indicative but we still present it here anyway.. Fig. 5 shows the computed fluctuatingvelocity deviates significant from the measured data for the displacement ventilation. Chen (1995 and 1996) reported the same problems with other k-ε models and Reynolds stress models. Nevertheless, the computation does predict the trend of the flow turbulence in the room.

In another study (Xu 1998) where turbulence quantities were measured by a Laser Doppler Anemometer, the agreement between the computed results and the measured data is much better. This further confirms that the omni-directional hot sphere anemometers have a poor accuracy in measuring the velocity fluctuations.

The investigation uses a tracer-gas (SF_6) to simulate a contaminant gas, because the background concentration of SF_6 in atmosphere is very low. Fig. 6 shows the SF_6 concentration profiles. The overall agreement between the computed results and measured data is reasonably good but worse than that with other quantities. Yuan *et al.* (1999) also found that the concentration prediction was difficult to match the experimental data since it was very sensitive to the position and boundary conditions.

Nevertheless, the accuracy of the prediction in the lower part as well as the upper part of the zones 4, 5, 6 and 9 is still acceptable.

This study shows that the computing cost with the two-layer model ($52 \times 48 \times 28$) is slightly higher than that with the RNG k-ε model ($48 \times 42 \times 24$, Yuan *et al.* 1999), but significantly lower than that would be required by a low-Reynolds-number k-ε modelA low-Reynolds-number model would require at least a million grid cells and result in very poor grid aspect ratios because there are many heated surfaces in the room and each surface needs at least 20 grid cells to calculate heat transfercorrectly.The computing effort would be tremendous and prohibitive for practical engineering design

This paper compares the computed results by the two-layer model with the experimental data obtained in a real room. Since the data quality is not very high, it is difficult to identify whether discrepancies are from the model or from the experiment. However, Xu (1998) has used the two-layer model to predict two-dimensional flows with high quality data, the performance of the two-layer model is much better.

Conclusions

This paper briefly reviews the effort to predict mixed convection with computational fluid dynamics. With a two-layer turbulence model recently developed from the data of direct numerical simulation and two-layer models for natural and forced convection, it is possible to accurately predict mixed convection in a room.

Since displacement ventilation design requires accurate information of airflow,This study used the two-layer model to predict the airflow pattern and distributions of air temperature, air velocity, air velocity distributions, and contaminant concentration simulated by a tracer gas in a room with displacement ventilation. Designing such ventilation system often requires accurate information of airflow. The model can predict accurately the airflow pattern, air velocity, and air temperature. However, it is more difficult to predict the turbulence (velocity fluctuation) and contaminant concentration.

The computing resources required by the simulation with the two-layer model is slightly higher than that with the RNG k-ε model, but significantly lower than a low-Reynolds number k-ε model.

Acknowledgements

This study is supported by the National Science Foundations through grant CMS-9623864.

References

Aicher, T. and Martin, H. (1997) "New correlations for mixed turbulent natural and forced convection heat transfer in vertical tubes," *Int. J. Heat Mass Transfer*, **40(15)**, 3617-3626.

Blay, D., Mergui, S. and Niculae, C. (1992) "Confined turbulent mixed convection in the presence of a horizontal buoyant wall jet," *Fundamentals of Mixed Convection*, HTD, **213**, 65-72.

Chen, Q. (1995) "Comparison of different k-ε models for indoor air flow computations," *Numerical Heat Transfer*, Part B, **28**, 353-369.

Chen, Q. (1996) "Prediction of Room Air Motion by Reynolds-Stress Models," *Building and Environment*, **31**, 233-244.

Chen, T.S., Armaly, B.F. and Ali, M.M. (1987) "Turbulent mixed convection along a vertical plate," *J. Heat Transfer*, **109**, 251-253.

Fedorov, G.A., Viskanta, R. and Mohamad, A.A. (1997) "Turbulent heat and mass transfer in an asymmetrically heated vertical parallel-plate channel," *Int. J. Heat and Fluid Flow*, **18**, 307-315.

Hu, S., Chen, Q. and Glicksman, L.R. (1999) "Comparison of energy consumption between displacement and mixing ventilation systems for different U.S. buildings and climates," *ASHRAE Transactions*, **105(2)**.

Inagaki, T, and Komori, K. (1995) "Numerical modeling on turbulent transport with combined forced and natural convection between two vertical parallel plates," *Numerical Heat Transfer*, Part A, **27**, 417-431.

Jackson, J.D., Cotton, M.A., Yu, L.S.L. and Rouai, M.M. (1990) "Experimental and computational studies of turbulent forced and mixed convection heat transfer to water in a vertical pipe," In *Engineering Turbulence Modeling and Experiments* (Rodi W. and Ganic E.N. Editors), p809-818, Elsevier Science Publishing Co., Inc., New York.

Kasagi, N. and Nishimura, M. (1997) "Direct numerical simulation of combined forced and natural convection in a vertical plane channel," *Int. J. Heat and Fluid Flow*, **18**, 88-99.

Launder, B.E. and Spalding, D.B. (1974) "The numerical computation of turbulent flows," *Computer Methods in Applied Mechanics and Energy*, 3: 269-289.

Nakajima, M. and Fukui, K. (1980) "Buoyancy effects on turbulent transport in combined free and forced convection between vertical parallel plates," *Int. J. Heat Mass Transfer*, **23(10)**, 1325-1336.

Nielsen, P.V., Restivo, A. and Whitelaw, J.H. (1979) "Buoyancy-affected flows in ventilated rooms," *Numerical Heat Transfer*, **2**, 115-127.

Patel, K., Armaly, B.F. and Chen, T.S. (1996) "Transition from turbulent natural to turbulent forced convection adjacent to an isothermal vertical plate," HTD, **324**, 51-56.

Rodi, W. (1991). “Experience with two-layer models combining the k-ε model with a one-equation model near the wall,” AIAA-91-0216, 29th Aerospace Sciences Meeting, Nevada.

Schwenke, H. (1975) “Ueber das Verhalten elener horizontaler Zuluftstrahlen im begrenzten Raum,” *Luft- und Kaltetechnik*, **5**, 241-246.

To W.M. and Humphrey, J.A.C. (1986) “Numerical simulation of turbulent buoyancy, turbulent flow—I. Free convection along a heated, vertical, flat plate,” *Int. J. Heat Mass Transfer*, **29**, 573-592.

Van Driest E.R. (1956) “On turbulent flow near a wall,” *Journal of the Aeronautical Science*, **23**, 1007

Xu, W. (1998) “New turbulence models for indoor airflow simulation,” Ph.D. Thesis, Department of Architecture, Massachusetts Institute of Technology, Cambridge, MA, USA.

Xu, W. and Chen, Q. (1998) “Numerical simulation of air flow in a room with differentially heated vertical walls,” *ASHRAE Transactions*, **104(1)**, 168-175.Xu, W., Chen, Q. and Nieuwstadt, F.T.M. (1998). “A new turbulence model for near-wall natural convection,” *Int. J. Heat and Mass Transfer*, **41**, 3161-3176.

Yuan, X., Chen, Q. and Glicksman, L.R. (1998) “A critical review on displacement ventilation,” *ASHRAE Transactions*, **104(1)**, 78-90.

Yuan, X., Chen, Q., Glicksman, L.R., Hu, Y. and X. Yang. (1999) “Measurements and computations of room airflow with displacement ventilation,” *ASHRAE Transactions*, **105(1)**, 340-352.

Table 1 Boundary conditions and computation details

	P	U	V	W	T	k	ε	C
B. C.[1] at inlet	-[2]	U_{in}	0	0	T_{in}	$3\times10^{-2}U_{in}^2$	$\frac{0.09k_{in}^{3/2}}{0.55l_{in}}$	C_{in}
B. C.[1] at outlet	0	$\frac{\partial U}{\partial z}=0$	$\frac{\partial V}{\partial z}=0$	$\frac{\partial W}{\partial z}=0$	$\frac{\partial T}{\partial z}=0$	$\frac{\partial k}{\partial z}=0$	$\frac{\partial \varepsilon}{\partial z}=0$	$\frac{\partial C}{\partial z}=0$
B. C.[1] at Walls	-[2]	0	0	0	[3]	0	Eq. (3)	Constant flux for human top surfaces, otherwise no flux
Relaxation method	Linear	False-time-step	False-time-step	False-time-step	False-time-step	False-time-step	False-time-step	False-time-step
Factor	0.5	0.1	0.1	0.1	0.5	0.001	0.001	0.1

[1]Boundary Conditions.
[2]No need to specify.
[3]Fixed temperature for the enclosure surfaces, constant heat flux for the surfaces of the computer, occupants and lights; no-flux condition for the other surfaces.

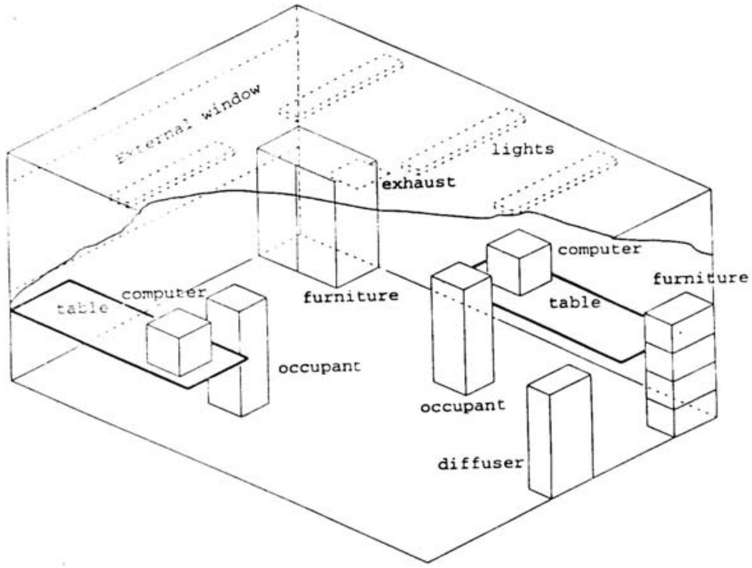
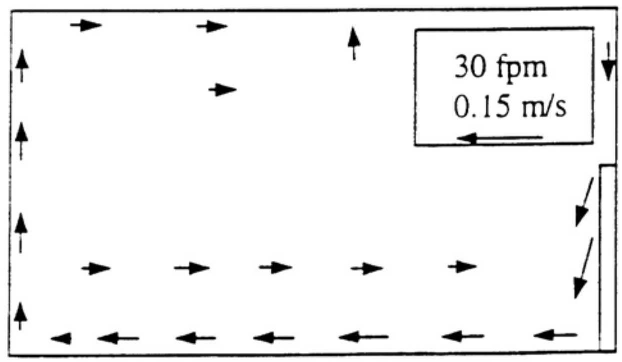
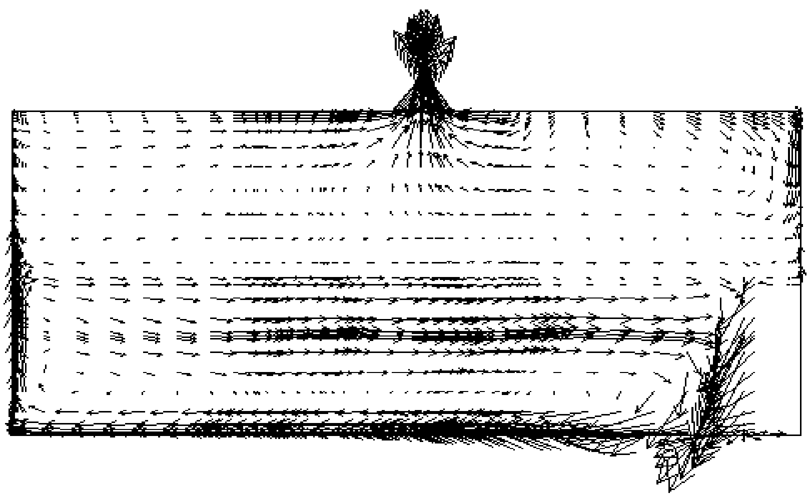


Fig. 1 Configuration of the simulated office.



(a) Observed



(b) Predicted

Fig. 2 The observed and predicted airflow pattern at the mid-width section via the displacement diffuser.