THE USE OF CFD TOOLS FOR INDOOR ENVIRONMENTAL DESIGN

Qingyan (Yan) Chen[*]
Professor of Mechanical Engineering
Ray W. Herrick Laboratories
School of Mechanical Engineering
Purdue University
585 Purdue Mall, West Lafayette, IN 47905-2088, USA
Email: yanchen@purdue.edu

Zhiqiang (John) Zhai
Assistant Professor of Architectural Engineering
Department of Civil, Environmental & Architectural Engineering
University of Colorado
428 UCB, Engineering Center, Room ECOT-441, Boulder, CO 80309-0428 USA
Email: John.Zhai@Colorado.edu

ABSTRACT

This paper presents a short review of the applications of CFD to indoor environment design and studies, and briefly introduces the most popular CFD models used. The paper concludes that, although CFD is a powerful tool for indoor environment design and studies, a standard procedure must be followed so that the CFD program and user can be validated and the CFD results can be trusted. The procedure includes the use of simple cases that have basic flow features interested and experimental data available for validation. The simulation of indoor environment also requires creative thinking and the handling of complex boundary conditions. It is also necessary to play with the numerical grid resolution and distribution in order to get a grid-independent solution with reasonable computing effort. This investigation also discusses issues related to heat transfer. It is only through these incremental exercises that the user and the CFD program can produce results that can be trusted and used for indoor environment design and studies.

Keywords: Computational Fluid Dynamics (CFD), Air distribution, Indoor environment, Experimental validation

NOMENCLATURE

A_o	Effective area of a diffuser	$\dot{m}$	Mass flow rate (kg/s)
C	Smagorinsky model coefficient	p	air pressure (Pa)
C_{SGS}	Smagorinsky model constant	S_{ij}	strain rate tensor (1/s)
$C_{\varepsilon 1}$	coefficient in k-ε model	t	time (s)
$C_{\varepsilon 2}$	coefficient in k-ε model	U_i, U_j	averaged air velocity components in the x_i and x_j directions (m/s)
C_μ	coefficient in k-ε model	U_o	face velocity at a diffuser
k	kinetic energy (J/kg)	u_i, u_j	air velocity components in the x_i and x_j directions (m/s)

1

P	averaged air pressure (Pa)	x_i, x_j	coordinates in i and j directions (m)

Greek Symbols

Δ	filter width (m)	ρ	air density (kg/m^3)
δ	Kronecker delta	σ_k	Prandlt number for k
ε	dissipation rate of kinetic energy (W/kg)	σ_ε	Prandlt number for ε
ν	air kinematic viscosity (m^2/s)	τ_{ij}	subgrid-scale Reynolds stresses (m^2/s^2)
ν_{SGS}	subgrid-scale eddy viscosity (m^2/s)	ϕ	scalar variables
ν_t	turbulent air kinematic viscosity (m^2/s)	Φ	averaged scalar variables

Superscripts

-	grid filtering or Reynolds averaging	'	fluctuating component of a variable

INTRODUCTION

Since human beings spend more than 90% of their time indoors in developed countries, design of indoor environment is crucial to the comfort and welfare of the building occupants. However, this is not an easy task. Woods (1989) reported that about 800,000 to 1,200,000 commercial buildings in the United States containing 30 to 70 million workers have had problems related to the indoor environment. If the problems can be fixed through technologies, Fisk (2000) estimated that for the U.S., the potential annual savings and productivity could be $15 to $40 billion from reduced sick building syndrome symptoms, and $20 to $200 billion from direct improvements in worker performance that are unrelated to health.

In addition, building safety is a major concern of building occupants. Smoke and fire has claimed hundreds of lives every year in the United States. After the anthrax scare following the September 11, 2001 attacks in the United States, how to protect buildings from terrorist attacks by releasing chemical/biological warfare agents becomes another major issue of building safety concerns.

In the past few years, Computational Fluid Dynamics (CFD) has gained popularity as an efficient and useful tool in the design and study of indoor environment and building safety, after having been developed for over a quarter of a century. The applications of CFD in indoor environment and building safety are very wide, such as some of the recent examples for natural ventilation design (Carriho-da-Graca et al. 2002), prediction of smoke and fire in buildings (Lo et al. 2002 and Yeoh et al. 2003), particulate dispersion in indoor environment (Quinn et al. 2001), building element design (Manz 2003), and even for space indoor environment analysis (Eckhardt and Zori 2002). Some other applications are more complicated and may deal with solid materials, and may integrate other building simulation models. Recent examples are the study of building material emissions for indoor air quality assessment (Murakami et al. 2003, Huang and Haghighat 2002, and Topp et al. 2001) and for more accurate building energy and thermal comfort simulations (Zhai and Chen 2003, Bartak et al. 2002, and Beausoleil-Morrison 2002). Often, the outdoor

environment has a significant impact on the indoor environment, such as in buildings with natural ventilation. To solve problems related to natural ventilation requires the study of both the indoor and outdoor environment together, such as simulations of outdoor airflow and pollutant dispersion (Sahm et al. 2002 and Swaddiwudhipong and Khan 2002) and combined indoor and airflow studies (Jiang and Chen 2002). CFD is no longer a patent for users with Ph.D. degrees. Tsou (2001) has developed online CFD as a teaching tool for building performance studies, including issues such as structural stability, acoustic quality, natural lighting, thermal comfort, and ventilation and indoor air quality.

Compared with experimental studies of indoor environment and building safety, CFD is less expensive and can obtain results much faster, due to the development in computing power and capacity as well as turbulence modeling. CFD can be applied to test flow and heat transfer conditions where experimental testing could prove very difficult, such as in space vehicles (Eckhardt and Zori 2002). Even if experimental measurements could be conducted, such an experiment would normally require hundreds of thousands dollars and many months of workers' time (Yuan et al. 1999).

However, CFD results cannot be always trusted, due to the assumptions used in turbulence modeling and approximations used in a simulation to simplify a complex real problem of indoor environment and building safety. Although a CFD simulation can always give a result for such a simulation, it may not necessarily give the correct result. A traditional approach to examine whether a CFD result is correct is by comparing the CFD result with corresponding experimental data. The question now is whether one can use a robust and validated CFD program, such as a well-known commercial CFD program, to solve a problem related to indoor environment and building safety without validation. This forms the main objective of the paper.

COMPUTATIONAL FLUID DYNAMICS APPROACHES

Indoor environment consists of four major components: thermal environment, indoor air quality, acoustics, and lighting environment. Building thermal environment and indoor air quality include the following parameters: air temperature, air velocity, relative humidity, environmental temperature, and contaminant and particulate concentrations, etc. The parameters concerning building safety are air temperature, smoke (contaminant and particulate) concentrations, flame temperature, etc. Obviously, normal CFD programs based on Navier-Stokes equations and heat and mass transfer cannot be used to solve acoustic and lighting components of an indoor environment. However, the CFD programs can be used to deal with problems associated with thermal environment, indoor air quality, and building safety, since the parameters are solved by the programs. Hereafter, the paper will use indoor environment to narrowly refer to thermal environment, indoor air quality, and building safety.

Almost all the flows in indoor environment are turbulent. Depending on how CFD solves the turbulent flows, it can be divided into direct numerical simulation, large eddy simulation (LES), and the Reynolds averaged Navier-Stokes equations with turbulence models (hereafter denotes as RANS modeling).

Direct numerical simulation computes turbulent flow by solving the highly reliable Navier-Stokes equation without approximations. Direct numerical simulation requires a very fine grid resolution to capture the smallest eddies in the turbulent flow at very small time steps, even for a steady-state flow. Direct numerical simulation would require a fast computer that currently does not exist and would take years of computing time for predicting indoor environment.

Large eddy simulation (Deardorff 1970) separates turbulent motion into large eddies and small eddies. This method computes the large eddies in a three-dimensional and time dependent way while it estimates the small eddies with a subgrid-scale model. When the grid size is sufficiently

small, the impact of the subgrid-scale models on the flow motion is negligible. Furthermore, the subgrid-scale models tend to be universal because turbulent flow at a very small scale seems to be isotropic. Therefore, the subgrid-scale models of LES generally contain only one or no empirical coefficient. Since the flow information obtained from subgrid scales may not be as important as that from large scales, LES can be a general and accurate tool to study engineering flows (Piomelli 1999 and Lesieur and Metais 1996). LES has been successfully applied to study airflow in and around buildings (Emmerich and McGrattan 1998, Thomas and Williams 1999, Murakami et al. 1999, Jiang and Chen 2002, Kato et al. 2003). Although LES requires a much smaller computer capacity and is much faster than direct numerical simulation, LES for predicting indoor environment demands a large computer capacity (10^{10} byte memory) and a long computing time (days to weeks).

The Reynolds averaged Navier-Stokes equations with turbulence models solve the statistically averaged Navier-Stokes equations by using turbulence transport models to simplify the calculation of the turbulence effect. The use of turbulence models leads to some errors, but can significantly reduce the requirement in computer memory and speed. The RANS modeling provides detailed information on indoor environment. The method has been successfully applied to the building indoor airflow and thermal comfort and indoor air quality analysis, as reviewed by Ladeinde and Nearon (1997) and Nielsen (1998). The RANS modeling can be easily used to study indoor environment. It would take only a few hours of computing time in a modern PC, should the RANS modeling be used to study a reasonable size of indoor environment.

In order to better illustrate the LES and RANS modeling, the following sections will discuss the fundamentals of the two CFD approaches. For simplicity, this paper only discusses how the two CFD approaches solve Navier-Stokes equations and the continuity equation. Namely, the flow in indoor environment is considered to be isothermal and no gaseous and particulate contaminants and chemical reactions, are taken into account. In fact, temperature (energy), various contaminants, and various chemical reactions are solved in a similar manner.

Large eddy simulation

By filtering the Navier-Stokes and continuity equations in the LES approach, one would obtain the governing equations for the large-eddy motions as

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\overline{u_i u_j}) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{2}$$

where the bar represents grid filtering. The subgrid-scale Reynolds stresses, τ_{ij}, in Eq. (1),

$$\tau_{ij} = \overline{u_i u_j} - \overline{u}_i \overline{u}_j \tag{3}$$

are unknown and must be modeled with a subgrid-scale model. Numerous subgrid-scale models have been developed in the past thirty years. The simplest and probably the most widely used is the Smagorinsky subgrid-scale model (Smagorinsky 1963) since the pioneering work by Deardorff (1970). The model assumes that the subgrid-scale Reynolds stress, τ_{ij}, is proportional to the strain rate tensor,

$$\overline{S}_{ij} = \frac{1}{2}(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}) \tag{4}$$

$$\tau_{ij} = -2\nu_{SGS}\overline{S}_{ij} \tag{5}$$

where the subgrid-scale eddy viscosity, ν_{SGS}, is defined as

$$\nu_{SGS} = (C_{SGS}\Delta)^2 (2\overline{S}_{ij} \cdot \overline{S}_{ij})^{\frac{1}{2}} = C\Delta^2 (2\overline{S}_{ij} \cdot \overline{S}_{ij})^{\frac{1}{2}} \tag{6}$$

The Smagorinsky constant, C_{SGS}, ranges from 0.1 to 0.2 determined by flow types, and the model coefficient, C, is the square of C_{SGS}. The model is an adaptation of the mixing length model of RANS modeling to the subgrid-scale model of LES.

RANS modeling

Reynolds (1895) introduced the Reynolds-averaged approach in 1895. He decomposed the instantaneous velocity and pressure and other variables into a statistically averaged value (denoted with capital letters) and a turbulent fluctuation superimposed thereon (denoted with ′ superscript). Taking velocity, pressure, and a scale variable as examples:

$$u_i = U_i + u_i' \qquad p = P + p' \qquad \phi = \Phi + \phi' \tag{7}$$

The statistical average operation on the instantaneous, averaged, and fluctuant variables have followed the Reynolds average rules. Taking velocity as an example, the Reynolds average rules can be summarized as:

$$\overline{u}_i = \overline{U}_i = U_i \qquad \overline{u_i'} = 0 \quad \overline{u_i'U_j} = 0 \qquad \overline{u_i + u_j} = U_i + U_j \qquad \overline{u_i u_j} = U_i U_j + \overline{u_i' u_j'} \tag{8}$$

Note that the bars in Eq. (8) stand for "statistical average" and are different from those used for LES. In LES, those bars represent grid filtering.

By applying the Reynolds averaging method to the Navier-Stokes and continuity equation, they become:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial U_i}{\partial x_j} - \overline{u_i' u_j'}\right) \tag{9}$$

$$\frac{\partial U_i}{\partial x_i} = \frac{\partial u_i'}{\partial x_i} = 0 \tag{10}$$

where $\overline{u_i' u_j'}$ is the Reynolds stress that is unknown and must be modeled. In the last century, numerous turbulence models have been developed to represent $\overline{u_i' u_j'}$. Depending on how the Reynolds stress is modeled, RANS turbulence modeling can be further divided into Reynolds stress models and eddy viscosity models. For simplicity, this paper discusses only eddy-viscosity

turbulence models that adopt the Boussinesq approximation (1877) to relate Reynolds stress to the rate of mean stream through an "eddy" viscosity ν_t.

$$\overline{u_i' u_j'} = \frac{2}{3} \delta_{ij} k - \nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \tag{11}$$

where δ_{ij} is the Kronecker delta (when i≠j, δ_{ij}=0; and when i=j, δ_{ij}=1), and k is the turbulence kinetic energy ($k = \dfrac{\overline{u_i' u_i'}}{2}$). Among hundreds of eddy viscosity models, the standard k-ε model (Launder and Spalding 1974) is most popular. The standard k-ε model solves eddy viscosity through

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{12}$$

where $C_\mu = 0.09$ is an empirical constant. The k and ε can be determined by solving two additional transport equations:

$$U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P - \varepsilon \tag{13}$$

$$U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \left[C_{\varepsilon 1} P - C_{\varepsilon 2} \varepsilon \right] \frac{\varepsilon}{k} \tag{14}$$

where,

$$P = \nu_t \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)^2 \tag{15}$$

and $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, $C_{\varepsilon 1} = 1.44$, and $C_{\varepsilon 2} = 1.92$ are empirical constants. The two-equation k-ε model is most popular but not the simplest one. The simplest ones are zero-equation turbulence models, such as the constant viscosity model and the one proposed by Chen and Xu (1998). The constant viscosity model and zero-equation models do not solve turbulence quantities by transport equations.

Be it LES or RANS modeling, the above-mentioned equations cannot be solved analytically because they are highly non-linear and inter-related. However, they can be solved numerically on a computer by discretizing them properly with an appropriate algorithm. Many textbooks have been devoted to this topic. Due to limited space available, this paper does not discuss this issue here. Finally, boundary conditions must be specified in order to make the equations solvable for a specific problem of indoor environment.

If one has used a CFD program with the above-mentioned equations and specified boundary conditions for a flow problem, can one trust the results obtained? The following section will use an example to illustrate how one could obtain CFD results for an indoor environment problem and how one could evaluate the correctness of the results.

SIMULATION AND ANALYSIS

The following example is a study of indoor air and contaminant distribution in a room with displacement ventilation, as shown in Fig. 1. The room was 5.16 m long, 3.65 m wide, and 2.43 m high. Cold air was supplied through a diffuser in the lower part of a room, and warm air was exhausted at the ceiling level. The two-person office contained many heated and unheated objects, such as occupants, lighting, computers, and furniture. For this case, Yuan et al. (1999) measured the air temperature, air velocity, and contaminant concentration by using SF_6 as a tracer-gas. The tracer-gas was used to simulate contaminant emissions from the two occupants, such as CO_2. The temperature of the inlet airflow from the diffuser was 17.0°C and the ventilation rate was 183 m^3/h. The total heat sources in the room were 636 W.
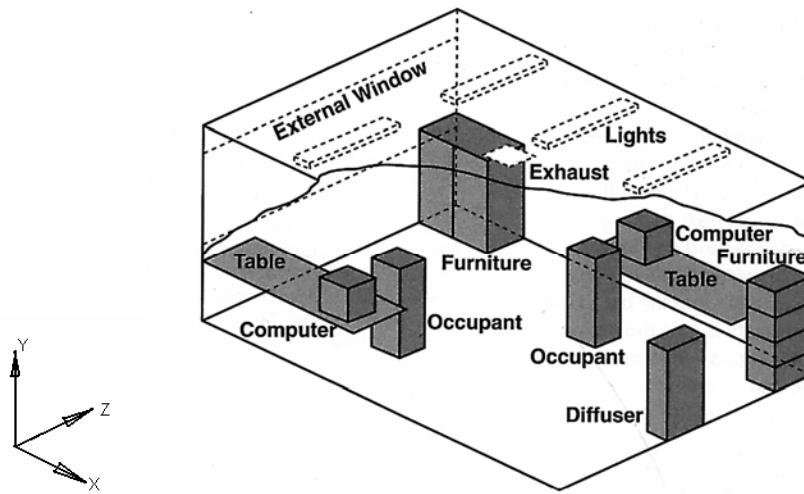
Fig. 1 The schematic of a room with mixed convection flow

General problems in using CFD programs

This is a project the author assigned to train his graduate students in gaining experience and confidence in using a well-validated commercial CFD program. The graduate students majored in mechanical engineering and had sufficient knowledge of fluid dynamics, heat transfer, and numerical methods. Without exception, no student could obtain correct results in the first instance when they attempted to directly solve such a problem. Their CFD results were compared with the experimental data from Yuan et al. (1999). The problems can be summarized as follows:

- Difficulty in selecting a suitable turbulence model
- Incorrect setting of boundary conditions for the air-supply diffuser
- Inappropriate selection of grid resolution
- Failure to estimate correctly convective portion of the heat from the heat sources, such as the occupants, computers, and lighting
- Improper use of numeric techniques, such as relaxation factors and internal iteration numbers

For such a problem as shown in Fig. 1, both the LES and RANS approaches were suitable. Through the RANS approach, many commercial CFD programs offer numerous turbulence models for CFD users. It is a very challenging job for a beginner to decide which model to use. Although

for some cases, more sophisticated models can generate more accurate results, our experience found that the Smagorinsky subgrid-scale model for LES and the standard k-ε model for RANS are more universal, consistent and stable. Unfortunately, they do not always produce accurate results and can perform poorer than other models in some cases.

Simulation of a specific problem of indoor environment requires creative approaches. One typical example is how to simulate the air-supply diffuser, which is a perforated panel with an effective area of less than 10%. Some commercial codes have a library of diffusers that can be used to simulate an array of complex diffusers, such as Airpak from Fluent. Without such a library, we found that only experienced CFD users may know how to simulate such a diffuser.

Since the geometry of the displacement ventilation case is rectangular, many of the students would select a grid distribution that fits the boundaries of the objects in the room. The grid size would be selected in such a way that no interpolation is needed to obtain results in places of interest. Not everyone would refine the grid resolution to obtain grid-independent results. It is hard to obtain grid-independent results, especially when LES is used. When a wall-function is used for boundary layers, it is very rare that a CFD user would check if the grid resolution near a wall is satisfactory.

ASHRAE (Chen and Srebric 2002) has developed a guide on using CFD to simulate indoor environment. One major emphasis is on establishing a CFD model that could simulate a specific problem. If we take the displacement ventilation case as an example, it is not an easy task to provide the thermal and fluid boundary conditions. For example, it is difficult to estimate temperature or heat fluxes for the building enclosure surfaces and the heated objects, such as computers, occupants, and lighting. As a consequence, the mean air temperature computed by different users with the same CFD program can differ as much as 3 K.

Most CFD programs, especially the commercial ones, are generalized and designed to solve flow and heat and mass transfer, not just for simulating indoor environment. As a result, the CFD programs provide many options. A user can fine tune the parameters to obtain a result. The parameters that can be tuned include, but are not limited to, model coefficients, relaxation factors, and iteration numbers. With different tuning values, the CFD results are often not the same.

Therefore, a CFD beginner, who attempted to solve flow and heat and mass transfer for the displacement ventilation case, became frustrated when he/she found that his/her CFD results were different from the measured data. If no measured data were available for comparison, the user would have no confidence about the correctness of the CFD results. In order to correctly perform a CFD simulation for a specific flow problem related to indoor environment, we strongly recommend the use of ASHRAE procedure for verification, validation, and reporting of indoor environment CFD analyses (Chen and Srebric 2002).

How to conduct CFD analyses of indoor environment

To design or study an indoor environment problem with CFD, one needs to

- Confirm the abilities of the turbulence model and other auxiliary models to predict all physical phenomena in the indoor environment
- Confirm the discretization method, grid resolution, and numerical algorithm for the flow simulation
- Confirm the user's ability to use the CFD code to perform indoor environment analyses

The confirmations are indeed a validation process through which a user can know his/her ability to perform a CFD simulation and the correctness of the CFD results. If the user is asked to simulate the displacement ventilation case, no experimental data is available for comparison, as in most

indoor environment designs and studies. The validation would use several subsystems that represent the complete flow, heat and mass transfer features of the case. For the displacement ventilation that has a mixed convection flow, the user may start a two-dimensional natural convection in a cavity and a forced convection in a cavity. Since mixed convection is a combination of natural and forced convection, the two subsystems can represent the basic flow features of the displacement ventilation. Of course, CFD validation is not only for flow type; the CFD validation should done in progressive stages. A typical procedure for correctly simulating the displacement ventilation would be:

- Simulation of a two-dimensional natural convection case
- Simulation of a two-dimensional forced convection case
- Simulation of a simple three-dimensional case
- Simulation of complex flow components
- Change in grid resolution, especially the resolution near walls
- Calculation of convective/radiative ratio for different heat sources
- Simulation of the displacement ventilation

This procedure is incremental in the complexity of the CFD simulations. Since it is relatively easy to judge the correctness of the CFD results for simple cases (many of them have experimental data available in literature), the user can gain confidence in the simulation exercise. While such a simulation seems to take longer time than direct simulation of the displacement ventilation, the procedure is more effective and can actually obtain the correct results for the displacement ventilation, rather than directly solving the case without the basic exercise. This is because the CFD user would have a hard time to find out where the simulation has gone wrong, due to the complexity of the displacement ventilation and inexperience in usesage of the CFD program.

The following sections illustrate the simulation procedure.

Simulation of a two-dimensional natural convection case

The two-dimensional natural convection case concerns flow in a cavity of 0.5m width and 2.5m height, as shown in Fig. 2. Cheesewright et al. (1986) conducted the experimental studies on this case. The experiment maintained isothermal conditions (64.8°C and 20°C) on the two vertical walls and insulated the two horizontal walls, even though they were not ideally insulated. The Rayleigh number (Ra) based on the cavity height (h) was $5x10^5$. The simulation employed both the zero-equation model (Chen and Xu 1998) and the standard k-ε model.
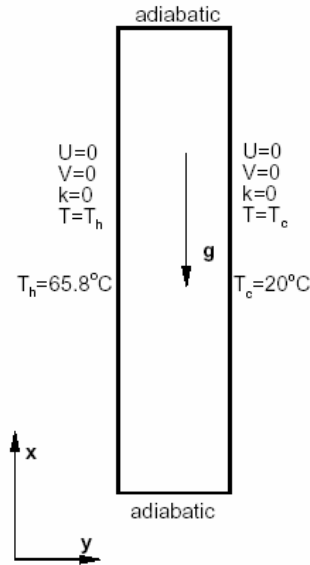
Fig. 2 Geometry and boundary conditions for 2-D natural convection in a cavity

Fig. 3(a) compares the computed and measured mean velocity at the mid-height of the cavity, which shows good agreement except at the near-wall regions. The standard k-ε model with the wall function appears to capture the airflows near the surfaces better than the zero-equation model. The predicted core air temperatures with the k-ε model, as shown in Fig. 3(b), also agree well with Cheesewright's measurements. The results with the zero-equation model are higher than the measurements, although the computed and measured temperature gradients in the core region are similar.

A beginner may not be able to find the reasons for the discrepancies. With the use of two models, it is possible to find that different models do produce different results.

(a)   (b)
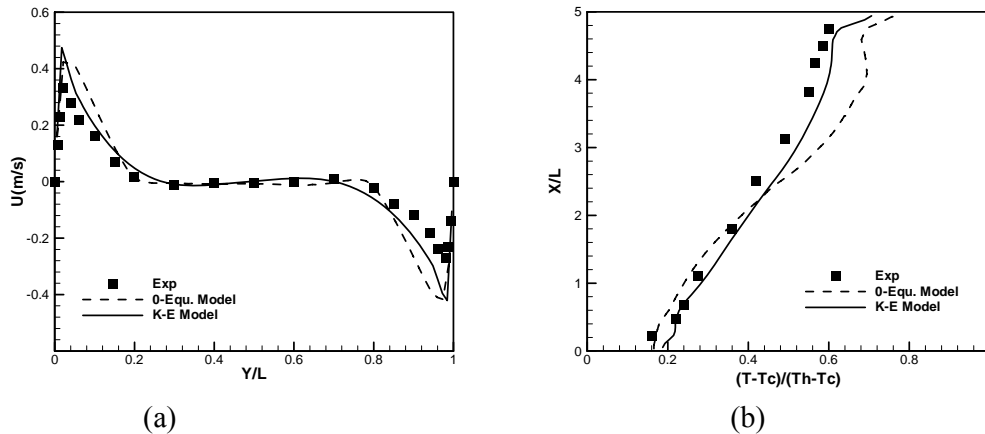
Fig. 3 The vertical velocity profile at mid-height (a) and temperature profile (b) in the mid-height for 2-D natural convection case

Since displacement ventilation consists of natural and forced convection, it is necessary to simulate a forced convection in order to assess the performance of the turbulence models. A case proposed by Nielsen et al. (1974) with experimental data is most appropriate. Due to limited space

10

available, this paper does not report the simulation results. In fact, the zero-equation model and the k-ε model have performed similarly for the two-dimensional forced convection case as they did for the natural convection case reported above.

Simulation of a three-dimensional case without internal obstacles

The next step is to simulate a three-dimensional flow. As the problem becomes more complicated, the experimental data often becomes less detailed and less reliable in terms of quality. Fortunately, with the experience of the two-dimensional flow simulation, the three-dimensional case selection is not critical. For example, the experimental data of mixed convection in a room as shown in Fig. 4 from Fisher (1995) seems appropriate for this investigation.
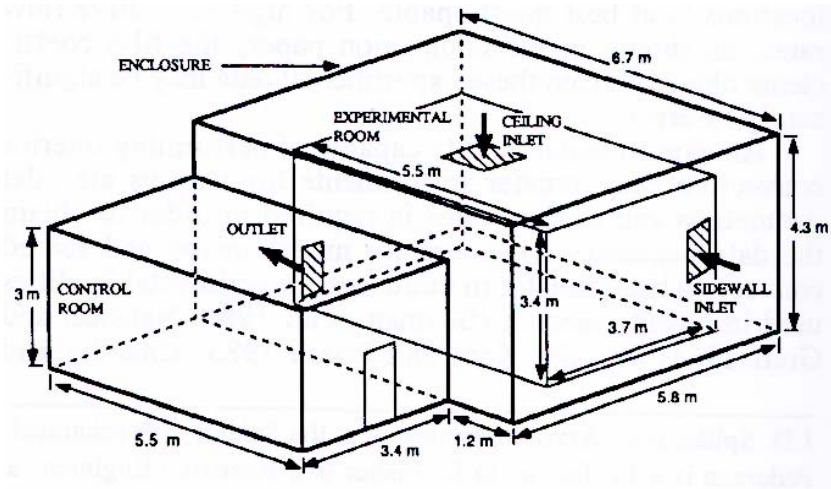
Fig. 4 Schematic of experimental facility (Fisher 1995)

Fig. 5 presents the measured and calculated air speed contours, which show the similarity between the measurement and simulation of the primary airflow structures. The results show that the jet dropped down to the floor of the room after traveling forward for a certain distance due to the negative buoyancy effect. This comparison is not as detailed quantitatively as the two-dimensional natural convection case. However, a CFD user would gain some confidence in his/her results through this three-dimensional simulation.
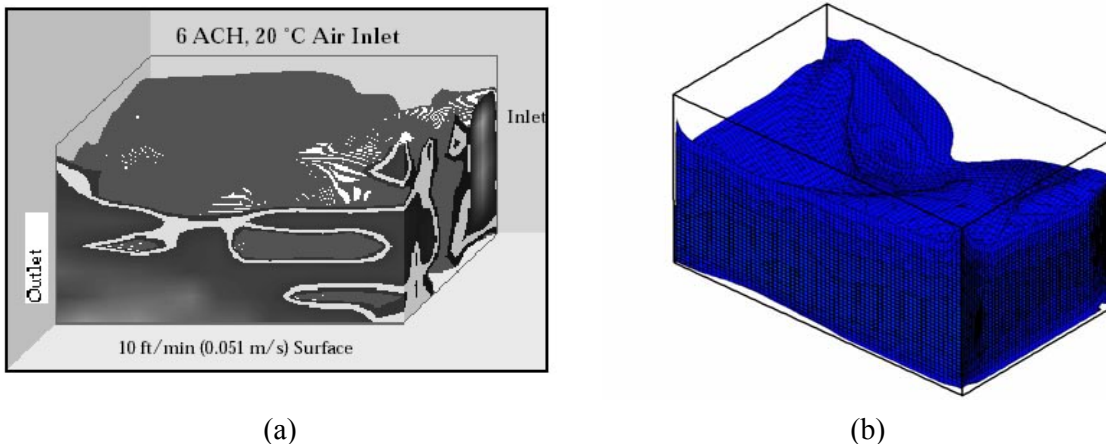

(a)   (b)

Fig. 5 Air speed contour in the room (a) as measured; (b) as simulated by CFD.

Simulation of complex flow components

A room normally consists of several complex flow elements, such as air-supply diffusers, irregular heat sources, and complicated geometry. Correct modeling of these flow components is essential for achieving accurate simulation of airflow in the room. This paper takes an air supply diffuser used for displacement ventilation as an example for illustrating how the complex flow components should be modeled.

Fig. 6 shows the flow development in front of a displacement diffuser. The jet drops immediately to the floor in the front of the diffuser because of the low air supply velocity and buoyancy effect. The jet then spreads over the floor and reaches the opposite wall. In front of the diffuser, the jet velocity profile changes along its trajectory. Close to the diffuser, no jet formula can be used since the jet is in a transition region. Only after 0.9 m (3.0 ft) does the jet form an attached jet, where a jet formula could be used. However, jet formulae can only predict velocities in the jet region that is less than 0.2 m above the floor, because the velocities above the region are influenced by the room conditions. In fact, the velocity profile above the jet region represents the backward airflow towards the displacement diffuser.
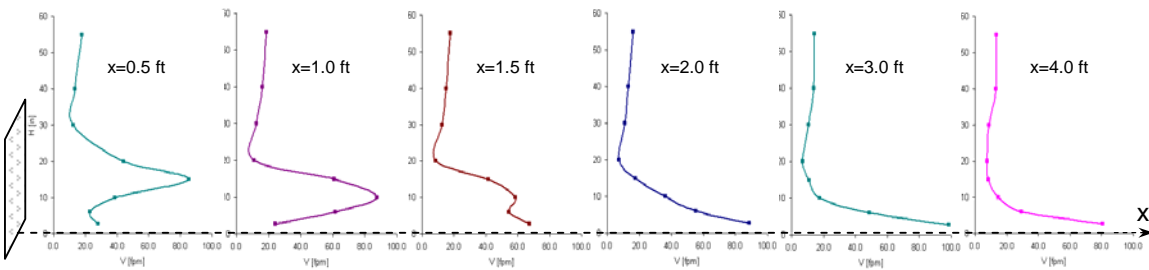

Fig. 6 Development of the wall jet in front of the displacement diffuser

Chen and Moser (1991) proposed a momentum method that de-couples momentum and mass boundary conditions for the diffuser in CFD simulation. The diffuser is represented in the CFD study with an opening that has the same gross area, mass flux, and momentum flux as a real diffuser does. This model enables specification of the source terms in the conservation equations over the real diffuser area. The air supply velocity for the momentum source term is calculated from the mass flow rate, $\dot{m}$, and the diffuser effective area A_0:

$$U_o = \dot{m}/(\rho\, A_o) \tag{16}$$

Srebric (2000) demonstrated that the momentum method can produce satisfactory results, and the method is thus used for this investigation. As one can see, modeling of a complex flow element requires substantial effort and knowledge.

Change in grid resolution, especially the resolution near walls

So far we have discussed the establishment of a CFD model for displacement ventilation. Numerical procedure is equally important in achieving accurate results. In most cases, one would demand a grid-independent solution. By using Fisher's case (1995) as an example, this investigation has used four sets of grids to simulate the indoor airflow: a coarse grid ($22\times17\times15 = 5,610$ cells), a moderate grid ($44\times34\times30 = 44,880$ cells), a fine grid ($66\times51\times45 = 151,470$ cells),

and a locally refined coarse grid (27×19×17 = 8,721 cells) that has the same resolution in the near-wall regions as the fine grid.

Fig. 7 presents the predicted temperature gradient along the vertical central line of the room with the different grid resolutions. Obviously, a coarse grid distribution can not produce satisfactory results. The moderate and fine grid systems produced similar temperature profile and could be considered as grid independent. It is also interesting to know that by using locally refined grid distribution, a coarse grid system can yield satisfactory results.
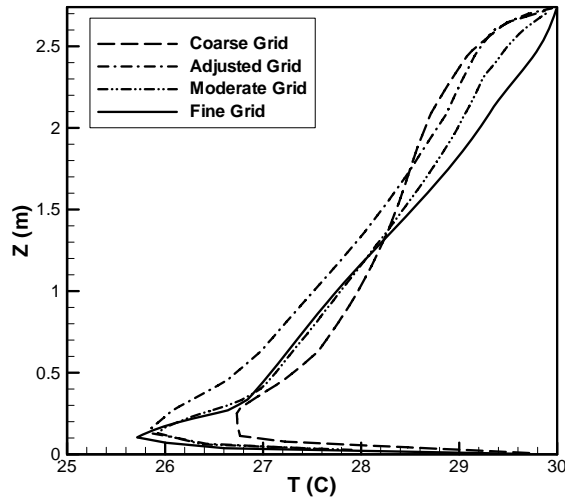

Fig. 7 Predicted temperature gradient along the vertical central line of the room

The grid distribution has a significant impact on the heat transfer. Fig. 8 shows the predicted convective heat fluxes from enclosures with different grid systems. The convective heat fluxes from the floor predicted with the refined grid systems are much closer to the measurement than those with the coarse grid. However, the difference between the measured and simulated results at wall Level 2 is still distinct, even with the fine grid. The analysis indicates that the impact of the high speed jet flow on Level 2 of the north wall is the main reason for the large heat flux at the entire wall Level 2. Since the vertical jet slot is very close to the north wall, the cold airflow from the jet inlet causes the strong shear flow at the north wall, introducing the extra heat transfer at this particular area. The experiment did not measure this heat transfer zone within the inner jet flow. If the north wall was removed from the analysis of the wall convective heat fluxes, the agreement between the computed results and measured data would be much better.
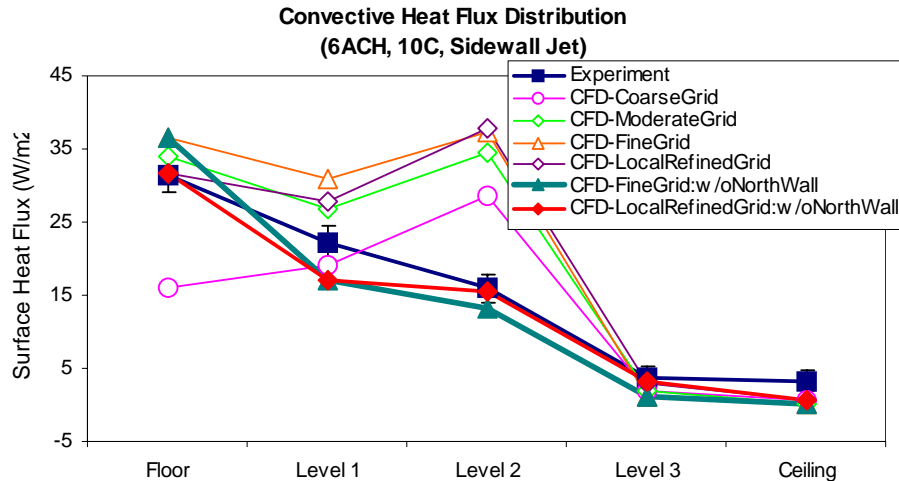
Fig. 8 Comparison of convective heat fluxes from enclosures with various grid resolutions

Fig. 8 also indicates that, instead of using a global refined grid that may need long computing time, a locally refined coarse grid can effectively predict the airflow and heat transfer for such an indoor case. Good resolution for the near-wall regions is much more important than for the inner space because the air temperature in the core of a space is generally more uniform than that in the perimeter of a space.

Calculation of convective/radiative ratio for different heat sources

In most indoor airflow simulations, the building interior surface temperatures are specified as boundary conditions. Then, the heat from heat sources must be split into convective and radiative parts. The convective part is needed as boundary conditions for the CFD simulation, while the radiative part is lumped into the wall surface temperatures. This split can be rather difficult, since the surface temperature of the heat sources and/or the surface area are unknown in most cases. Without a correct split, the final air temperature of the room could deviate a few degrees from the correct one. Therefore, the split would require a good knowledge of heat transfer. This problem will not be discussed in detail here, since it is problem dependent. For the displacement ventilation, the convective/radiative ratio should be 80/20 for occupants, 56/44 for computers, and 60/40 for lighting.

Simulation of displacement ventilation

With all the above exercises, a CFD user would gain sufficient experience in indoor environment simulation by CFD. The user could use CFD to study indoor environment, such as airflow in a room with displacement ventilation (as shown in Fig. 1), with confidence. The results will then be somewhat trusted.

This section shows the CFD results computed by the co-author for the displacement ventilation case (Fig. 1). The experimental data from Yuan et al. (1999) was available for this case. The data is used as a comparison in showing if the CFD results can be trusted.

This investigation used a CFD program with the zero-equation turbulence model and the standard k-ε model. The computational grid is 55×37×29, which is sufficient for obtaining the grid-independent solution, according to Srebric (2000) and our experience in Fisher's case (1995). Fig. 9(a) shows the calculated air velocity and temperature distributions in the middle section of the

room with the zero-equation model. The solutions with the standard k-ε model are fairly similar. The computed results are in good agreement with the flow pattern observed by smoke visualization, as illustrated in Fig. 9(b). The large re-circulation in the lower part of the room, which is known as a typical flow characteristic of displacement ventilation, is well captured by the CFD simulation. The airflow and temperature patterns in the respective sections across a person and a computer, as shown in Figs. 9(c) and (d), clearly exhibit the upward thermal plumes due to the positive buoyancy from the heat sources.

Fig. 9 Velocity and temperature distributions for the displacement ventilation case (a) calculated results in the middle section, (b) observed airflow pattern with smoke visualization in the middle section, (c) calculated results in the section across a computer, and (d) calculated results in the section across an occupant.

The study further compared the measured and calculated velocity, air temperature, and tracer-gas concentration (SF_6 used to simulate bio-effluent from the two occupants) profiles at five locations where detailed measurements were carried out. The locations in the floor plan are illustrated in the lower-right of Figs. 10-12. The figures show the computed results by RANS modeling with the zero-equation model and the standard k-ε model, and large-eddy simulation with the Smogrinsky subgrid-scale model (SSGS).

Clearly, the computed results are not exactly the same as the experimental data. In fact, the two results will never be the same due to the approximations used in CFD and errors in the measuring equipment and experimental rig. The agreement is better for temperature than the velocity and tracer-gas concentration. Since omni-directional anemometers were used to measure air velocity and the air velocity is low, the convection caused by probes would generate a false velocity of the same magnitude. Therefore, the accuracy of the measured velocity is not very high. For tracer-gas concentration, the airflow pattern is not very stable and measuring SF_6 concentration at a single point would take 30 seconds. The measurement has a great uncertainty as well.

On the other hand, the performance of the CFD models is also different. The LES results seem slightly better than the others. Since LES uses at least one-order magnitude computing time than the RANS modeling, LES seems not worth in such an application. The profile curves are not very smooth that may indicate more averaging time needed.

Nevertheless, the CFD results do reproduce the most important features of airflow in the room, and can quantitatively predict the air distribution. The discrepancies between the computed results and experimental data can be accepted for indoor environment design and study. We may conclude that the CFD results could be trusted for this case even if no experimental data were available for validation.
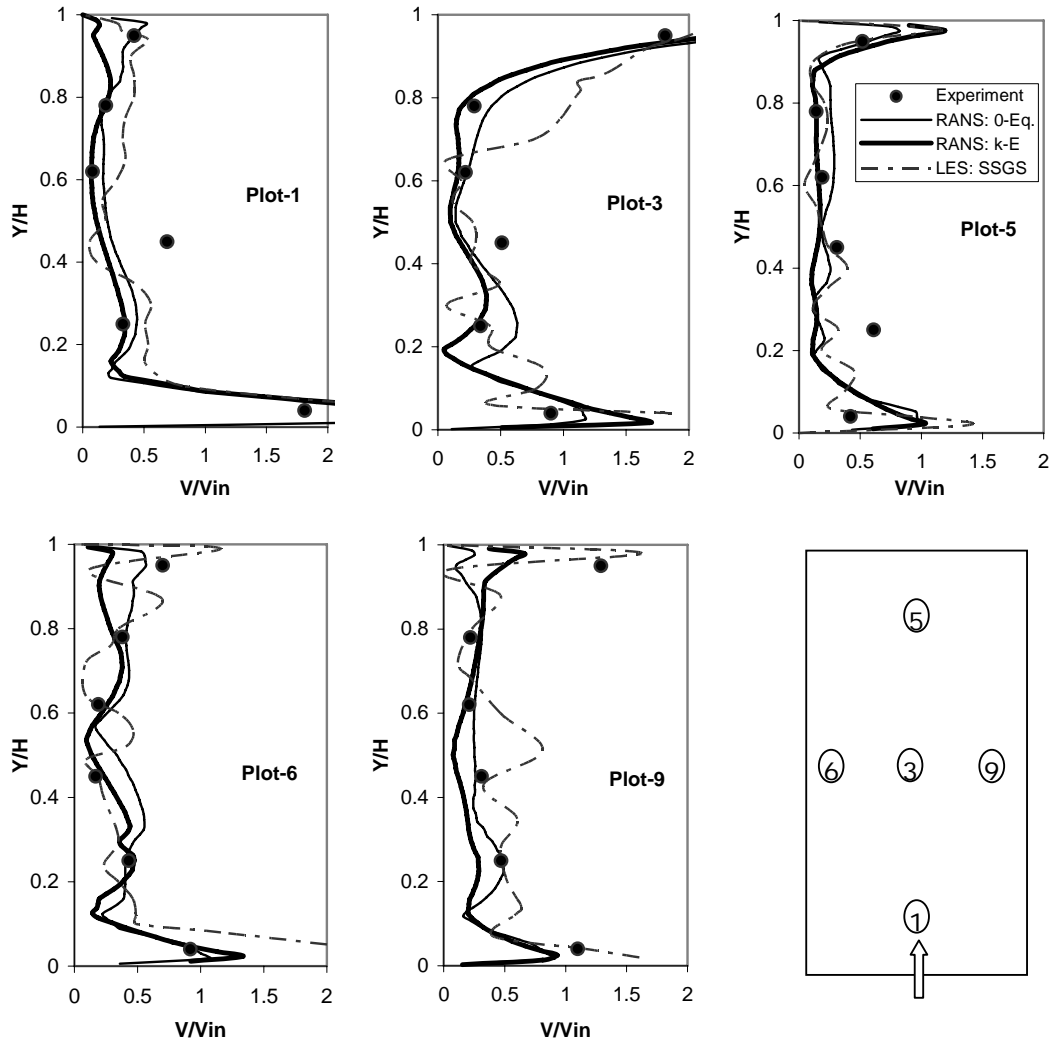
Fig. 10 The comparison of the velocity profiles at five positions in the room between the calculated and measured data for the displacement ventilation case. Z=height/total room height (H), V=velocity/inlet velocity (V$_{in}$), H=2.43m, V$_{in}$=0.086m/s
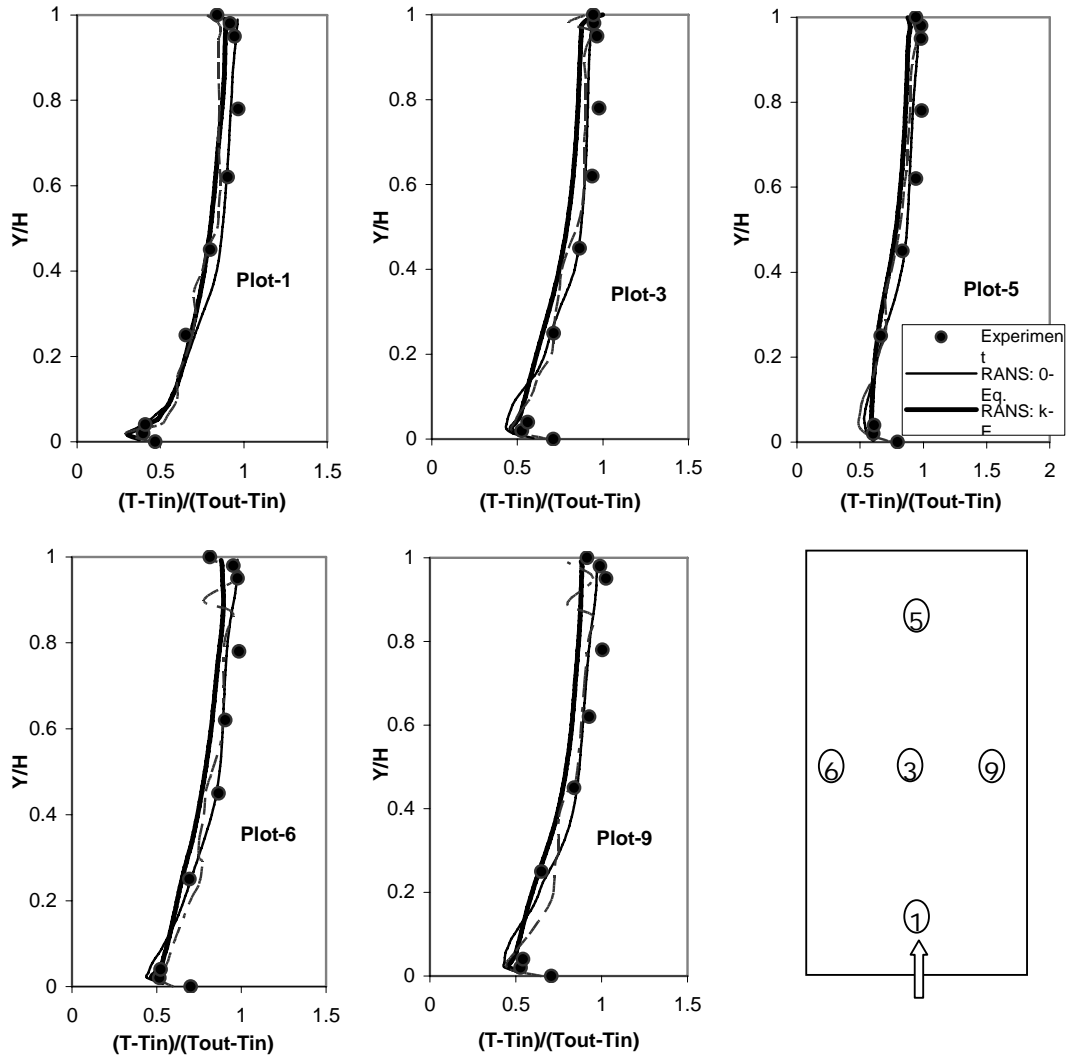
Fig. 11 The comparison of the temperature profiles at five positions in the room between the calculated and measured data for the displacement ventilation case. Z=height/total room height (H), T=(T_{air}-T_{in}/T_{out}-T_{in}), H=2.43m, T_{in}=17.0^{o}C, T_{out}=26.7^{o}C
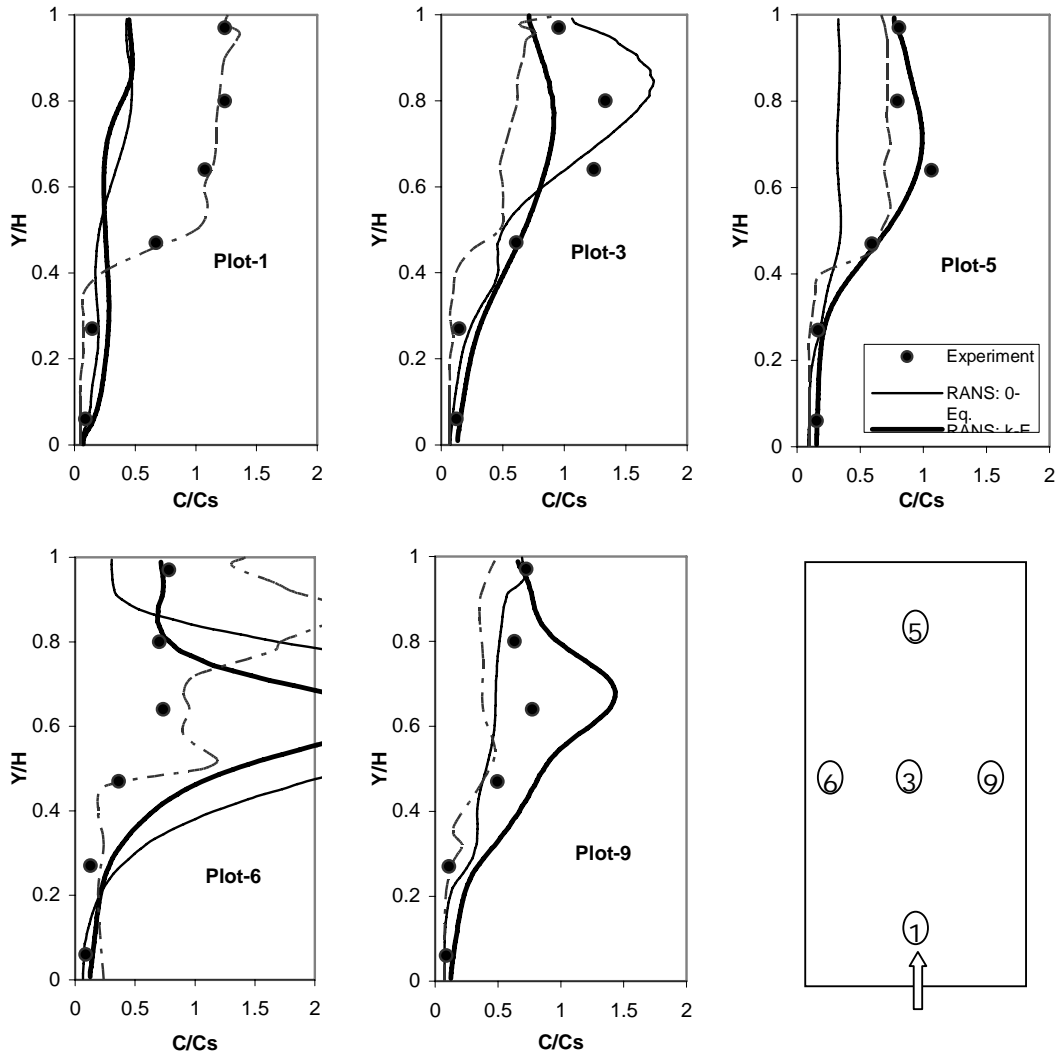
Fig. 12 The comparison of the tracer-gas concentration profiles at five positions in the room between the calculated and measured data for the displacement ventilation case. Z=height/total room height (H), H=2.43m, C_s = 0.42 ppm

CONCLUSIONS

This paper shows that applications of CFD program to indoor environment design and studies need some type of validation of the CFD results. The validation is not only for the CFD program but also for the user. The validation process will be incremental, since it is very difficult to obtain correct results for a complex flow problem in indoor environment.

This paper demonstrated the validation procedure by using displacement ventilation in a room as an example. The procedure suggests using two-dimensional cases for selecting a turbulence model and employing an appropriate diffuser model for simplifying complex flow components in the room, such as a diffuser. This paper also demonstrates the importance in performing grid-independent studies and other technical issues. With the exercises, one would be able to use a CFD program to simulate airflow distribution in a room with displacement ventilation, and the CFD results can be trusted.

ACKNOWLEDGEMENT

REFERENCES

Bartak, M., Beausoleil-Morrison, I., Clarke, J.A., Denev, J., Drkal, F., Lain, M., Macdonald, I.A., Melikov, A., Popiolek, Z. and Stankov, P. 2002. 'Integrating CFD and building simulation', *Building and Environment*, 37(8), 865-871.

Beausoleil-Morrison, I. 2002. 'The adaptive conflation of computational fluid dynamics with whole-building thermal simulation', *Energy and Buildings,* 34(9), 857-871.

Boussinesq, J. 1877. 'Théorie de l'écoulement tourbillant', *Mem. Présentés par Divers Savants Acad. Sci. Inst. Fr.,* 23, 46-50.

Carrilho da Graca, G., Chen, Q., Glicksman, L.R. and Norford, L.K. 2002. 'Simulation of wind-driven ventilative cooling systems for an apartment building in Beijing and Shanghai', *Energy and Buildings,* 34(1), 1-11.

Cheesewright, R., King, K.J. and Ziai, S. 'Experimental data for the validation of computer codes for the prediction of two-dimensional buoyant cavity flows', In: *Significant Questions in Buoyancy Affected Enclosure or Cavity Flows* (Ed. by J. A. C. Humphrey, C. T. Adedisian and B. W. le Tourneau), ASME, 75-81, 1986.

Chen, Q. and Moser A. 1991. 'Simulation of a multiple-nozzle diffuser', *Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within Buildings*. Ottawa, Canada, Vol. 2, pp. 1-13.

Chen, Q. and Xu, W. 1998. 'A zero-equation turbulence model for indoor airflow simulation', *Energy and Buildings*, 28(2), 137-144.

Chen, Q. and Srebric, J. 2002. 'A procedure for verification, validation, and reporting of indoor environment CFD analyses', *International Journal of HVAC&R Research*, 8(2), 201-216.

Deardorff, J.W. 1970. 'A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers', *Journal of Fluid Mechanics*, 41, 453-480.

Eckhardt, B. and Zori, L. 2002. 'Computer simulation helps keep down costs for NASA's "lifeboat" for the international space station', *Aircraft Engineering and Aerospace Technology: An International Journal*, 74(5), 442-446.

Emmerich, S.J. and McGrattan, K.B. 1998. 'Application of a large eddy simulation model to study room airflow', *ASHRAE Transactions*, 104, 1128-1140.

Fisk, W.J. 2000. 'Health and productivity gains from better indoor environments and their relationship with energy efficiency', *Annual Review of Energy and the Environment,* 25, 537-566.

Fisher, D.E. 1995. 'An experimental investigation of mixed convection heat transfer in rectangular enclosure', Ph.D. Thesis, University of Illinois at Urbana-Champaign, IL.

Huang, H. and Haghighat F. 2002. 'Modelling of volatile organic compounds emission from dry building materials', *Building and Environment*, 37(12), 1349-1360.

Jiang, Y. and Chen, Q. 2002. 'Effect of fluctuating wind direction on cross natural ventilation in building from large eddy simulation', *Building and Environment*, 37(4), 379-386.

Kato, S., Ito, K. and Murakami, S. 2003. 'Analysis of visitation frequency through particle tracking method based on LES and model experiment', *Indoor Air*, 13(2), 182-193.

Ladeinde, F. and Nearon, M. 1997. 'CFD applications in the HVAC&R industry', *ASHRAE Journal,* 39(1), 44.

Launder, B.E. and Spalding, D.B. 'The numerical computation of turbulent flows', In *Computer Methods in Applied Mechanics and Energy*, vol. 3, pp. 269-289, 1974.

Lesieur, M. and Metais, O. 1996. 'New trends in large eddy simulations of turbulence', *Annual Review of Fluid Mechanics*, 28, 45-82.

Lo, S.M., Yuen, K.K., Lu W.Z., and Chen, D.H. 2002. 'A CFD study of buoyancy effects on smoke spread in a refuge floor of a high-rise building', *Journal of Fire Sciences*, 20(6), 439-463.

Manz, H. 2003. 'Numerical simulation of heat transfer by natural convection in cavities of facade elements', *Energy and Buildings*, 35(3), 305-311.

Murakami, S., Kato, S., Ito, K. and Zhu, Q. 2003. 'Modeling and CFD prediction for diffusion and adsorption within room with various adsorption isotherms', *Indoor Air*, 13(6), 20-27.

Murakami, S., Iizuka, S. and Ooka, R. 1999. 'CFD analysis of turbulent flow past square cylinder using dynamic LES', *Journal of Fluids and Structures*, 13(78), 1097-1112.

Nielson, P.V. 1974. 'Flow in air conditioned rooms,' Ph.D. Thesis, Technical University of Denmark, Copenhagen.

Nielsen, P.V. 1998. 'The selection of turbulence models for prediction of room airflow', *ASHRAE Transactions*, 104(1).

Piomelli, U. 1999. 'Large eddy simulation: achievements and challenges', *Progress in Aerospace Sciences*, 35, 335-362.

Quinn, A.D., Wilson, M., Reynolds, A.M., Couling, S.B. and Hoxey, R.P. 2001. 'Modelling the dispersion of aerial pollutants from agricultural buildings - an evaluation of computational fluid dynamics (CFD)', *Computers and Electronics in Agriculture*, 30(1), 219-235.

Reynolds, O. 1895. 'On the dynamical theory of incompressible viscous fluids and the determination of the criterion', *Philosophical Transactions of the Royal Society of London, Series A*, 186, p. 123.

Sahm, P., Louka, P., Ketzel, M., Guilloteau, E. and Sini, J.-F. 2002. 'Intercomparison of numerical urban dispersion models – part I: street canyon and single building configurations', Water*, Air and Soil Pollution: Focus*, 2(5-6), 587-601.

Smagorinsky, J. 1963. 'General circulation experiments with the primitive equations. I. The basic experiment', *Monthly Weather Review*, 91, 99-164.

Srebric, J. 2000. 'Simplified methodology for indoor environment design,' Ph.D. Thesis, Department of Architecture, Massachusetts Institute of Technology, Cambridge, MA.

Swanddiwudhipong, S. and Khan, M.S. 2002. 'Dynamic response of wind-excited building using CFD', *Journal of Sound and Vibration*, 253(4), 735-754.

Thomas, T.G. and Williams, J.J.R. 1999. 'Generating a wind environment for large eddy simulation of bluff body flows - a critical review of the technique', *Journal of Wind Engineering and Industrial Aerodynamics*, 82(1), 189-208.

Topp, C., Nielsen, P.V. and Heiselberg, P. 2001. 'Influence of local airflow on the pollutant emission from indoor building surfaces', *Indoor Air*, 11(3), 162-170.

Tsou J.-Y. 2001. 'Strategy on applying computational fluid dynamic for building performance evaluation', *Automation in Construction*, 10(3), 327-335.

Woods, J.E. 1989. 'Cost avoidance and productivity in owning and operating buildings', *Occupational Medicine: State of the Art Reviews*, 4(4): 753-770.

Yeoh, G.H., Yuen, R.K.K., Lo, S.M. and Chen D.H.2003. 'On numerical comparison of enclosure fire in a multi-compartment building', Fire Safety Journal, 38(1), 85-94.

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

Zhai, Z. and Chen, Q. 2003. 'Solution characters of iterative coupling between energy simulation and CFD programs', *Energy and Buildings*, 35(5), 493-505.