Original Paper

Indoor+Built Environment

Indoor Built Environ 1997;6:140–149

Accepted: February 1, 1997

Qingyan Chen
Nien-Tzu Chao

Department of Architecture,
Massachusetts Institute of Technology,
Cambridge, Mass., USA

Comparing Turbulence Models for Buoyant Plume and Displacement Ventilation Simulation

Key Words
Turbulence model
Computational fluid dynamics
Thermal plume
Displacement ventilation

Abstract
Computational fluid dynamics may be used to predict the details of airflow in rooms served by displacement ventilation systems, provided a suitable turbulence model can be found. Since buoyant plumes are central to the displacement ventilation strategy, four turbulence models – three eddy-viscosity models (the 'standard' k-ε model, a modified k-ε model, and an RNG k-ε model) and the Reynolds stress model – were applied to simulate airflow in a turbulent buoyant plume. Corresponding experimental data from the literature were used for validation, although for a plume stronger than expected in rooms as no reliable plume data for room air flow were found. The Reynolds stress model predicted velocity, temperature, and turbulence quantities satisfactorily while the eddy-viscosity models performed poorly. The eddy-viscosity models were then applied to predict airflow in a furnished room with displacement ventilation. The computed airflow patterns, mean velocities, temperatures, and contaminant concentrations agree reasonably well with the experimental data obtained from a full-scale test chamber, but the discrepancies in some locations were large.

Introduction

To remove indoor pollutants efficiently, a displacement ventilation system, shown in figure 1, has been used in Europe, especially in Scandinavian countries. Recently, it has attracted considerable attention in the US and the rest of the world, as evidenced by the numerous papers in conferences of Roomvent '96 [1] and Indoor Air '96 [2] conferences.

In the displacement ventilation system, colder fresh air is supplied into the lower part of a space. Heat sources in the space, such as occupants and equipment, generate thermal plumes that transport contaminated air from the occupied zone to the upper part of the space from where it is extracted. This method can give a better indoor air quality than the traditional mixed ventilation strategies, because the fresh air is supplied directly to the occupied zone. On the other hand, the displacement ventilation strategy tends to increase vertical temperature stratification that can decrease the thermal comfort of the occupants and may impact on the energy consumption of the HVAC systems.

KARGER
E-Mail karger@karger.ch
Fax + 41 61 306 12 34
http://www.karger.ch

1420–326X/97/0063–0140$12.00/0

Prof. Qingyan (Yan) Chen
Department of Architecture, Massachusetts Institute of Technology, Room 5-418
77 Massachusetts Avenue, Cambridge, MA 02139-4307 (USA)
Tel. (617) 253-7714, Fax (617) 253-6152
E-Mail QChen@MIT.EDU, http://web.mit.edu/qchen/www/

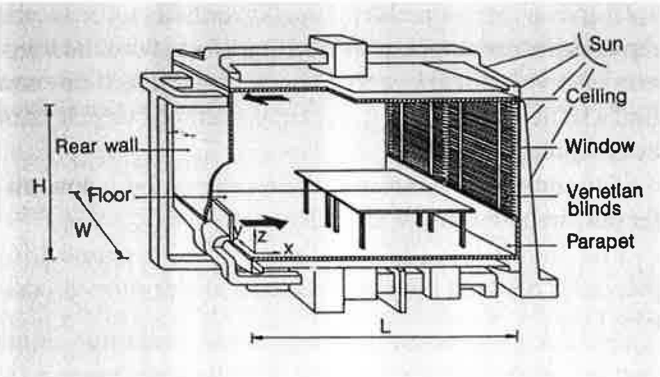

Fig. 1. Sketch of the room with displacement ventilation. (In this laboratory facility, two separated HVAC systems are used to condition the small spaces above and below the room to control the room ceiling and floor temperature).

Both experimental measurements [3] and computer simulation [4] may be used to study the feasibility of displacement systems for US buildings. Since measurements are often expensive, there is a need for a computational tool that can predict the airflow pattern, and distributions of temperature and contaminant concentrations in a displacement ventilation system. One such computational tool, the computational fluid dynamics (CFD) technique, solves the Navier-Stokes equations using turbulence models for time-averaged Reynolds stresses and heat fluxes. This paper compares several turbulence models for predicting a plume flow and flow in a room with displacement ventilation.

Research Methods

The Navier-Stokes equations, with appropriate boundary conditions, describe the conservation laws controlling airflow, temperature, and pollutant transport in a space. However, even with the present capacity and speed of available super computers, it is still not possible to solve room air flow problems of practical importance. This is because numerical solutions of these equations require temporal and spatial discretization small enough to catch the smallest eddies in the flow. Although the details of turbulent flow are difficult to calculate, only the mean values of the flow variables are of primary interest. Averaging the Navier-Stokes equations over time, and using turbulence models to approximate the Reynolds stress and heat flux terms, allows a solution within the present capacity and power of computers.

Considerable success has been achieved by using the CFD technique for a number of problems concerning the airflow and air quality in buildings. However, there are still some uncertainties in the numerical simulation [5]. Experimental validation of the numerical simulation is always needed. In this study, the numerical simulations using different turbulence models are validated with the experimental data in order to search for a suitable turbulence model for displacement ventilation.

Most turbulence models were developed for specific flows. A turbulence model may work well for one case and poorly for another. At present, the 'standard' k-ε model [6] is still the most popular one for predicting room airflows. This model works reasonably well in many cases, but there are many problems associated with specific applications. For example, the model has difficulties in predicting turbulence in room airflows and heat transfer in boundary layers [7]. New turbulence models, developed in the past two decades that better predict turbulence in different flows, should be tested for room airflows.

Recently, we have used five eddy-viscosity models and three Reynolds stress models to simulate natural convection, forced convection, and mixed convection in rooms, as well as impinging jet flows [8, 9]. No universal model has been found for indoor airflow simulation. A suitable turbulence model should be identified for each type of flow. Among the models tested, the Reynolds stress models perform better than the eddy-viscosity models (i.e. for the problems considered), and the renormalization group (RNG) of the k-ε model [10] is the best among the eddy-viscosity models. Therefore, one of the Reynolds stress models [11] and the RNG model were selected for the present study. Since the standard k-ε model has been widely used, it was also used in this study as a reference.

This section presents the mathematical models used to describe turbulence in the flows. All the turbulence models supplement the basic relationships describing any flow. For steady, high Reynolds number, incompressible, and buoyant flow, the time-averaged Navier-Stokes equations, written in Cartesian tensor form, become [11]:

$$(\rho U_i)_{,i} = 0$$

$$(\rho U_j U_i)_{,j} = -P_{,i} - (\rho\overline{u_i u_j})_{,j} + (\rho - \rho_r) g_i$$

$$(\rho U_j T)_{,j} = -(\rho\overline{u_i t'})_{,j}$$

$$\rho = \rho_r T_r / T,$$

where $\overline{u_i u_j}$ and $\overline{u_j t'}$ are, respectively, the unknown Reynolds stresses and heat fluxes. The terms $(\overline{u_i u_j})$ and $(\overline{u_j t'})$

are higher-order correlations that must be approximated by model assumptions in order to close the system of equations. The equations used to approximate these terms are called turbulence models, such as the commonly used eddy-viscosity models and Reynolds-stress models.

Eddy-Viscosity Turbulence Models

Eddy-viscosity models use the Buossinesq eddy-viscosity assumption to determine the Reynolds stresses $\overline{u_i u_j}$ and heat fluxes $\overline{u_j t'}$ by:

$$\overline{u_i u_j} = -\nu_t (U_{i,j} + U_{j,i}) + \frac{2}{3} k$$

$$\overline{u_j t'} = -\frac{\nu_t}{\sigma_T} T_{,j}$$

where the turbulent Prandtl number $\sigma_T = 0.9$. The above approach connects Reynolds stress to the mean velocities and turbulent kinetic energy, k, using eddy-viscosity, ν_t. The eddy-viscosity is an artificial flow parameter rather than a measurable fluid property. The Buossinesq approximation does not have a solid theoretical background. In addition, it assumes turbulence to be isotropic which may not be true in room airflows. The models derived from the assumption can be problematic. Nevertheless, the model replaces the instantaneous velocities, $U + u$, with the time-averaged velocities, U, and a kinetic energy, k. If one can find a solution for k and ν_t, the time-averaged velocity can be computed directly.

The three eddy-viscosity models presented below differ only in how k and ν_t are calculated.

The Standard k-ε Model. The standard k-ε model [6] (hereafter denoted *k-ε model*) calculates the eddy viscosity ν_t from:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}$$

where $C_\mu = 0.09$. The following transport equation determines the kinetic energy of turbulence, k:

$$(\rho U_i k)_{,i} = \left(\rho \frac{\nu_t}{\sigma_k} k_{,i}\right)_{,i} + \rho(P_k + G_k - \varepsilon)$$

The above equations introduce a new parameter – the dissipation rate of kinetic energy, ε, solved by another transport equation:

$$(\rho U_i \varepsilon)_{,i} = \left(\rho \frac{\nu_t}{\sigma_\varepsilon} \varepsilon_{,i}\right)_{,i} + \rho \frac{\varepsilon}{k}(C_{1\varepsilon} P_k + C_{3\varepsilon} G_k - C_{2\varepsilon} \varepsilon)$$

Nomenclature

B_o	source buoyancy flux of the thermal plume
C's	coefficients in turbulence models
d	diameter
f	buoyant source term in the momentum equation
Fr	source Froude number
g	gravity
g_i	component i of the gravitation vector
G_{ij}	buoyancy production of $\overline{u_i u_j}$
G_{it}	buoyancy production of $\overline{u_i t'}$
G_k	buoyancy production of k
k	turbulent kinetic energy
l_m	Morton length scale
M_o	source-specific momentum flux of the thermal plume
P	static pressure
P_{ij}	stress production of $\overline{u_i u_j}$
P_k	stress production of k
r	coordinate in radial direction
R	time-scale ratio, source term in the RNG model
t'	fluctuation temperature
$\overline{t'^2}$	mean square temperature fluctuations
T	mean temperature
T_r	mean reference temperature
u	velocity fluctuation in the x-direction/radial velocity fluctuation
$\overline{u_i u_j}$	Reynolds stresses
$\overline{u_j t'}$	heat fluxes
U	mean velocity in the x-direction/radial velocity
U_i, U_j	mean velocity in the x_i- and x_j-direction
v	velocity fluctuation in the y-direction/tangential velocity fluctuation
V	mean velocity in the y-direction/tangential velocity
w	velocity fluctuation in the z-direction/streamwise velocity fluctuation
W	mean velocity in the z-direction/streamwise velocity
x, y, z	coordinates

Greek Symbols

β	volumetric expansion coefficient
δ_{ij}	Kronecker delta
ε	dissipation of k
ε_{ij}	viscous dissipation
ν_l	laminar viscosity
ν_t	turbulent viscosity
ρ	air density
σ's	diffusion coefficients in turbulence models
ϕ_{ij}	pressure redistribution of $\overline{u_i u_j}$
ϕ_{it}	pressure redistribution of $\overline{u_i t'}$

Superscripts

′	fluctuating quantities
—	mean quantities

Subscripts

i, j	spatial coordinates
o	plume source value
r	reference

where the model coefficients are taken as (σ_k, σ_ε, $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$) = (1.0, 1.314, 1.44, 1.92, 1.44). The shear production term, P_k, and the buoyancy production term, G_k, are defined by

$$P_k = \nu_t(U_{i,j} + U_{j,i})U_{i,j}$$

$$G_k = -\nu_t g_i \frac{\rho_{,i}}{\rho \sigma_T}$$

The RNG k-ε Model. The k-ε model uses constant model coefficients determined from a set of experiments for simple turbulent flows. Although this set of coefficients has a broad applicability, they are not universal. Yakhot and Orszag [10] derived a k-ε model based on RNG methods. Their approach represents the effects of small-scale velocity fluctuations by a universal random force, chosen to give the resulting flow field the same global properties as those in a flow driven by the mean strain. This RNG k-ε model (hereafter denoted *the RNG model*) has the same form as the k-ε model, except that all the model coefficients are assumed to have different values. The model coefficients in the RNG model (σ_k, σ_ε, $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ) = (0.7194, 0.7194, 1.42, 1.68, 0.0845).

The dissipation-rate transport equation

$$(\rho U_i \varepsilon)_{,i} = \left(\rho \frac{\nu_t}{\sigma_\varepsilon} \varepsilon_{,i}\right)_{,i} + \rho \frac{\varepsilon}{k}(C_{1\varepsilon} P_k + C_{3\varepsilon} G_k - C_{2\varepsilon}\varepsilon) + R$$

has an additional source term R in the right side.

$$R = \frac{C_\mu \eta^3 (1 - \eta/\eta_o)}{1 + \beta\eta^3} \frac{\varepsilon^2}{k}$$

where $\eta_o = 4.8$, $\beta = 0.012$ and the dimensionless parameter, η, is defined by

$$\eta = S\frac{k}{\varepsilon}, \quad S = (2S_{ij}S_{ij})^{1/2}, \quad S_{ij} = \frac{1}{2}(U_{i,j} + U_{j,i})$$

The Modified k-ε Model. Marlin [13] pointed out that two-equation models produce excessive spreading of axisymmetric jets and hence, forced plumes. The deficiency can be corrected by using different values of C_μ and $C_{2\varepsilon}$. They suggested using $C_\mu = 0.067$ and $C_{2\varepsilon} = 1.87$. Hereafter this model is denoted as the k-ε' model.

The Differential Reynolds Stress Turbulence Model

The Reynolds stress turbulence model (RSTM) solves additional transport equations to find the Reynolds stresses ($\overline{u_i u_j}$) and heat fluxes ($\overline{u_j t'}$) without using the problematic Buossinesq eddy-viscosity assumption. The formulation of the model presented here is from Marlin and Younis [10].

Reynolds Stresses. The following transport equation obtains the Reynolds stresses, $\overline{u_i u_j}$:

$$(\rho U_k \overline{u_i u_j})_{,k} = \rho(d_{ijk} + P_{ji} + G_{ij} + \phi_{ij} - \varepsilon_{ij})$$

where the term on the left-hand side of the equation is the convective transport, d_{ijk} stands for diffusion of Reynolds stress, P_{ij} is the shear production of Reynolds stresses, G_{ij} is the production due to buoyancy, ϕ_{ij} controls the redistribution of turbulent energy among the normal stresses (pressure-strain), and ε_{ij} represents viscous dissipation. Unfortunately, the terms d_{ijk}, ϕ_{ij} and ε_{ij} are unknowns and need to be modeled if the Reynolds stresses and heat fluxes are to be closed at the second-moment level. Therefore, the Reynolds stress models are much more complicated than the eddy-viscosity models. However, the Reynolds-stress models do not use the eddy-viscosity concept and do not assume turbulence to be isotropic.

The present investigation models d_{ijk} by

$$d_{ijk} = C_s\left(\frac{k}{\varepsilon} \overline{u_k u_l} (\overline{u_i u_j})_{,l}\right)_{,k}$$

where $C_s = 0.22$. The P_{ij} and G_{ij} need no approximation and are defined by

$$P_{ij} = -(\overline{u_i u_k}\, U_{j,k} + \overline{u_j u_k}\, U_{i,k})$$

$$G_{ij} = -\beta(g_i \overline{u_j t'} + g_j \overline{u_i t})$$

where β is the volumetric coefficient of expansion. The pressure-strain term ϕ_{ij} can be further decomposed into purely turbulent interactions ϕ_{ij1}, interactions between mean strain and fluctuating velocities ϕ_{ij2}, and buoyancy forces ϕ_{ij3}, as follows:

$$\phi_{ij} = \phi_{ij1} + \phi_{ij2} + \phi_{ij3}$$

Each term on the right hand side of the equation is modeled individually by the following equations:

$$\phi_{ij1} = -C_1 \frac{\varepsilon}{k}\left(\overline{u_i u_j} - \frac{2}{3}\delta_{ij} k\right)$$

$$\phi_{ij2} = -C_2\left(P_{ij} - \frac{1}{3}\delta_{ij} P_{kk}\right)$$

$$\phi_{ij3} = -C_3\left(G_{ij} - \frac{1}{3}\delta_{ij} G_{kk}\right)$$

where (C_1, C_2, C_3) = (3.0, 0.3, 0.3). The viscous dissipation term ε_{ij} is modeled by

$$\varepsilon_{ij} = \frac{2}{3}\delta_{ij}\varepsilon$$

where ε is the dissipation rate of turbulent energy.

Dissipation Rate of Turbulent Energy. The ε is determined by the following transport equation:

$$(\rho U_k \varepsilon)_{,k} = C_\varepsilon\left(\rho \frac{k}{\varepsilon} \overline{u_k u_l} \varepsilon_{,l}\right)_{,k} + \rho \frac{\varepsilon}{k}\frac{1}{2}(C_{1\varepsilon} P_{kk} + C_{3\varepsilon} G_{kk} - 2C_{2\varepsilon}\varepsilon)$$

where $(C_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}) = (0.15, 1.4, 1.8, 0.98)$. The first term on the right side of the equation accounts for diffusion by turbulent motion. The remaining term accounts for the difference between the production and dissipation of turbulent energy.

Turbulent heat fluxes. The Reynolds stress model calculates the turbulent heat fluxes, $\overline{u_i t'}$, by

$$(\rho U_k \overline{u_i t'})_{,k} =$$

$$C_t\left(\rho \frac{k}{\varepsilon} \overline{u_k u_l}(\overline{u_i t'})_{,l}\right)_{,k} + \rho(-\overline{u_k u_i} T_{,k} - \overline{u_k t'} U_{i,k} - \beta g_i \overline{t'^2} + \phi_{it})$$

where $C_t = 0.15$. The pressure-temperature gradient correlation ϕ_{it} corresponds to the pressure-strain term in the Reynolds stress equation. It is the sum of three contributions of purely turbulent interactions, interactions between mean strain and fluctuating quantities, and buoyancy forces. The Reynolds stress model uses the following equation for ϕ_{it}:

$$\phi_{it} = -C_{1t} \frac{\varepsilon}{k} \overline{u_i t'} + C_{2t}(\overline{u_k t'} U_{i,k}) + C_{3t} \beta g_i \overline{t'^2}$$

where $(C_{1t}, C_{2t}, C_{3t}) = (2.85, 0.55, 0.55)$.

Temperature Fluctuations. The transport equation of the temperature fluctuations is:

$$(\rho U_k \overline{t'^2})_{,k} = C_\theta\left(\rho \frac{k}{\varepsilon} \overline{u_k u_l}\, \overline{t'^2}_{,l}\right)_{,k} + \rho\left(-2\overline{u_j t'} T_{,j} - 2\frac{\overline{t'^2}\varepsilon}{2kR}\right)$$

where $R = 0.56$ is the time scale ratio.

Note that all the models use many model coefficients to close the system of the equations in order to make them solvable. All the model coefficients drive from experimental data of simple flows. The flows in practice are more complicated, hence the values used for these empirical coefficients may be inappropriate and may be expected to lead to discrepancies between computed and measured results.

Numerical Solution

The computations were carried out using PHOENICS [14], a commercial CFD code used by many ventilation engineers. This code has several routines accessible to users, allowing them to check and modify the models as they wish. The governing equations were solved using the finite-volume method in a staggered grid system. The hybrid scheme was used for the numerical solution. The algorithm employed was SIMPLEST. As a convergence criterion, the sum of the normalized absolute residuals in each control volume for all the variables was controlled to be less than 10^{-3} of the mass inflow.

Turbulent Buoyant Plume

In displacement ventilation, flows are driven by the supply air through a ventilation diffuser, and by the buoyancy effect from heat sources. The flows driven by the heat sources are turbulent buoyant plumes. Validation of the turbulence models for buoyant plumes is therefore necessary. Since turbulent buoyant plumes have simple geometry and have been studied in other areas of research, such as studies of spreading of smoke and other pollutants in the atmosphere and dispersal of volcano exhaust, many high-quality experimental data sets are available.

For room airflow studies, spatially detailed data are required for validation of turbulence models for:

- mean velocities
- mean temperatures
- Reynolds stresses
- heat fluxes

For the study of displacement ventilation, it is best to use the plume data obtained from a room with a displacement ventilation system. Kofoed [15] conducted many measurements under such conditions. He derived a correlation useful for design in practice. However, he primarily measured mean velocities and temperatures, and included little information on turbulence. To the best of our knowledge, there are no detailed experimental data available for rooms with displacement ventilation.

For these reasons, this study used the experimental data of buoyant flows in an unstratified environment. This study uses the data from Shabbir and George [16]. The experiment controlled the ambient air to be at rest in order to eliminate the impact of the ambient flow on the plume. This set of data contains most detailed information, such as mean velocity and temperature as well as Reynolds stresses and heat fluxes. Since hot-wire anemometers were used, the data accuracy normally is not very good. However, in the turbulent plume considered, the measuring error from equipment should be acceptable because most of the air velocities used for validation were higher than $0.2\ m \cdot s^{-1}$. Reviewing a number of experimental data, Dai et al. [17] and Shabbir and George [16] found there are discrepancies among measured data for turbulent buoyant plumes. This may be attributed to different measuring techniques and instrumentation used. In addition, the buoyant plumes studied are also different.

In the experiment [16], the plume time-averaged air velocities shown in figure 2 were created by a source with a velocity of $0.98\ m \cdot s^{-1}$ and an air temperature of 295 °C.

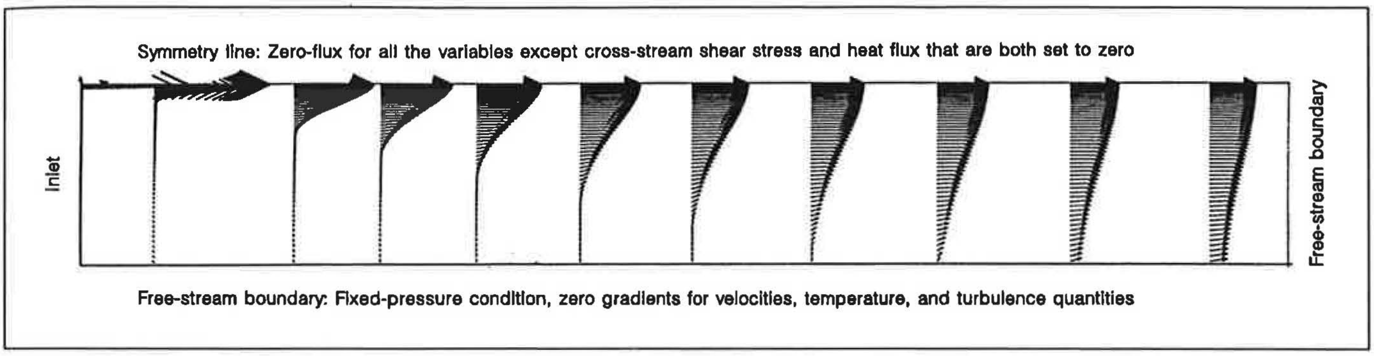

Fig. 2. Sketch of the plume and boundary conditions used in the computations.

The gravity acceleration is opposite to the jet inlet. The diameter of the supply jet was 0.0635 m. The ambient air temperature was 25 °C. Although the velocity and temperature of the plume source are much higher than a plume found in a room, the flow characteristics are similar to those in a room. For example, self-similarity is observed in both plumes. This flow corresponded to a source momentum flux, M_o, of 0.003 $m^4 \cdot s^{-2}$ and a source buoyancy flux, B_o, of 0.127 $m^4 \cdot s^{-3}$. The M_o and B_o are defined as:

$$M_o = (\pi/4)d^2 W_o^2$$

$$B_o = (\pi/4)d^2 W_o g|\rho_o - \rho_\infty|/\rho_\infty$$

The plume is indeed thermal dominant. In room airflows, the plume is weaker and the velocity is lower.

The turbulent buoyant plume is symmetrical about the jet axis, so a zero-flux boundary condition was used along the line of symmetry for all the variables except the cross-stream shear stress and heat flux. These were set to zero in the Reynolds stress model. At free-stream boundaries, a fixed-pressure condition was used, and zero gradients were used for the streamwise velocities, temperatures, and all turbulent quantities. The inlet velocity profile was assumed uniform. The turbulence intensity was estimated to be 0.5%, according to many similar types of jet flows.

How a source of momentum and buoyancy evolves into a plume can be characterized by the length scale l_m given by Shabbir and George [16]:

$$l_m = M_o^{3/4}/B_o^{1/2} = 0.114 \text{ m}$$

The ratio, l_m/d, is proportional to the source Froude number, defined as follows [17]:

$$Fr_o = (4/\pi)^{1/4} l_m/d$$

The source Froude number is a convenient measure of the buoyancy dominance at the source, e.g., $Fr_o = 0$ and infinity are for purely buoyant and for purely nonbuoyant sources, respectively. Buoyancy-dominated conditions for mean and fluctuating quantities are reached when z/l_m is greater than around 6–14 [17].

Numerous experiments reviewed by Dai et al. [17] show that thermal plumes are self-similar. The self-preserving conditions for mean and fluctuating variables are achieved when z/d is greater than 10 [16]. The self-similar phenomenon, which means the velocity and temperature profiles are similar in down stream, was also obtained from the computations presented in this paper. The results presented in this section are those in the self-similar region of the buoyant plume.

Within the self-preserving region, radial profiles of mean streamwise velocities (W) can be reasonably approximated by a Gaussian fit as follows [17]:

$$W(z/B_o)^{1/3} = A_w \exp[-B_w(r/z)^2]$$

The best fit of the experimental data [16] in the self-preserving region yielded $A_w = 3.4$ and $B_w = 60$. Dai et al. [17] and Shabbir and George [16] reviewed a number of experimental data and found that A_w is in the range of 3.4–4.7 and B_w of 48–93. In most cases, A_w is 3.4 and B_w from 50–60. The experimental data selected reflect these typical values. Table 1 compares the A_w and B_w with the predicted values.

A similar method approximates the radial profiles of temperature within the self-preserving region:

$$g\beta\Delta T z(z/B_o)^{2/3} = A_T \exp[-B_T(r/z)^2]$$

where A_T and B_T are also compared with the computed results in table 1.

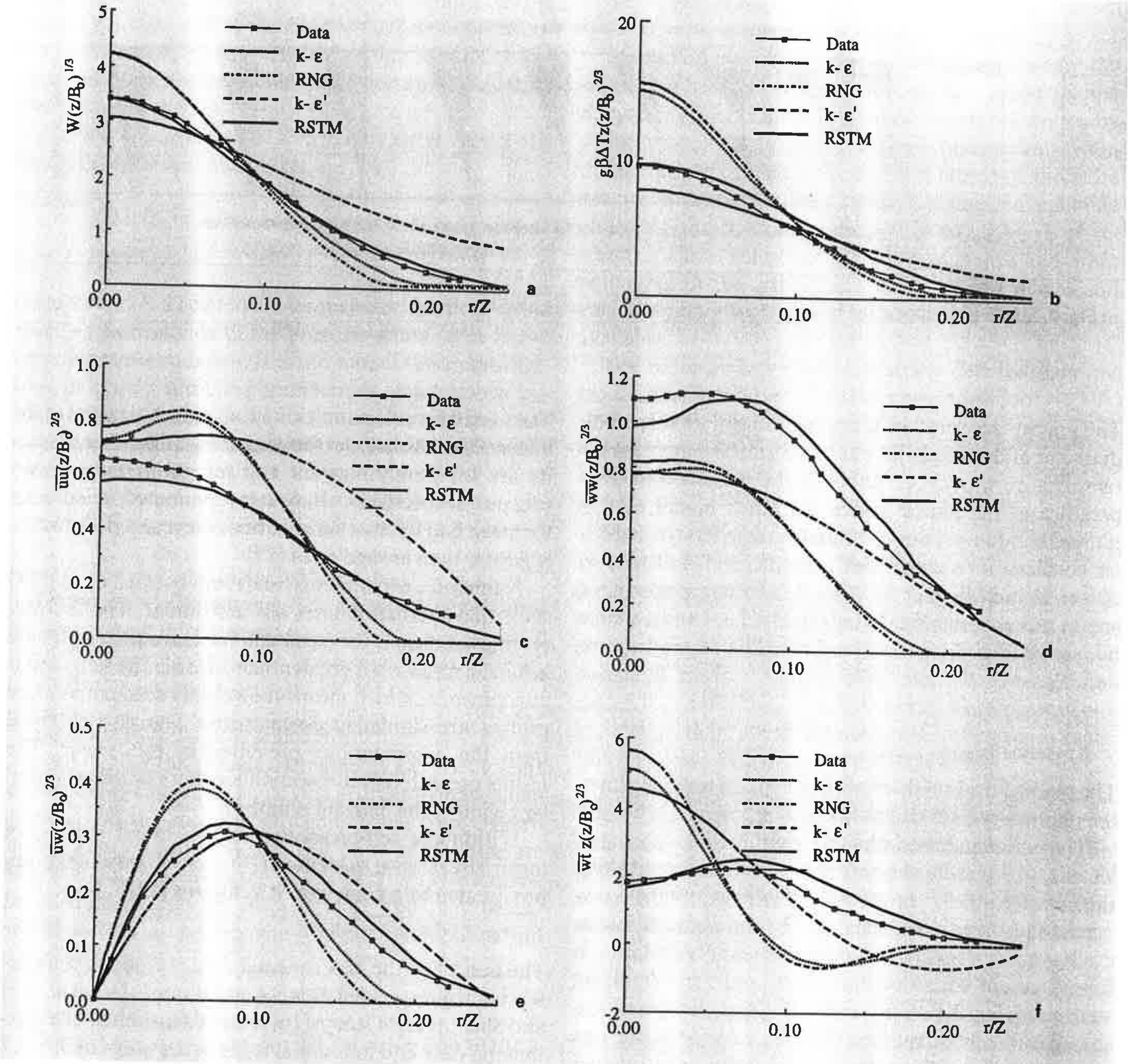

Fig. 3. Computed profiles of the plume and comparison with the experimental data from Shabbir and George [16]. **a** Streamwise mean velocity. **b** Temperature. **c** Radial velocity fluctuation. **d** Streamwise velocity fluctuation. **e** Cross-stream shear stress. **f** Streamwise turbulent heat flux.

Table 1. Summary of velocity and temperature profiles of the self-preserving buoyant plume

	A_w	B_w	A_T	B_T
Data [16]	3.4	60	9.4	68
k-ε	4.2	79	14.9	105
RNG	4.2	78	15.5	104
k-ε'	3.1	28	7.71	40
RSTM	3.4	61	9.6	51

Figure 3 shows the radial profiles of the buoyant plume computed by the four turbulence models, and their comparison with the experimental data [16]. Figure 3a illustrates the profiles of streamwise velocity. The results by the RSTM are in good agreement with the experimental data. The performance of the k-ε and RNG models is similar. They yield narrower profiles with larger values near the axis ($r/z = 0$) than the experimental data. In contrast, the k-ε' model predicts a wider profile.

Similar results are found for the temperature as shown in figure 3b. The defects of the eddy-viscosity models are a consequence of the poor prediction of the Reynolds stresses and heat fluxes, as shown in figures 3c–f. The poor performance of eddy-viscosity models was also found in other flows within a room [8, 9].

Comparing the predictions by the k-ε and k-ε' models, it seems possible to adjust the coefficients in the turbulence models to achieve a better agreement between the computations and the measured data. However, this type of fine tuning is of little use in practice. Only a set of prescribed coefficients is acceptable. In addition, the experimental data show the turbulence to be anisotropic. This is well captured by the RSTM. However, none of the eddy-viscosity models can predict the anisotropic turbulence because of their assumption of isotropic turbulence.

Displacement Ventilation

The second step of validating turbulence models is to calculate the airflow pattern and the distributions of air velocity, air temperature, and contaminant concentrations in a room with a displacement ventilation system. Ideally, all four models used in the previous section should be tested for room flows with displacement ventilation. Unfortunately, the RSTM in the current version of the PHOENICS code does not work for the flow in a room with obstacles, such as the tables in the room shown in figure 1. The RSTM has been applied to predict room airflows without obstacles [8]. In general, the RSTM performed better than eddy-viscosity models but used two to three times more computing time. At present, eddy-viscosity models are still widely used by engineers because of their simplicity. A good eddy-viscosity model can give reliable results with little computing effort. Hence, it is valuable to test the performance of different eddy-viscosity models for displacement ventilation.

This section presents the computed and measured flow results in a room with a displacement ventilation system. In the experimental setup, the test room, 5.6 m long, 3.0 m wide, and 3.2 m high, had two tables as shown in figure 1. A convective heat source of 530 W on the venetian blinds was used to simulate a summer cooling condition. The supply airflow rate was 5 air changes per hour (ACH). The supply air temperature was 19 °C. A helium source was introduced to a box near the table as a tracer gas to simulate contaminant from the occupant, such as CO_2 or tobacco smoke. Since helium is much lighter than the air and the helium concentration was relatively high in the room (0.5%), only a 25-watt lamp was used to simulate a person sitting next to the tables. Although the heat strength was considerably lower than that generated from an occupant, the combined buoyant effect from the mass source (helium) and the thermal source (heat from the lamp) is equivalent to a human body (about 75 W).

In the computations, the buoyant sources are normally set with the Buossinesq approximation of constant air density in the momentum equations:

$$f = \rho_r \beta (T_r - T) g$$

The temperature difference is caused by heat gains and losses in the room. The present study uses the method of variable air density. The momentum equations use the following term for the buoyant sources due to helium density and heat gains and losses:

$$f = (\rho - \rho_r) g$$

In addition, the three turbulence models employed nonequilibrium wall functions [6] for the rigid walls of the room. The inlet velocity was again assumed to be uniform and the turbulent intensity 10%. The outlet boundary condition was zero-pressure and zero-gradient for other variables.

Figure 4 shows the computed sectional distributions of air velocity, temperature, and helium concentration in the room by the RNG model. The computed airflow pattern is very similar to that of smoke. However, it is difficult to compare the computed results by different models with the measured data in the form presented in figure 4, because of too few experimental data available. Figure 5 illustrates the comparison in specific locations. The horizontal axis presents the room height. The results are shown only in the center of the room (x = 2.8 m and y = 1.5 m) for velocities and temperatures, and at a location near the center (x = 2.8 m and y = 0.8 m) for helium concentration where measured data are available.

In general, the computed results are in good agreement with the measured data, except in the regions near the floor and ceiling. It should be noted that the measured

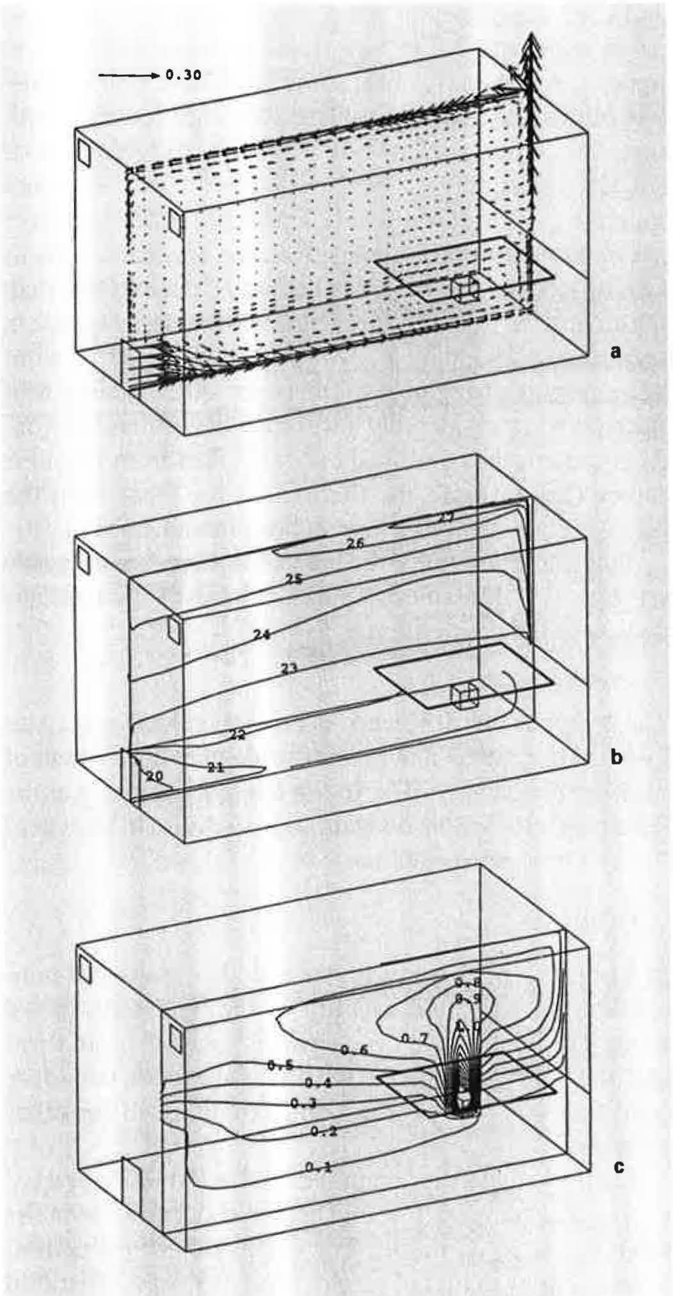

Fig. 4. Computed results for the room with displacement ventilation by the RNG k-ε model. **a** Velocity in mid-section (y = 1.5 m) ($m \cdot s^{-1}$). **b** Temperature in mid-section (y = 1.5 m) (°C). **c** Helium concentration in the section via the occupant (y = 0.8 m) (%).

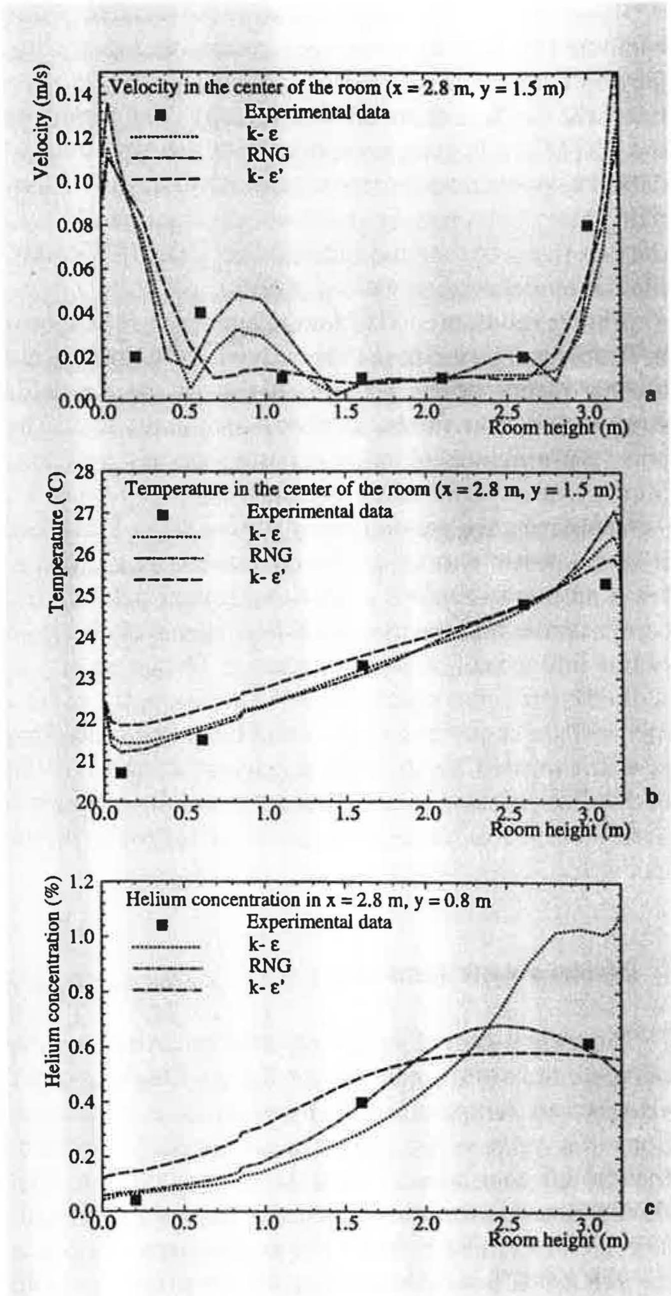

Fig. 5. Comparison between the measured data and computed results by the three eddy-viscosity models. **a** Velocity. **b** Temperature. **c** Helium concentration.

velocities near the floor were less accurate, because there were small obstacles on the floor, such as measuring wires, that may disturb the airflows. In addition, the velocities were measured by hot-wire anemometers, for which convection induced by the heated sensors can be significant at low velocities. In many cases, measured velocities less than 0.1 $m \cdot s^{-1}$ are regarded as unreliable, although the equipment specifications indicate a much lower value. Hence, it is difficult to judge whether it was a defect of the models, or the low accuracy in the measurements that caused the differences between the computed results and the measured data.

The computed temperature profiles seem in agreement with the measured data. However, there is about 1–1.5 K difference near the ceiling and floor. The helium concentration computed near the ceiling by the k-ε model is much higher than the measured data. The k-ε' model overpredicted the helium concentration in the lower part of the room. Only the profile predicted by the RNG model agrees with experimental data. The results show that the RNG model performs slightly better than the other models in concentration prediction. However, there are still some large discrepancies.

Discussion

Arguably, the eddy-viscosity models predicted the flow more accurately for displacement ventilation than they did for the plume. In fact, the quality of the experimental data for the plume is much higher than for the room airflow. The validation in the plume case was done not only for the mean variables but also for the second-order variables – variables that are much more difficult to predict. The validation for the room airflow was investigated only for the mean variables as the measured data were limited. Nevertheless, the differences were considerable in some locations, especially near the ceiling and floor. It should also be noted that the quality of the experimental data is not very high near the floor. From our previous experiences [8, 9] and the plume case, it is anticipated that the Reynolds stress model should present better results. We will continue to report the progress by using the Reynolds stress model in the future.

Conclusions

Three eddy-viscosity models and a Reynolds stress model were used to predict airflows in a strong turbulent buoyant plume for which experimental data from the literature were available for validation. The computed results using the Reynolds stress model were in good agreement with the experimental data but the eddy-viscosity models were not able to correctly predict turbulence levels in the plumes investigated. This shortcoming led to a poor prediction of the mean velocity and temperature profiles in the plume. In addition, the eddy-viscosity models were unable to predict the anisotropic turbulence in the plume, as these models are inherently isotropic.

The three eddy-viscosity models also were used to predict the flow in a room with a displacement ventilation system. The predicted results agreed fairly well with experimental data. However, discrepancies were significant between the computed results and measured data for some specific locations, especially near the ceiling and floor. The RNG k-ε model performed slightly better than the other eddy-viscosity models studied in the prediction of the contaminant concentration.

References

1 Roomvent, 1996. Proc Roomvent '96 Conference, Yokohama, Japan.
2 Indoor Air, 1996. Proc Indoor Air '96 Conference, Nagoya, Japan.
3 Sandberg M: Measurement techniques in room airflow; In Proc Roomvent '94: Air Distribution in Rooms, Krakow, 1994.
4 Nielsen PV: Air distribution in rooms – research and design methods; in Proc Roomvent '94: Air Distribution in Rooms, Krakow, 1994.
5 Chen Q: Computational fluid dynamics for HVAC: Successes and failures. ASHRAE Trans 1997;103(1).
6 Launder BE, Spalding DB: The numerical computation of turbulent flows; in Computer Methods in Applied Mechanics and Energy, North Holland Publishing Company, 1974, vol 3, pp 269–289.
7 Chen Q, Jiang Z: Significant questions in predicting room air motion. ASHRAE Trans 1992;98(part 1):929–939.
8 Chen Q: Comparison of different k-ε models for indoor airflow computations. Numerical Heat Transfer (B) 1995;28:353–369.
9 Chen Q: Prediction of room air motion by Reynolds-stress models. Build Environ 1996;31: 233–244.
10 Yakhot V, Orzag SA, Thangam S, Gatski TB, Speziale CG: Development of turbulence models for shear flows by a double expansion technique. Phys Fluids A 1992;4:1510–1520.
11 Malin MR, Younis BA: Calculation of turbulent buoyant plumes with a Reynolds stress and heat flux transport closure. Int J Heat Mass Transfer 1990;33:2247–2264.
12 Tennekes H, Lumley JL: A First Course in Turbulence. MIT Press, Cambridge, 1972.
13 Marlin, MR: Private communications. London, Cham, 1995.
14 Spalding DB: The PHOENICS Encyclopedia. London, CHAM, 1994.
15 Kofoed, P: Thermal plumes in ventilated rooms, PhD thesis, Aalborg, 1991.
16 Shabbir A, George WK: Experiments on a round turbulent buoyant plume. J Fluid Mech 1994;275:1–32.
17 Dai Z, Tseng LK, Faeth GM: Velocity statistics of round, fully developed, buoyant turbulent plumes. J Heat Transfer 1995;117:138–145.