Simplified Method for Indoor Airflow Simulation

Qirigyan Chen and Weiran Xu
Building Technology Program, Department of Architecture
Massachusetts Institute of Technology
Room 5-418, 77 Massachusetts Avenue, Cambridge, MA 02139-4307, USA
Phone: 617-253-7714, Fax: 617-253-6152
E-mail: qchen@mit.edu
URL: http://web.mit.edu/qchen/www/

ABSTRACT

At present, numerical simulation of room airflows is mainly conducted by either the Computational-Fluid-Dynamics (CFD) method or various zonal/network models. The CFD approach needs a large capacity of computer and a skillful expert. The results obtained with zonal/network models have great uncertainties.

This paper proposes a new simplified method to simulate three-dimensional distributions of air velocity, temperature, and contan-finant concentrations **in** rooms. The method assumes turbulent viscosity to be a function of length-scale and local mean velocity. The new model has been used to predict natural convection, forced convection, mixed convection, and displacement ventilation in a room. The results agree reasonably with experimental data and the CF1) computations. The simplified method uses much less computer memory and the computing speed is at least 10 times faster, compared with the CFD method. The grid number can often be reduced so that the computing time needed for a three-dimensional case can be a few minutes in a PC.

INTRODUCTION

Proper design of indoor environment requires detailed information of indoor air distribution, such as airflow pattern, velocity, temperature, and contarruinant concentrations. The information can be obtained by experimental measurements and computational simulations. Experimental measurements are reliable but need large laboreffort and time. Therefore, the experimental approach is not feasible as a general design tool. Two approaches of computational simulations are available for the study of indoor air distribution. The first approach is the computational-fluld-dynamics (C FD) methods and the second is simplified flow simulation methods.

Computational-Fluid-Dynamics Methods

The CFD methods solve the Navier-Stokes equations for flows. For laminar flows the computed results are accurate and reliable. However, it is difficult to predict turbulent

flows. Very fine numerical resolution is required to capture all the details of the indoor turbulent flow. This type of simulation is direct numerical simulation. The direct numerical simulation for a practical flow needs a huge computer system that **is** not available.

Indoor airflow simulations use turbulence models to compute the mean values. This can be done with the capacity and speed of present computers. Eddy-viscosity models are the most popular turbulence models. The CFD program with eddy-viscosity models solves air velocities, temperature, contaminant concentration, and turbulent quantities in a space. The space is divided into 10,000 to one million cells to achieve a reasonable accuracy for a three-dimensional flow problem.

In addition, the CFD program user should have good knowledge of fluid dynamics, numerical technique, and indoor air distribution. However, a large computer and a skillful ill not user wi guarantee success. Chen (1997) reported many failures in using the CFD method in a group with more than ten-year experience. Obviously, most HVAC designers and architects do not have the computer capacity and the CFD knowledge. Therefore, in predicting indoor air distribution and designing a comfortable indoor environment, application of the CFD method is limited.

Simplified Flow Simulation Methods

The second approach does not use a turbulence model. The approach uses a much coarse cell system. In most cases, the total cell number for a space is less than 1 0,000.

A very simple method (Lebrun and Ngendakumana 1987) is to fix airflow patterns and use empirical flow laws for different flow components, such as 'ets, plumes, etc. In many cases, the airflow patterns are difficult to impose even by an experienced fluid dynamics engineer. The method has limited applications.

Another popular method is the network model (Walton 1989). The model determines flow within a space by Bernoulli's equation. The method works reasonably for parabolic flows and is useful to analyze combined problems of HVAC systems, infiltration, and multi-room airflow simultaneously. However, the uncertainty is large if the method is applied for a room presented by several different cells or sub-volumes.

The method proposed by Wurtz and Nataf (1994) is to calculate indoor air pressure using a degraded equation for the momentum. The airflow between two zones is determined by the pressure differential. Because of the poor representation of the momentum, the method does not work for pressure and buoyancy driven flows, i.e. flows set up by temperature differences in the air.

A recent zonal model developed by Inard et al. (1996) calculates flow rate for zones with small momentum through pressure distribution. Although the results are consistent with experimental data, the model may not be applied for high momentum

flows. In addition, the method uses a discharge coefficient that must be determined through experiment.

When a room is subdivided by a partition wall or a large opening, all the above models use a discharge coefficient to calculate flow due to pressure or temperature difference. This will further reduce the reliability of the methods since a general form for the discharge coefficient

.J has not been established. Calculations for new geometries require an CFD run or experiment to determine the discharge coefficient.

Justification of Need

Many HVAC design engineers and architects have limited knowledge of fluid flow and do not have the access to a large computer. It is important to develop a simplified model to simulate indoor airflow in a personal computer. The flow program should then be coupled with an energy analysis program to simulate simultaneously airflow, thermal comfort, and energy consumption of HVAC systems. The program will also allow the temperature of interior walls to be predicted. The program would serve as a tool to accurately provide design information and to properly size HVAC systems and assure comfort conditions exist at all important locations within the space.

The goal of the present investigation is to develop a program which will provide design information to establish acceptable comfort conditions through the interior space. Precise rigor and exact predictions will be relaxed to allow the program to be easily used by 11VAC engineers with a minimum of training and modest desktop personal computers. The following section describes a new simplified method.

NEW SIMPLIFIED METHOD

Governing Flow Equations

Most indoor airflows are turbulent. Often airflow calculations use the Buossinesq approximation. The approximation takes air density as constant in the momentum terms and considers the buoyancy influence on air movement by the difference between the local air weight and the pressure gradient. With an eddy-viscosity model, the indoor airflows can be described by the following time-averaged Navier-Stokes equations for the conservation of mass, momentum, energy, and species concentrations:

Mass continuity:

$$\frac{\partial Vi}{\partial x_i} = 0$$

where Vi   = mean velocity component in xi-direction
        xi   = coordinate (for 1= 1, 2, 3, xi corresponds to three perpendicular axes).

3

Momentum:

$$apvi \quad apvi\, Vj \qquad ap + a \quad avj + \quad av \quad i$$

PO(T(- T)gi   (2)

$$at \qquad axi \quad axi \quad ex\ ex \quad axi$$

where p = air density

p = pressure pt,ff = effective viscosity

T,,   temperature in a reference point
T   temperature
g gravity acceleration

Vj = velocity component in xj-direction

p = thermal expansion coefficient of air

The last term on the right side of the equation is the buoyancy term.

The turbulent influences are lumped into the effective viscosity is the sum of the turbulent viscosity and larninar viscosity, @l:

$$PLCff = pt\ t + R \tag{3}$$

The Prandtl-Kolmogorov assumption, the turbulent viscosity expresses as the product of turbulence ki energy, k, and turbulent macroscale, 1, that is a proper length scale for turbulence interactions:

$$PLt\ Cvpk\ 1/21 \tag{4}$$

where C, 0.5478, an empirical constant. Depending on how to solve the unknown parameters k an eddy-viscosity models have different forms. The simplest model is probably the Prandtl's mixing-le model (Prandtl 1926) and complicated ones use multi-equations for turbulence transport. The standar E model (Launder and Spalding 1974) is the most widely used two-equation model.

From the results of our previous investigations, Tu is about 10% for many room airflows. We h found a single algebraic function to express the turbulent viscosity as a function of local mean velocity and a length scale, 1:

$$PL, = 0.03874\ p\ V\ 1 \tag{5}$$

This equation is a universal one without adjustable constants between different flow conditions.

4

- Momentum:

$$\frac{\partial \rho V_i}{\partial t} + \frac{\partial \rho V_i V_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{\text{eff}}\left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}\right)\right] + \rho\beta\left(T_o - T\right)g_i \tag{2}$$

where ρ = air density
V_j = velocity component in x_j-direction
p = pressure
μ_{eff} = effective viscosity
β = thermal expansion coefficient of air
T_o = temperature in a reference point
T = temperature
g = gravity acceleration

The last term on the right side of the equation is the buoyancy term.

The turbulent influences are lumped into the effective viscosity is the sum of the turbulent viscosity, μ_t, and laminar viscosity, μ:

$$\mu_{\text{eff}} = \mu_t + \mu \tag{3}$$

The Prandtl-Kolmogorov assumption, the turbulent viscosity expresses as the product of turbulence kinetic energy, k, and turbulent macroscale, l, that is a proper length scale for turbulence interactions:

$$\mu_t = C_v \rho k^{1/2} l \tag{4}$$

where C_v = 0.5478, an empirical constant. Depending on how to solve the unknown parameters k and l, eddy-viscosity models have different forms. The simplest model is probably the Prandtl's mixing-length model (Prandtl 1926) and complicated ones use multi-equations for turbulence transport. The standard k-ε model (Launder and Spalding 1974) is the most widely used two-equation model.

From the results of our previous investigations, Tu is about 10% for many room airflows. We have found a single algebraic function to express the turbulent viscosity as a function of local mean velocity, V, and a length scale, l:

$$\mu_t = 0.03874\,\rho\,V\,l \tag{5}$$

This equation is a universal one without adjustable constants between different flow conditions.

4

Energy:

To determine the temperature distribution and the buoyancy term in Equation (2), the conservation of energy must be solved.

$$\frac{\partial \rho T}{\partial t} + \frac{\partial \rho V_j T}{\partial x_i} = \frac{\partial}{\partial x_i}\Gamma_{T,eff}\frac{\partial T}{\partial x_i} + q/C_p \qquad (6)$$

where $\Gamma_{T,ff}$ = effective turbulent diffusion coefficient for T
q = thermal source
C_P = specific heat

In our work we have estimated the effective diffusive coefficient for temperature in Equation $\Gamma_{T,ff}$, by:

$$\Gamma_{T,ff} = \frac{R_{eff}}{Pr_{,ff}} \qquad (7)$$

where the effective Prandd number, $Pr_{,ff}$, is 0.9.

· Species concentrations:

For determination of pollutant or water vapor concentration distribution the conservation of mass must be combined with the equation of transfer of the species.

$$\frac{\partial \rho c}{\partial t} + \frac{\partial \rho V_i c}{\partial x_i} = \frac{\partial}{\partial x_i}\Gamma_{c,eff}\frac{\partial c}{\partial x_i} + S_C \qquad (8)$$

where C = species concentration
$\Gamma_{c,ff}$ = effective turbulent diffusion coefficient for C
S_c = source term of C

Similar method to the energy equation is used to determine the effective diffusive coefficient species concentration in Equation (8), $\Gamma_{c,,ff}$:

$$\Gamma_{c,ff} = \frac{\mu_{eff}}{Sc_{eff}} \qquad (9)$$

where effective Schmidt number, $Sc_{,ff}$, is 1.0.

Equations (1) to (9) form the new simplified model.

5

- Energy:

To determine the temperature distribution and the buoyancy term in Equation (2) the conservation of energy must be solved.

$$\frac{\partial \rho T}{\partial t} + \frac{\partial \rho V_j T}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\Gamma_{T,eff}\frac{\partial T}{\partial x_j}\right) + q/C_p \tag{6}$$

where $\Gamma_{T,eff}$ = effective turbulent diffusion coefficient for T
q = thermal source
C_p = specific heat

In our work we have estimated the effective diffusive coefficient for temperature in Equation (6), $\Gamma_{T,eff}$, by:

$$\Gamma_{T,eff} = \frac{\mu_{eff}}{Pr_{eff}} \tag{7}$$

where the effective Prandtl number, Pr_{eff}, is 0.9.

- Species concentrations:

For determination of pollutant or water vapor concentration distribution the conservation of mass must be combined with the equation of transfer of the species.

$$\frac{\partial \rho C}{\partial t} + \frac{\partial \rho V_j C}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\Gamma_{C,eff}\frac{\partial C}{\partial x_j}\right) + S_C \tag{8}$$

where C = species concentration
$\Gamma_{C,eff}$ = effective turbulent diffusion coefficient for C
S_C = source term of C

Similar method to the energy equation is used to determine the effective diffusive coefficient for species concentration in Equation (8), $\Gamma_{C,eff}$:

$$\Gamma_{C,eff} = \frac{\mu_{eff}}{Sc_{eff}} \tag{9}$$

where effective Schmidt number, Sc_{eff}, is 1.0.

Equations (1) to (9) form the new simplified model.

5

Boundary Conditions

Boundary conditions are necessary for the mathematical solution of the govern' ing flow equations. There are three types of boundaries of practical importance: free boundary, symme surface, and conventional boundary.

Free boundary

The boundary surface may be adjacent to an inviscid stream. Examples are air supply outle and return inlet. For a supply outlet. the boundary conditions are:

$$Vi \quad V_{,.}ppiy$$
$$T \quad Tsupply \quad (1\ 0)$$
$$c \quad Csupply$$

where subscripts "supply" are the parameter values at the supply outlet.

Pressure is normally given for a return inlet and zero gradients normal to the surface are assumed for other parameters:

$$p = preturil$$

$$0$$

$$\frac{axi}{aT}$$
$$\frac{axi}{ac}$$

$$ex$$

where preturn is the pressure at a return inlet.

Symmetry surface

If the xi coordinate is normal to the symmetry surface, the following equations describe the boundary conditions of the surface:

$$avi$$

$$0$$

axi

a̲T̲

axi

a̲c̲

axi

6

Boundary Conditions

Boundary conditions are necessary for the mathematical solution of the governing flow equations. There are three types of boundaries of practical importance: free boundary, symmetry surface, and conventional boundary.

- Free boundary

The boundary surface may be adjacent to an inviscid stream. Examples are air supply outlet and return inlet. For a supply outlet, the boundary conditions are:

$$
\begin{aligned}
V_i &= V_{supply} \\
T &= T_{supply} \\
C &= C_{supply}
\end{aligned}
\tag{10}
$$

where subscripts "supply" are the parameter values at the supply outlet.

Pressure is normally given for a return inlet and zero gradients normal to the surface are assumed for other parameters:

$$
\begin{aligned}
p &= p_{return} \\
\frac{\partial V_i}{\partial x_i} &= 0 \\
\frac{\partial T}{\partial x_i} &= 0 \\
\frac{\partial C}{\partial x_i} &= 0
\end{aligned}
\tag{11}
$$

where p_{return} is the pressure at a return inlet.

- Symmetry surface

If the x_i coordinate is normal to the symmetry surface, the following equations describe the boundary conditions of the surface:

$$
\begin{aligned}
\frac{\partial V_i}{\partial x_i} &= 0 \\
\frac{\partial T}{\partial x_i} &= 0 \\
\frac{\partial C}{\partial x_i} &= 0
\end{aligned}
\tag{12}
$$

Conventional boundary

This type of boundary surfaces includes wall, ceiling, and floor Surfaces and the surfaces of furniture, appliance, and occupants. If xi coordinate is parallel to the surface, the boundary conditions are:

$$\underline{\frac{\partial v_i}{\partial x_i}}$$

$$T = @\text{leIT } \frac{\partial}{\partial x_i}$$

$$q = h(T_w - T) \qquad (13)$$

$$S_C \quad C_{source}$$

where T = shear stress

   h = convective heat transfer coefficient

   C_{source} = species concentration source

The convective heat transfer coefficient is determined from the follows](equation which

is similar to the Reynolds analogy:

$$\frac{P_t \text{ eff } C_p}{P_{reff} \; \Delta X_j} \qquad (14)$$

where Δx_j is the distance between the surface and the first arld close to the surface.

W

APPLICATION EXAMPLES

This section demonstrates the new simplified method by applying it to predict indoor airflows of.

   Natural convection
   Forced convection

   Mixed convection

0   Displacement ventilation

Natural, forced, and mixed convection represent the basic elements of room airflows. For simplicity, two-dimensional case selected to demonstrate the new simplified model. The displacement ventilation case used **is** three-dimensional with complicated boundary conditions. The displacement ventilation case is a test of the overall performance of the new model.

Natural Convection

For natural convection, the experimental data of 01son and Glicksman (1991) as shown in Fig. 1 will be used.

- Conventional boundary

This type of boundary surfaces includes wall, ceiling, and floor surfaces and the surfaces of furniture, appliance, and occupants. If x_i coordinate is parallel to the surface, the boundary conditions are:

$$\tau = \mu_{eff} \frac{\partial V_i}{\partial x_j}$$

$$q = h(T_w - T)$$

$$S_C = C_{source} \qquad (13)$$

where τ = shear stress
h = convective heat transfer coefficient
C_{source} = species concentration source

The convective heat transfer coefficient is determined from the following equation which is similar to the Reynolds analogy:

$$h = \frac{\mu_{eff}}{Pr_{eff}} \frac{C_p}{\Delta x_j} \qquad (14)$$

where Δx_j is the distance between the surface and the first grid close to the surface.

APPLICATION EXAMPLES

This section demonstrates the new simplified method by applying it to predict indoor airflows of:
- Natural convection
- Forced convection
- Mixed convection
- Displacement ventilation

Natural, forced, and mixed convection represent the basic elements of room airflows. For simplicity, two-dimensional cases are selected to demonstrate the new simplified model. The displacement ventilation case used is three-dimensional with more complicated boundary conditions. The displacement ventilation case is a test of the overall performance of the new model.

Natural Convection

For natural convection, the experimental data of Olson and Glicksman (1991) as shown in Fig. 1 will be used.

7

Hot wall 20

cr

C:L

rD

CD   Pi   pi

Z4 ciL

rD   CD   <   Pj   W-

CJ   rD

rD   rD   CD

rD

n   ID   S@
Cl.

pj   -,:i

<   ID   zi

@g-   r)

nG@5   -   ::;'

,p   <   C)

CD C)   (D

(D

n

PI   CD

rD m;.   rD

rD   <

n   ID   CD rD
=

-1

rD   .,tt

n

rD   Cold wall 12@@C

r)

rD   CD   CL

n   **rD**

rD

c)
(D   2.5 m

rD

rD

n

_n

::7

@u'   0
cr   c:l_

n   7

layer thickness are not correct for the simplified model due to the large cell size used. However, we found only the Lam-Bremhorst model could predict the reversed flow when we tested quite a few eddy-viscosity models.

Fig. 3 presents the dimensionless temperature profiles in the vertical center line. The simplified model predicts the temperature profile better than the CFD model in this particular case.

Forced Convection

The forced convection case uses the experimental data from Restivo (1979) shown in Fig. 4. The Reynolds number is 5000 based on bulk supply velocity and the height of air supply outlet. The air supply outlet h = 0.056 H, and exhaust inlet h' = 0.16 H.

Fig. 5 compares the airflow patterns by the simplified model and the CFD method with standard k-ε model (Launder and Spalding 1974). The computed velocity profiles are compared in Fig. 6 with experimental data in two vertical sections x/H =1 and x/H=2 respectively and two horizontal sections, y/H = 0.972 (through the air supply outlet) and y/H = 0.028 (through the air exhaust inlet). The results of the simplified model show a jet decay that is too strong. Hence, the primary flow near the ceiling and the return flow near the floor are smaller than the data. In this case, the k-ε model predicts a satisfactory result. Nevertheless, the simplified model could predict the second recirculation on the upper right corner, though the recirculation is too large. However, the k-ε model fails to predict the recirculation.

8

(b)

liar   - ------------

ct

C..L,

(c)

Fig. 2. Comparison of the airflow patternsfor natural convection: (a) simplified method,
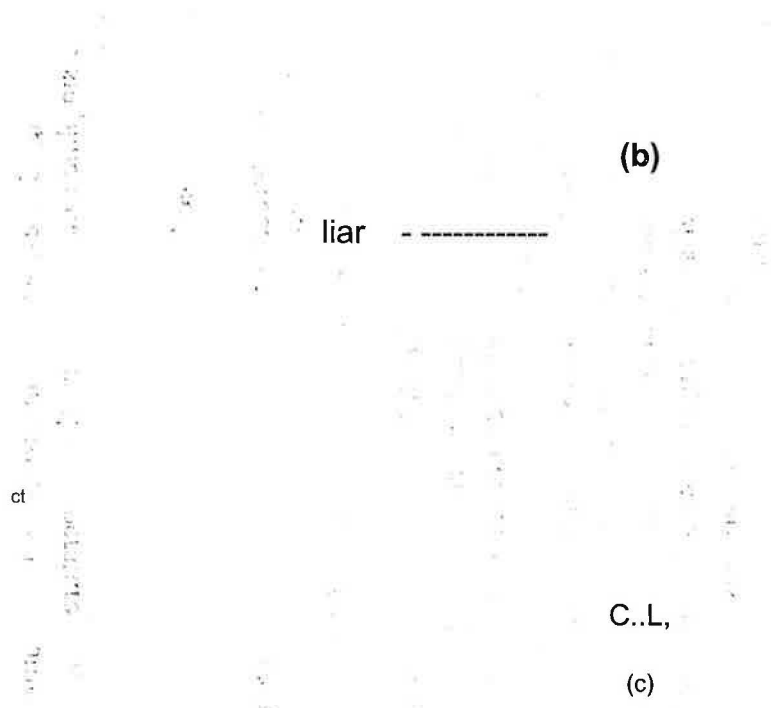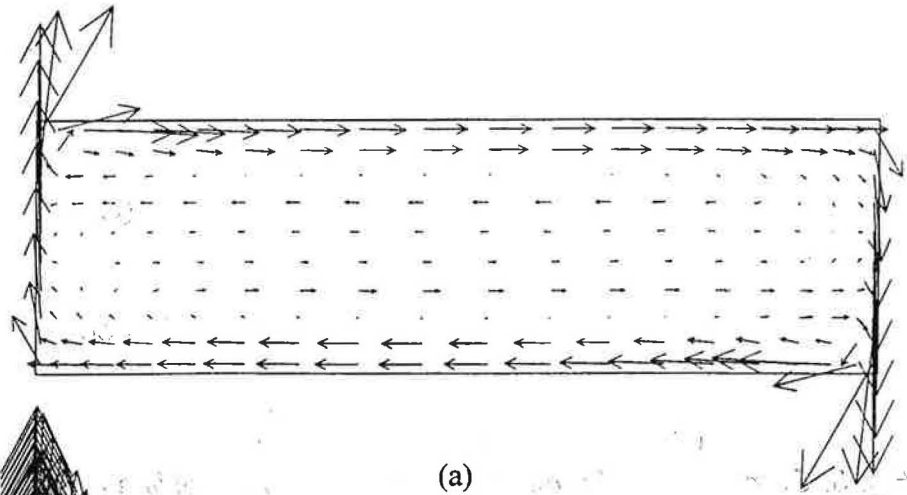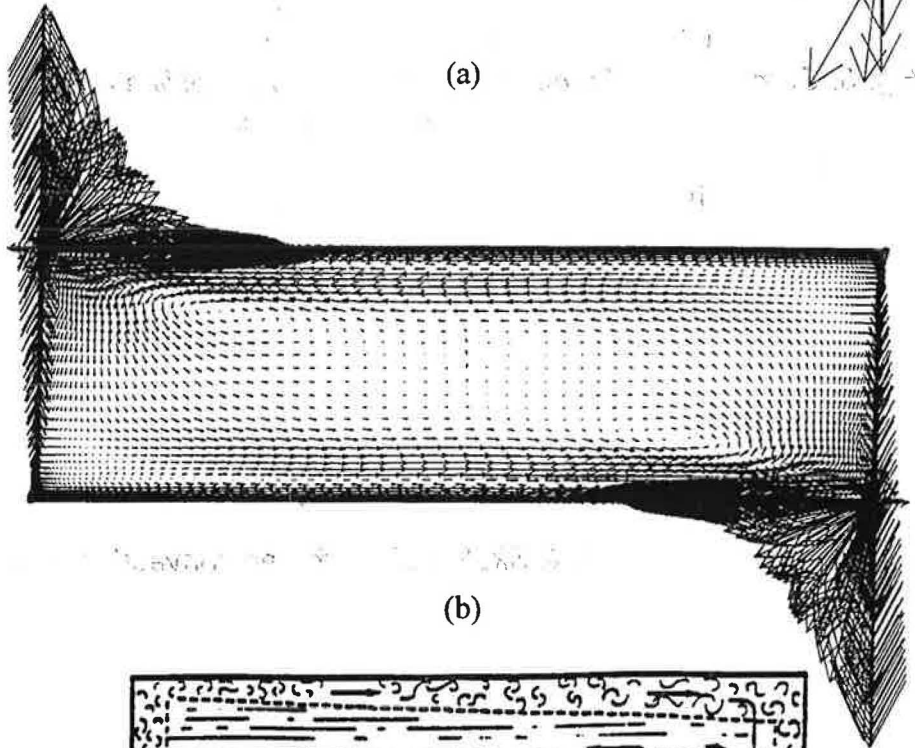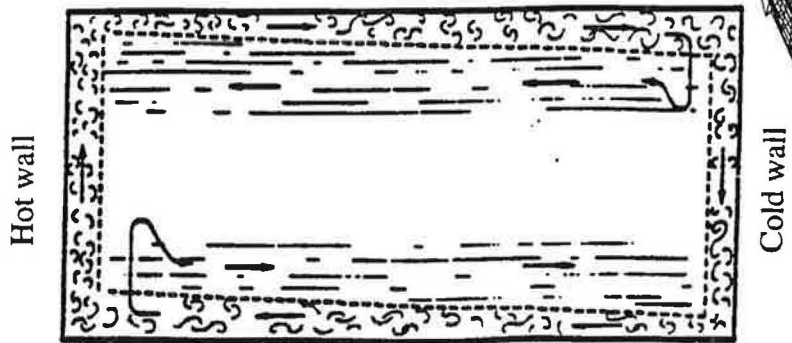*(b) CFD method, (c) smoke visualization.*

9

Hot wall

Cold wall

Fig. 2. Comparison of the airflow patterns for natural convection: (a) simplified method, (b) CFD method, (c) smoke visualization.

1.0

0.8

0.6

0.4

Simplified Model

0.2

-------- CFD Model

Data

.0   0.2   0.4 0 0.6   0.8   1.0

Fig. 3. Comparison of temperature profile in vertical line at the middle of the room with natural convection.

h

Isothen-nal forced convectionH'

AY   h'
                                X

3H

Fig. 4. Sketch of the forced convection case.

(a)

(b)

Fig. 5. Comparison of the airflow patternsfor theforced convection: (a) simplified method, (b) CFD method..
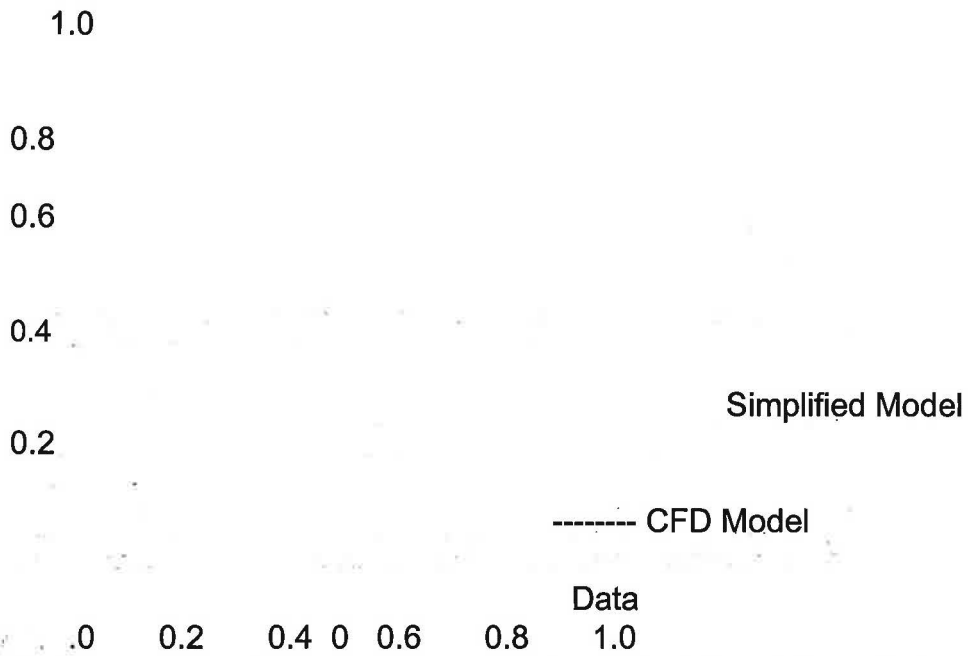
10

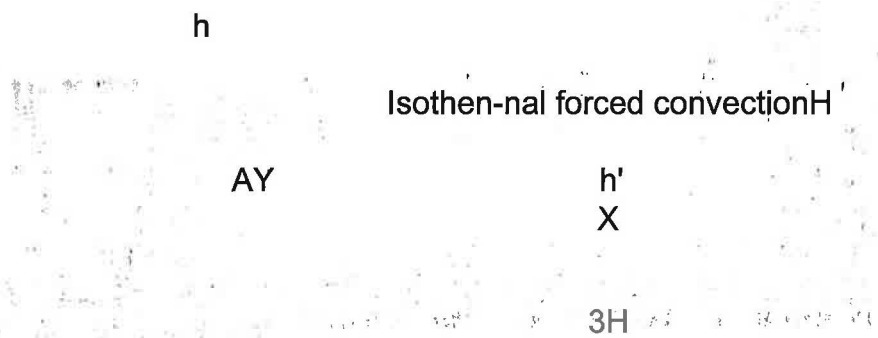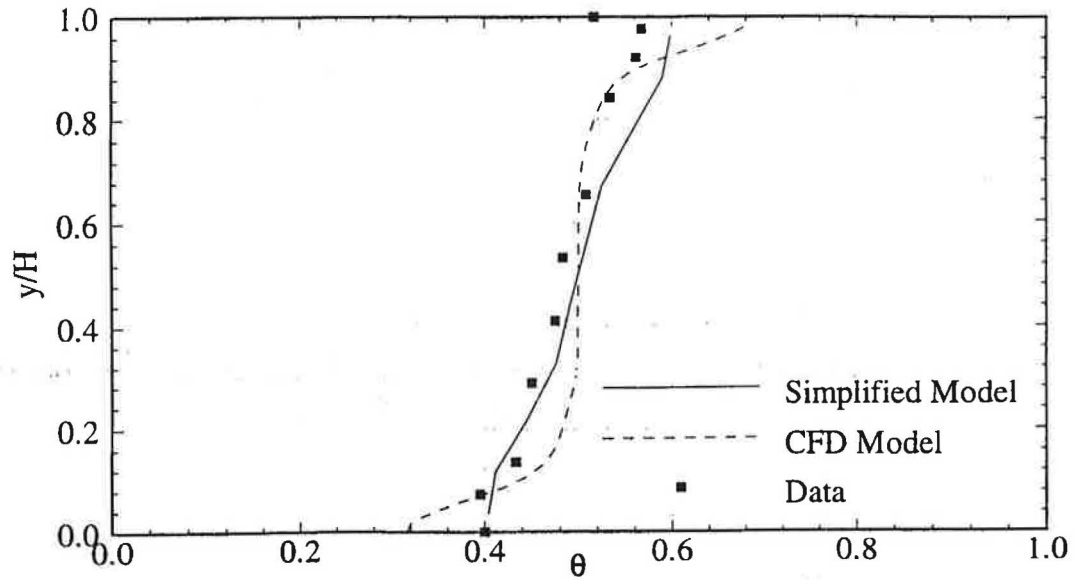*Fig. 3. Comparison of temperature profile in vertical line at the middle of the room with natural convection.*

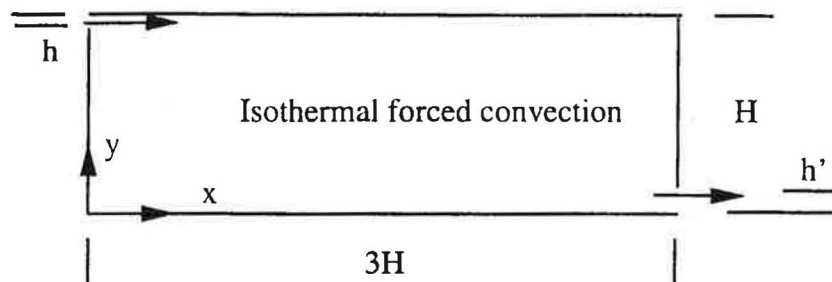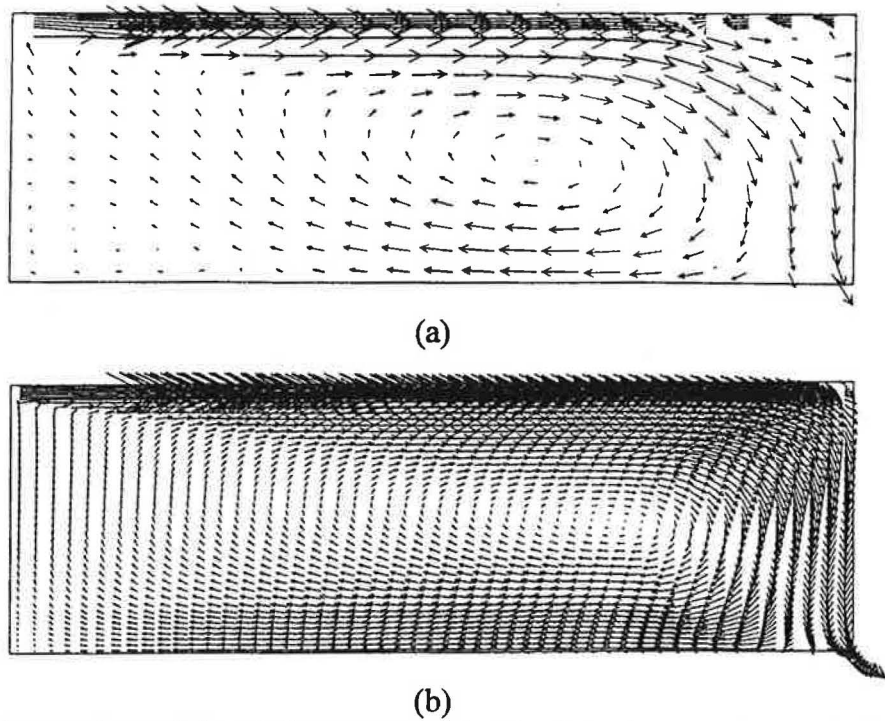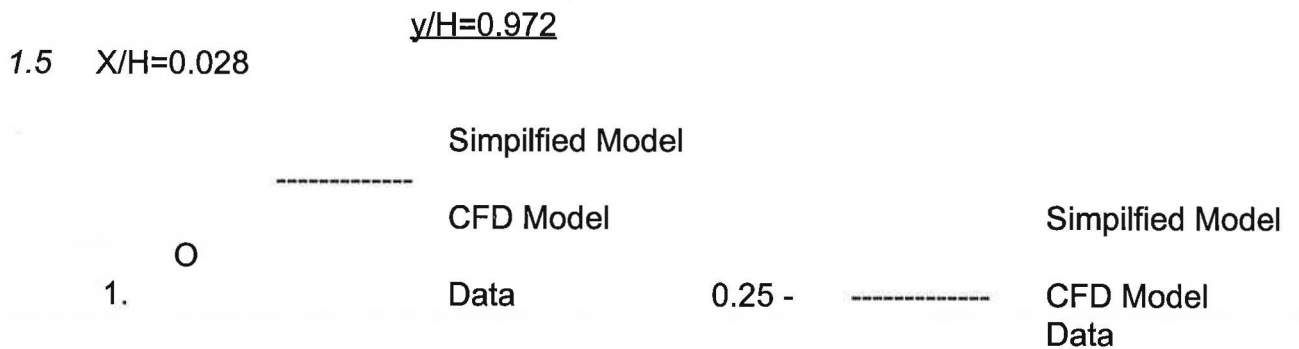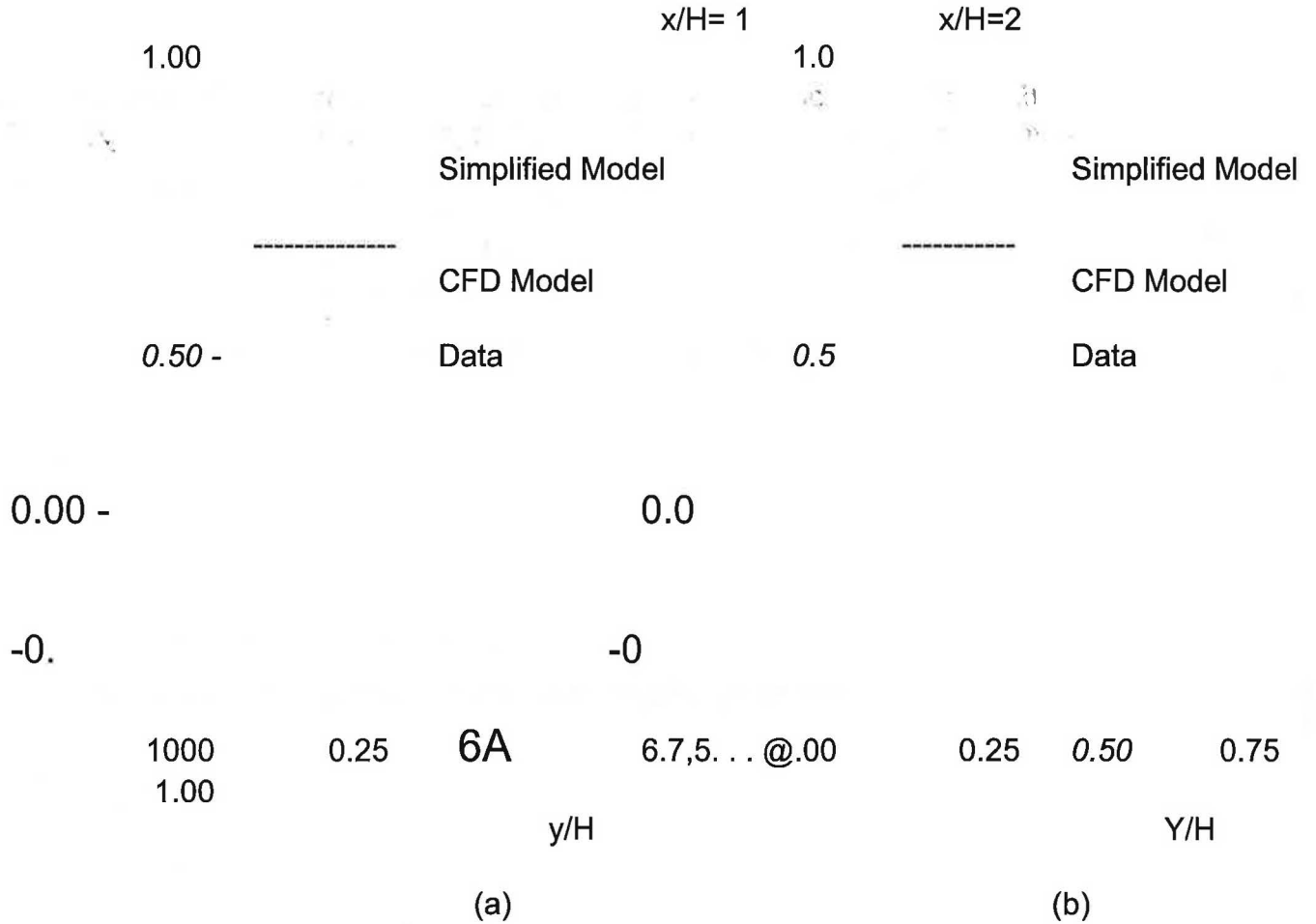*Fig. 4. Sketch of the forced convection case.*

(a)

(b)

Fig. 5. Comparison of the airflow patterns for the forced convection: (a) simplified method, (b) CFD method..

Mixed Convection

The mixed convection case uses the experimental data from Schwenke (1975). The cas
similar to the forced convection but the room len-th is 4.7 H and the height of the air su
outlet h = 0.025 H. The right wall is heated but the ceiling and floor are adiabatic. Schwe
conducted a series measurements with different Archimedes numbers, Ar, ranging from 0.
to 0.02.

x/H= 1   x/H=2

1.00   1.0

                    Simplified Model   Simplified Model

        ---------------   CFD Model   ----------   CFD Model

0.50 -   Data   0.5   Data

0.00 -   0.0

-0.   -0

1000   0.25   6A   6.7,5. . . @.00   0.25   0.50   0.75
1.00
                    y/H   Y/H

            (a)   (b)

y/H=0.972

1.5   X/H=0.028

                    Simpilfied Model

        -------------   CFD Model   Simpilfied Model

    O
1.   Data   0.25 -   -------------   CFD Model
                                                    Data

0.5 -

0 . 00

0.0 -

-0.25 -

-0.

1.0   2.0   3.0 -0.5@.0

x/H

1.0   x/H   2.0   3.0

(c)

(d)

Fig. 6. Comparison of velocity profiles in differentsections of the room with forced convection: (a) atvW = 1, (b) at x1H =2, (c) (it y1H = 0. 9 72, and (d) at y1H = 0. 028.

11

Mixed Convection

The mixed convection case uses the experimental data from Schwenke (1975). The case is similar to the forced convection but the room length is 4.7 H and the height of the air supply outlet h = 0.025 H. The right wall is heated but the ceiling and floor are adiabatic. Schwenke conducted a series measurements with different Archimedes numbers, Ar, ranging from 0.001 to 0.02.
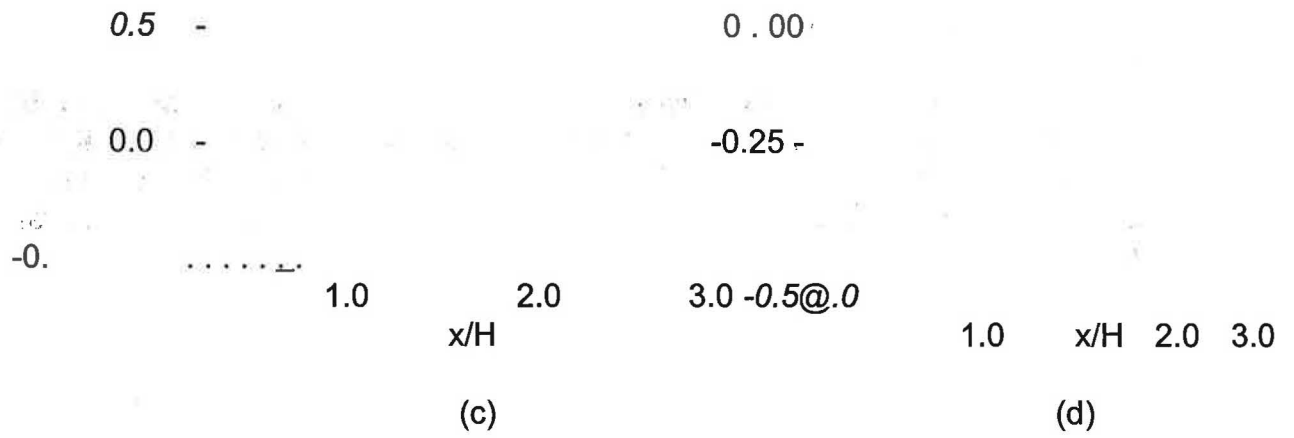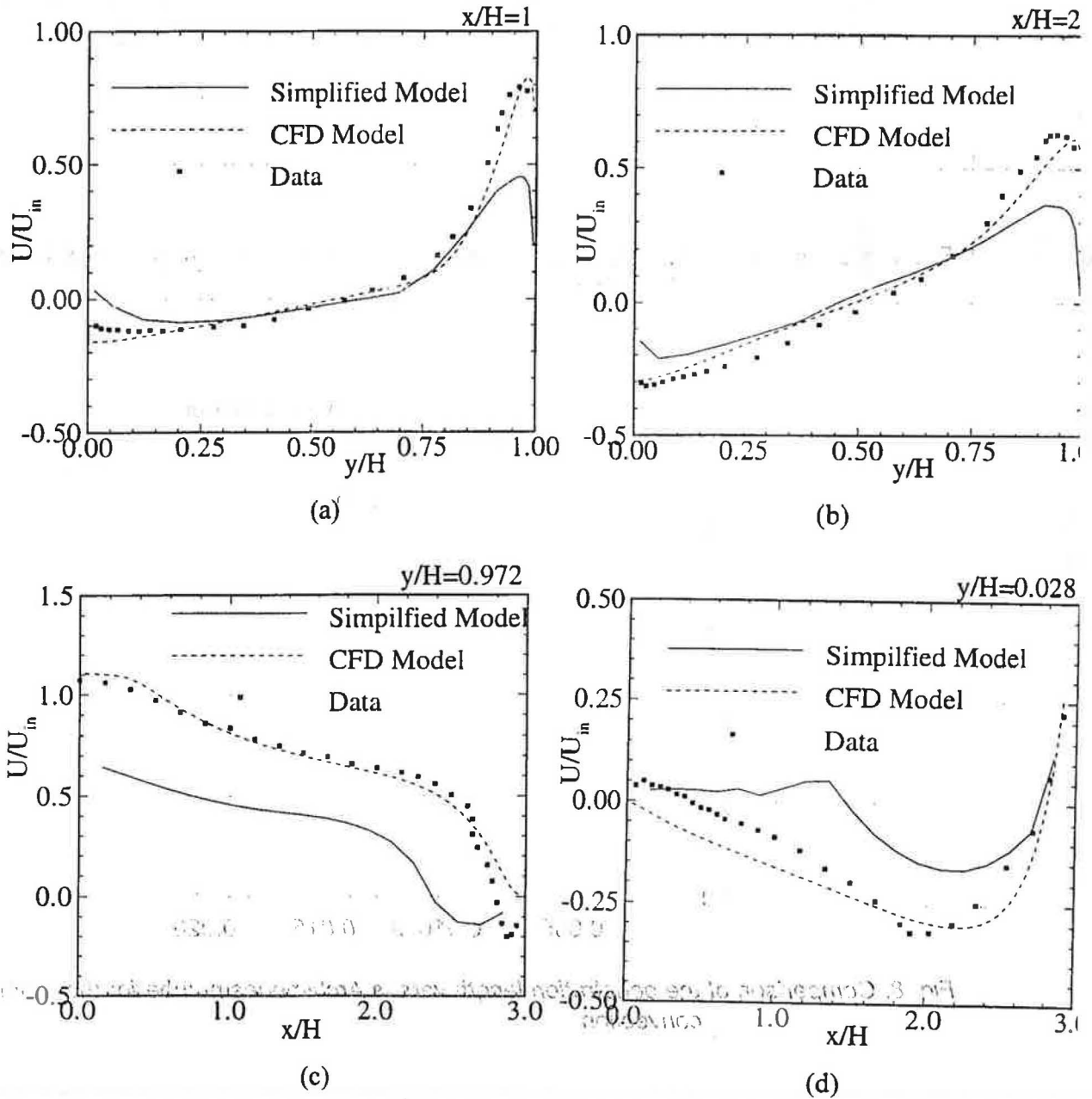

Fig. 6. *Comparison of velocity profiles in different sections of the room with forced convection: (a) at x/H =1, (b) at x/H =2, (c) at y/H = 0.972, and (d) at y/H = 0.028.*

11

Fig. 7 compares the computed airflow pattern by the simplified method with that by the CFD method with the standard k-F- model. The two results are similar. The airflow pattern is very sensitive to the Ar. The computed and measured penetration depths, x_e, versus different Ar numbers are compared in Fig. 8. The x, is the horizontal distance of air movement along the ceiling before it falls to the floor. The simplified model works better in high Ar but the CFD model better in low Ar.

7.7

(b)

Fig. 7. Comparison of the aifflow patternsfor the mixed convection: (a) simplified method, (b) CFD method..

3.0................

Simplified Model

2.5   ------- CFD Model

Data

2.0

1.5

1.0

0.5
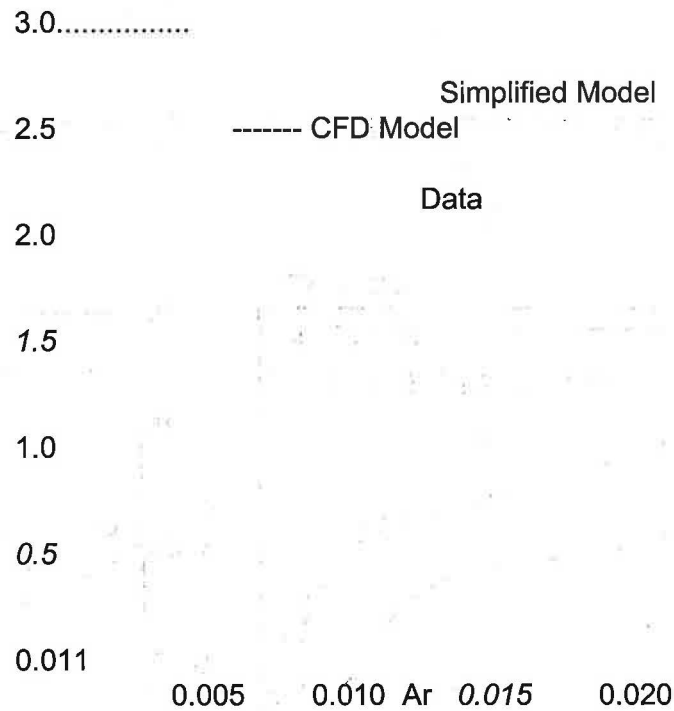
0.011
        0.005   0.010 Ar *0.015*   0.020

Fig. 8. Comparison of the penetration length versus Archimedes numberjor the room with mixed convection.

Fig. 7 compares the computed airflow pattern by the simplified method with that by the CFD method with the standard k-ε model. The two results are similar. The airflow pattern is very sensitive to the Ar. The computed and measured penetration depths, x_e, versus different Ar numbers are compared in Fig. 8. The x_e is the horizontal distance of air movement along the ceiling before it falls to the floor. The simplified model works better in high Ar but the CFD model better in low Ar.
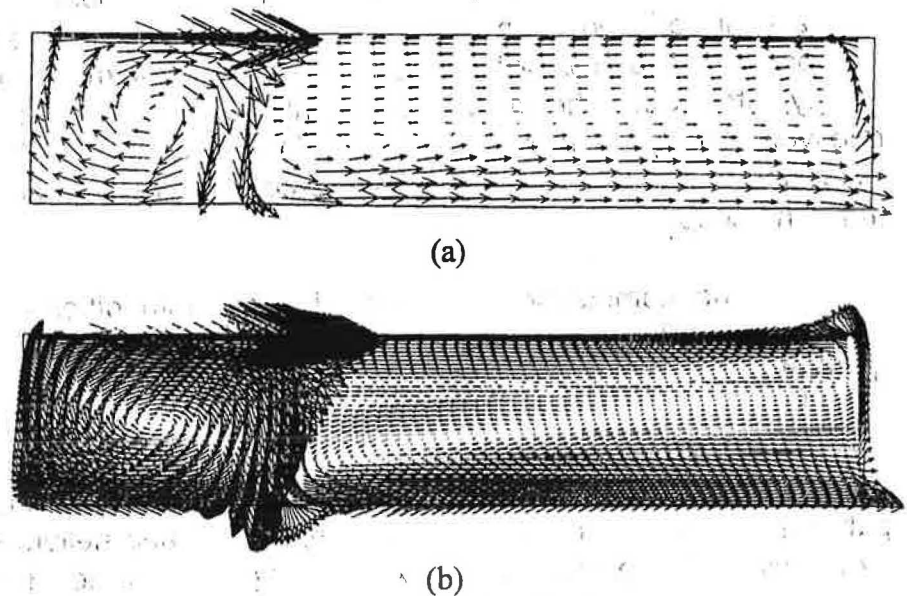

(a)

(b)

Fig. 7. *Comparison of the airflow patterns for the mixed convection: (a) simplified method, (b) CFD method..*
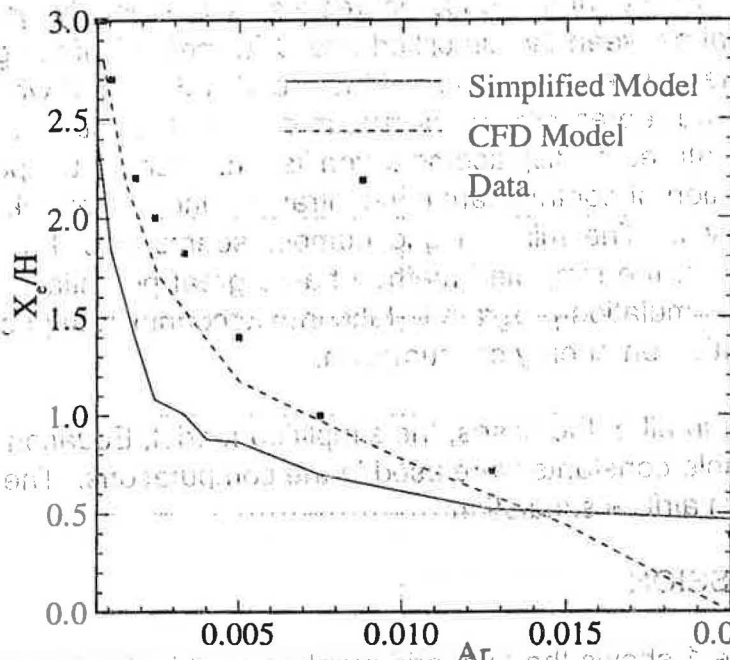

Fig. 8. *Comparison of the penetration length versus Archimedes number for the room with mixed convection.*

Displacement Ventilation

Fig. 9 shows the application of the simplified method and the CFD method with the standard k-F, model for the prediction of room airflow with a displacement ventilation system. The room dimension is 5.6 m long, 3.0 m wide, and 3.2 m high. A convective heat source, of 530 W on the window was used to simulate a summer cooling condition. The supply airflow rate was five air-change per hour. The corresponding supply air temperature was 19 OC. A box placed near the table was heated by a 25 W lamp to simulate a person sitting next to the table. The heat strength is considerably lower than that generated from an occupant. However, a helium source was also introduced in the box as a tracer gas to simulate contaminant from the occupant, such as CO_2 or tobacco smoke. The helium flow rate was 0.5% of the air supply rate. Since helium is much lighter than the air and the helium source was relatively strong in the room, the combined buoyant effect from the thermal source (heat from the lamp) and the mass source (helium) was as strong as that generated from an occupant.

The computations were carried out with different grid numbers with the simplified method: 31 x 28 x 26 (the same as the CFD method), 16 x 14 x 12, 10 x 10 x 10, and 6 x 7 x 6.. A grid number of 16 x 14 x 12 is minimum in order to represent the room geometry, such as the inlet, outlets, window, and table. Fig. 9 shows similar airflow patterns and the distributions of air temperature and helium concentration computed by the simplified method with 16 x 14 x 12 grids and the CFD method with 31 x 28 x 26 grids. Fig. 10 further compares the computed results with experimental data. The velocity and temperature profiles are at the center of the room and helium concentration profile at a line near to the center of the room. The agreement between the computed and measured results is reasonably good. The results are nearly identical between the simplified and CFD methods if the grid number is the same. It is possible to use a minimum grid number of 6 x 7 x 6 with which the table in the room cannot be represented. The accuracy of the results is relaxed but it does predict the main features of displacement ventilation, such as temperature gradient, non-uniform distribution of contaminant concentration, and higher risk of draft near the inlets at the floor level. The minimal grid number seems less than that used in zonal models. Therefore, the simplified method has a great potential to be used in an hour-by-hour energy simulation program to take into account the impact of non-uniform temperature distribution on energy consumption.

Note in all of the cases, the simplified model, Equation (5), is exactly the same. No adjustable constants were used in the computations. The simplified model is universal for room airflow simulation.

DISCUSSION

Table 1 shows the total grid number used in the four cases by the simplified model and the CFD models. It also shows the memory needed and CPU time used. The

convergence residuals are the same between the simplified and CFD computations. The residuals, R, are defined as:

13

Displacement Ventilation

Fig. 9 shows the application of the simplified method and the CFD method with the standard k-ε model for the prediction of room airflow with a displacement ventilation system. The room dimension is 5.6 m long, 3.0 m wide, and 3.2 m high. A convective heat source of 530 W on the window was used to simulate a summer cooling condition. The supply airflow rate was five air-change per hour. The corresponding supply air temperature was 19 °C. A box placed near the table was heated by a 25 W lamp to simulate a person sitting next to the table. The heat strength is considerably lower than that generated from an occupant. However, a helium source was also introduced in the box as a tracer gas to simulate contaminant from the occupant, such as CO_2 or tobacco smoke. The helium flow rate was 0.5% of the air supply rate. Since helium is much lighter than the air and the helium source was relatively strong in the room, the combined buoyant effect from the thermal source (heat from the lamp) and the mass source (helium) was as strong as that generated from an occupant.

The computations were carried out with different grid numbers with the simplified method: 31 x 28 x 26 (the same as the CFD method), 16 x 14 x 12, 10 x 10 x 10, and 6 x 7 x 6. A grid number of 16 x 14 x 12 is minimum in order to represent the room geometry, such as the inlet, outlets, window, and table. Fig. 9 shows similar airflow patterns and the distributions of air temperature and helium concentration computed by the simplified method with 16 x 14 x 12 grids and the CFD method with 31 x 28 x 26 grids. Fig. 10 further compares the computed results with experimental data. The velocity and temperature profiles are at the center of the room and helium concentration profile at a line near to the center of the room. The agreement between the computed and measured results is reasonably good. The results are nearly identical between the simplified and CFD methods if the grid number is the same. It is possible to use a minimum grid number of 6 x 7 x 6 with which the table in the room cannot be represented. The accuracy of the results is relaxed but it does predict the main features of displacement ventilation, such as temperature gradient, non-uniform distribution of contaminant concentration, and higher risk of draft near the inlets at the floor level. The minimal grid number seems less than that used in zonal models. Therefore, the simplified method has a great potential to be used in an hour-by-hour energy simulation program to take into account the impact of non-uniform temperature distribution on energy consumption.

Note in all of the cases, the simplified model, Equation (5), is exactly the same. No adjustable constants were used in the computations. The simplified model is universal for room airflow simulation.

DISCUSSION

Table 1 shows the total grid number used in the four cases by the simplified model and the CFD models. It also shows the memory needed and CPU time used. The convergence residuals are the same between the simplified and CFD computations. The residuals, R, are defined as:

0 . 3 0

3.00

(a)   (d)

(b)   (e)

Fig. 9. Comparison of the airflow patterns and distribution of air temperature ('C) and helium concentration (%): (a), (b), and (c) simplified method and (d), (e), and (1) CFD method

14

(f)

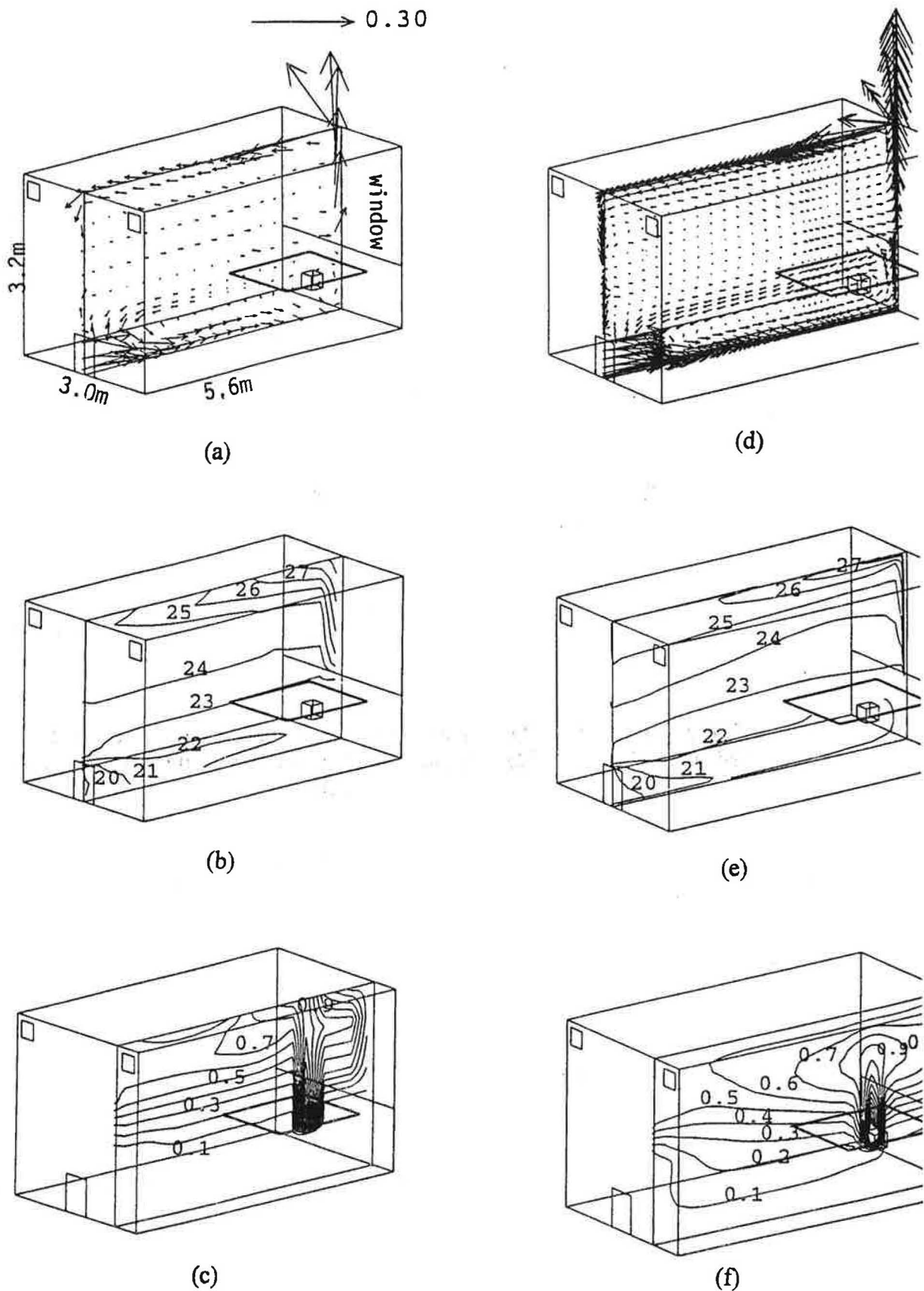Fig. 9. Comparison of the airflow patterns and distribution of air temperature (°C) and helium concentration (%): (a), (b), and (c) simplified method and (d), (e), and (f) CFD method.

14

0.15

Simplified Mode

i ---- ---------------- CF1) Model 0. Iffi-'11,

Data

0.05

0.   09.0,   1,0   2. 1 0   3.   1 0
H(M)

28   . 1 1
                                Simplified Mo
26   CFD Model
                                Data
24 -

22

20
1.0 H(m)   2.0   3.0

1.0

Simplified Model
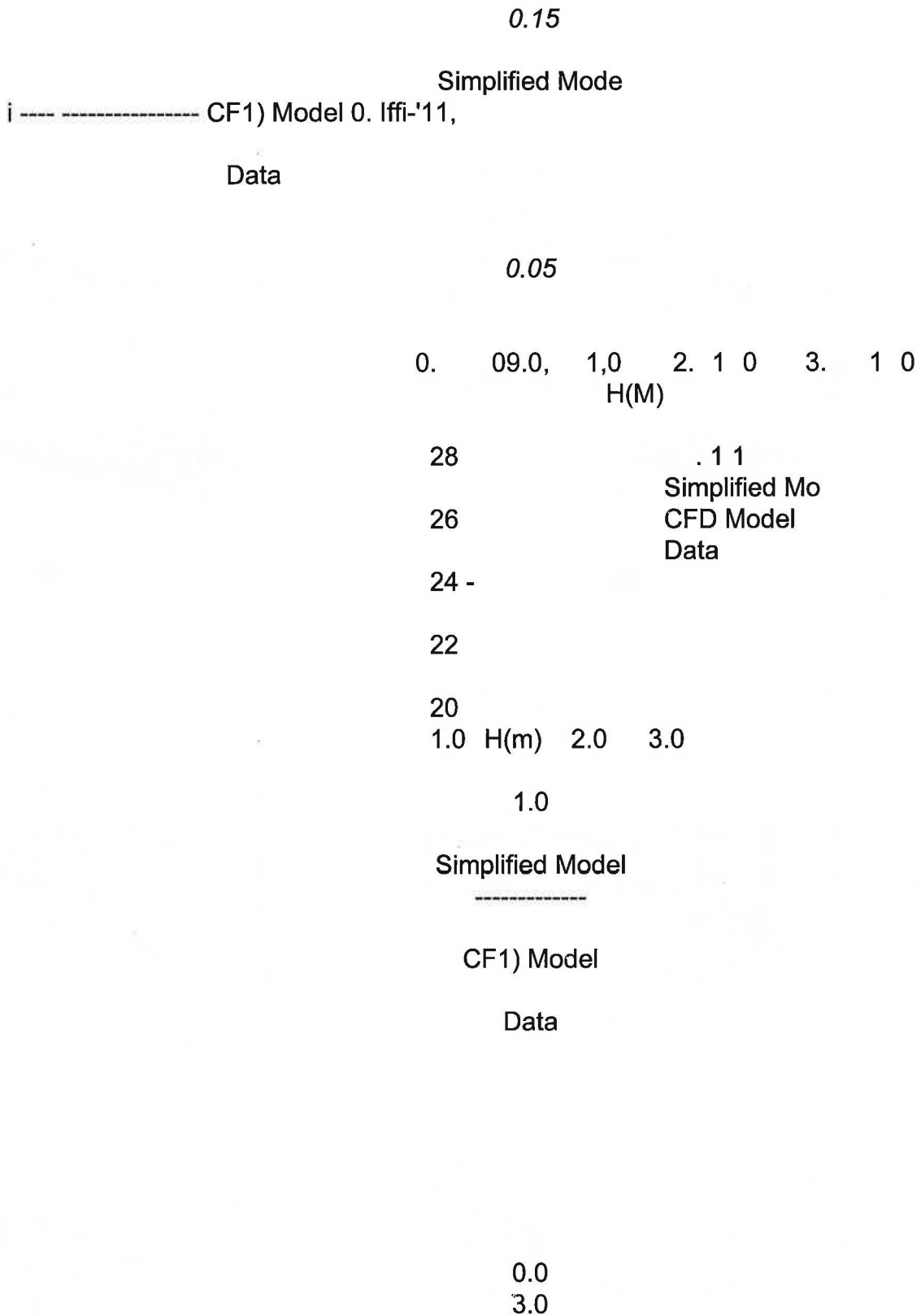

CF1) Model

Data

0.0
3.0

Fig. 1 0. Comparison of the profiles of air velocity, air temperature, and helium concentration in a vertical line of the room.
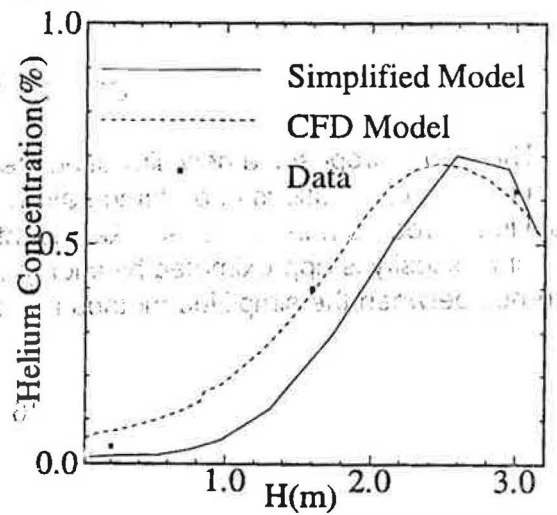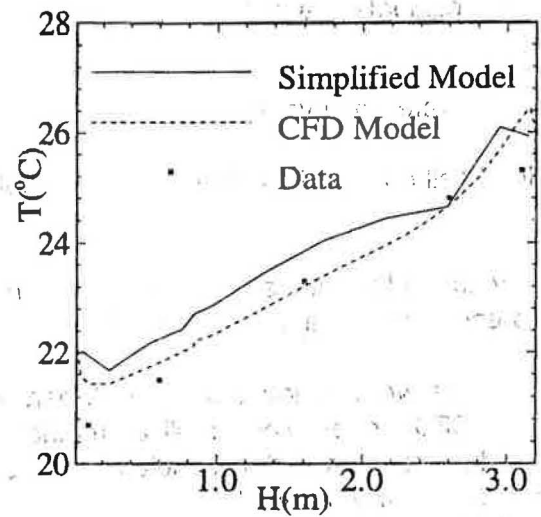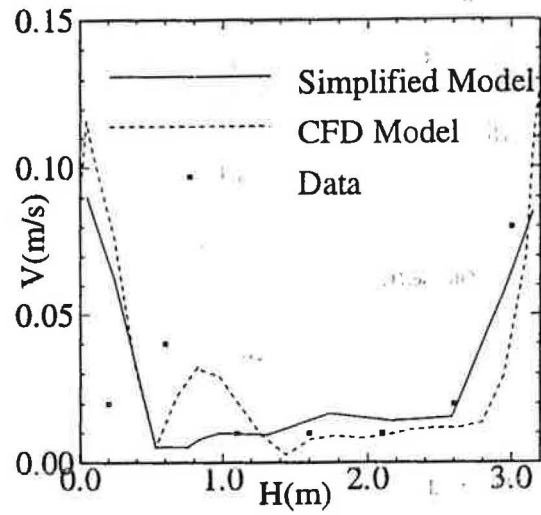
Fig. 10. Comparison of the profiles of air velocity, air temperature, and helium concentration in a vertical line of the room.

15

Table 1. Comparison *of computizg performance* (@f the *simplified* and CFD *models*

Case	Model	Grid number	Core memory	CPU time (sec
Natural	Simplified	20 x 10	*15,000*	18
	CFD	96 x 60	*158,000*	3,238
Forced	Simplified	20 x 18	25,000	9
	CFD	50 x 45	177,000	593
Mixed	Simplified	25 x 18	31,000	33
	CFD	70 x 45	263,000	1,438
		31 x 28 x 26	555,000	5,400
	Simplified	16 x 14 x 12	75,000	311
Displacement		10 X 10 X 10	27,000	119
		6 x 7 x 6	9,000	33
	CFD	31 x 28 x 26	770,000	58, 1'63

NXWNZI

Y_ Y, Y, 1residuals in a cell

R 1 j=|k=I

$$\text{reference value} \qquad (15)$$

where NX = total cell number in x direction NY = total cell number in y direction NZ = total cell number direction

The reference value is the total air supply rate for rnass continuity and heat from a heated/cooled wal energy. The present investigation uses R < 0.001 for mass continuity and R< 0.0 1 for energy.

The computations were conducted in a 486 personal computer. The simplified model uses much memory than the CFD model. The simplified method is at least 10 times faster than the CFD method. results show that most room airflow simulation can be done with a personal computer and the compu time for each case is in the order W
of a few seconds for a two-dimensional problem and a few minutes for a three~dimensional case.

CONCLUSIONS

This paper proposes a new simplified method for the prediction of room airflow pattern and the distributions of air temperature and contaminant concentrations. The model is derived from the Navier-Stokes equations. Using the concept of eddy-viscosity, turbulent viscosity is approximated by a length scale and mean velocity. The main difference between the simplified method and the conventional CFD approach with a k-F-

16

Table 1. Comparison of computing performance of the simplified and CFD models

Case	Model	Grid number	Core memory	CPU time (sec)
Natural	Simplified	20 x 10	15,000	18
	CFD	96 x 60	158,000	3,238
Forced	Simplified	20 x 18	25,000	9
	CFD	50 x 45	177,000	593
Mixed	Simplified	25 x 18	31,000	33
	CFD	70 x 45	263,000	1,438
Displacement	Simplified	31 x 28 x 26	555,000	5,400
		16 x 14 x 12	75,000	311
		10 x 10 x 10	27,000	119
		6 x 7 x 6	9,000	33
	CFD	31 x 28 x 26	770,000	58,163

$$R = \frac{\sum_{i=1}^{NX}\sum_{j=1}^{NY}\sum_{k=1}^{NZ} |residuals \ in \ a \ cell|}{reference \ value} \qquad (15)$$

where NX = total cell number in x direction
NY = total cell number in y direction
NZ = total cell number in z direction

The reference value is the total air supply rate for mass, continuity and heat from a heated/cooled wall for energy. The present investigation uses $R < 0.001$ for mass continuity and $R < 0.01$ for energy.

The computations were conducted in a 486 personal computer. The simplified model uses much less memory than the CFD model. The simplified method is at least 10 times faster than the CFD method. The results show that most room airflow simulation can be done with a personal computer and the computing time for each case is in the order of a few seconds for a two-dimensional problem and a few minutes for a three-dimensional case.

CONCLUSIONS

This paper proposes a new simplified method for the prediction of room airflow pattern and the distributions of air temperature and contaminant concentrations. The model is derived from the Navier-Stokes equations. Using the concept of eddy-viscosity, turbulent viscosity is approximated by a length scale and mean velocity. The main difference between the simplified method and the conventional CFD approach with a k-ε

model is that the former does not solve transport equations for turbulent quantities. The simplified method use a new zero-equation model.

The study demonstrates the capability of the simplified model by applying it to predict the airflow with natural convection, forced convection, mixed convection, and displacement ventilation in rooms. The predicted results are compared with experimental data and the results of CFD simulations. The simplified method can predict reasonably good indoor airflow patterns and the distributions of air temperature and contaminant concentrations.

Since the simplified model does not solve transport equations for turbulence, the computer memory needed is much smaller, and the convergence speed is 10 times faster than that with a CFD model. With the simplified model, simulation of a three-dimensional, steady-state flow in a room can be made in a personal computer. In addition, the user does not need the knowledge of turbulence modeling.

ACKNOWLEDGMENT

REFERENCES

Chen, Q. 1997. "Computational fluid dynamics for HVAC: successes and failures," *ASHRAE Trans, 103(1).*

Incard, C. Bouia, H., and Dalicieux, P. 1996. "Prediction of air temperature distribution in
    buildings with a zonal model," *Energy and Buildings,* 24, 125-132.
Lam, C.K.G. and Brernhorst, K. 1981. "A modified form of the k-F- model for predicting
    wall turbulence," *ASME J. Fluid Engineering,* 103, 456-460.
Launder, B.E. and Spalding, D.B. 1974. ---The numerical computation of turbulent flows,"
    Comp. Meth. Appl. Meeh. Energy, 3, 269-289.
Lebrun, J. and Ngendakurnana, Ph. 1987. "Air circulation induced by heating emitters
    and corresponding heat exchanges along the walls: test room results and
    modelling,"
    Proc. ROOMVENT'87, Session 2a, Paper 6, Stockholm, Sweden.
Olson, D.A. and Glicksman, L.R. 1991. '7ransient natural convection in enclosures at
    high Rayleighnumber,"ASMEJ. *Heat Transfer,* 113, 635-642.
Prandd, L. 1926. "Ueber die ausgebidete Turbulenz," *Proc. 2nd Int. Congress for Appl.
    Mech.,* pp. 62-74, Zurich, Switzerland.

Restivo, A. 1979. *Turbulent Flow in Ventilated Rooms,* Ph.D. Thesis, University of
    London,U.K.

Schwenke, H. 1975. "Ueber das Verhalten elener horizontaler Zuluftstrahlen im begrenzten Raurn," *Luft- und Kaltetechnik, 5, 241-246.*

1 7

Walton, G.N. 1989. "Airflow network models for element-based building airflow modeling," *ASHRAE Trans,* 95(2), 611-620.
Wurtz, E. and Nataf, **LM.** 1994. 'Talidation des modeles zonaux decrits par 1'envirronment oriente objet SPARK," *Proc. European Conf on Energy Performance and Indoor Climate in Buildings,* pp. 785-790, Lyon, France.

1 8