5654

AN-92-9-1

SIGNIFICANT QUESTIONS IN PREDICTING ROOM AIR MOTION

Q. Chen, Ph.D.
Associate Member ASHRAE

Z. Jiang, Ph.D.

ABSTRACT

Based on a state-of-the-art review, some computational results, and experimental data, a few questions one usually encounters in numerical simulation of room air movement are discussed in this paper. The following conclusions can be drawn:

- *The standard k-ε model is still the most appropriate detailed model used in computing room airflows.*
- *The standard k-ε model may correctly predict turbulent air motion in a room if the thermal and flow boundary conditions are provided properly.*
- *It is difficult to predict unstable airflow and airflows with multiple solutions.*
- *The wall function method is not suitable for predicting heat exchange coefficient near a wall.*
- *Many complex diffusers could be numerically simulated by a number of approximated methods.*

INTRODUCTION

The quality of indoor air is increasingly being recognized as an essential factor for overall health and comfort because up to 90% of a typical person's time is spent indoors and a large fraction of that time is spent in a residential or commercial environment. During the past decade, indoor air pollution emerged as an international health issue. Increased awareness of the potential health risks associated with indoor air pollutants has stimulated interest in improving our knowledge about how ventilation air is distributed and how indoor contaminants are transported in buildings. Indoor air motion is also an important component for thermal comfort. Therefore, it is necessary to provide the means to investigate air distributions in rooms. The nature and severity of indoor air quality and thermal comfort problems could be assessed by airflow analysis, and effective numerical tools may help a designer choose the optimum design from a number of possible alternatives.

Investigation of indoor airflow pattern, air quality, and thermal comfort is mostly conducted by two approaches: experimental measurement and numerical simulation. In this paper, a number of significant questions concerning detailed numerical simulation of room airflow will be discussed.

WHAT IS THE MOST APPROPRIATE MODEL FOR PREDICTING ROOM AIR MOTION?

Turbulence can be characterized as a chaotic state of fluid motion. As yet, no complete theory on turbulence exists because its nonlinear dynamics are not well understood. Turbulence is characterized in terms of irregularity, diffusivity, large Reynolds numbers, three-dimensional vorticity fluctuations, dissipation, and continuum (Tennekes and Lumley 1972). Due to these features, it is difficult to identify whether a room airflow is a locally artificially induced turbulent airflow, transitional airflow, or fully developed turbulent airflow. However, very few room airflows are laminar. All nonlaminar room airflows could be defined as turbulent ones in the present study. Such turbulent flow prediction currently is done by three approaches: direct simulation, large-eddy simulation, and simulation by turbulence transport models.

Direct Simulation

Direct simulation is to compute a turbulent flow by solving the highly reliable Navier-Stokes equation without assumptions. According to turbulence theory (Nieuwstadt 1990), the number of grid points required to describe turbulent motions should be at least $N \geq \mathrm{Re}^{9/4}$. This number quickly becomes beyond the capacity of modern computers. Thus direct simulation is restricted to flows characterized by modest Reynolds numbers. Summaries of the state of the art of direct simulation are given by Nieuwstadt (1990) and Schumann (1991).

Large-Eddy Simulation

Based on the hypothesis that the turbulent motion could be separated into large eddies and small eddies so that the separation between the two does not have a significant effect on the evolution of large eddies, Deardorff (1970) developed a method called "large-eddy simulation" for meteorological applications. In the large-eddy simulation, although the Reynolds number is not

Qingyan Chen is an associate professor in the Laboratory for Aerodynamics and Hydrodynamics, Delft University of Technology, and a project leader in the TNO Institute of Applied Physics, Delft, the Netherlands. **Zheng Jiang** is a postdoctoral fellow in the Department of Mechanical Engineering, Concordia University, Montreal, Quebec, Canada.

THIS PREPRINT IS FOR DISCUSSION PURPOSES ONLY, FOR INCLUSION IN ASHRAE TRANSACTIONS 1992, V. 98, Pt. 1. Not to be reprinted in whole or in part without written permission of the American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc., 1791 Tullie Circle, NE, Atlanta, GA 30329. Opinions, findings, conclusions, or recommendations expressed in this paper are those of the author(s) and do not necessarily reflect the views of ASHRAE. Written questions and comments regarding this paper should be received at ASHRAE no later than Feb. 7, 1992.

used explicitly, it can be related implicitly to the separation of the two scales. The attempt is made to resolve the large-eddy motion by numerically solving a "filtered" set of equations governing the three-dimensional time-dependent motion. All scales of motion smaller than the grid size are filtered out so that only the large eddies (the macro structure) remain. In other words, the large eddies corresponding to the three-dimensional time-dependent equation are chosen so that they can be simulated on existing computers. Turbulent transport approximations are then made for small eddies, and the small-eddy motions can be modeled independently from the flow geometry. The philosophy behind this approach is that the macroscopic structure is characteristic for a turbulent flow. Moreover, the large scales of motion are primarily responsible for all transport processes, such as the exchange of momentum and heat.

The success of the method stems from the fact that the main contribution to turbulent transport comes from the large-eddy motion. Thus, the large-eddy simulation is clearly superior to turbulent transport closure wherein the transport terms (e.g., Reynolds stresses, turbulent heat fluxes, etc.) are treated with full empiricism. In addition, the ability to compute the large-scale structure of a turbulent flow is not only an important tool in view of applications but also gives insight into the fundamental nature of turbulence.

Although the large-eddy simulation can be involved in the solution of many turbulence problems, Nieuwstadt's (1990) review indicated that the large-eddy simulation still requires too much computational time to be useful for applications predicting room airflow motion. The expense arises from the need to simulate, three-dimensionally, the flow of interest with a mesh sufficiently fine and a time step sufficiently small. The fine mesh and small time step are necessary to capture all influential (large) spatial and temporal scales associated with random turbulence over a time interval covered by several thousands or even tens of thousands of time steps (Leschziner 1990). However, the large-eddy simulation can be quite valuable as an aid in developing transport closure models.

Turbulence Transport Models

Since the details of turbulent flow are difficult to calculate and engineers are mainly interested in the mean values, one turns to so-called turbulence transport models. Turbulence transport models are the basis of the engineer's approach where attempts are concentrated on looking for simplified models of turbulent flows. They are based on good physical insight and are applicable to the complicated flows encountered in reality. These models treat dynamic quantities as some sort of statistically averaged turbulent field and simulate only the gross features of the turbulent flows.

With a turbulence transport model, it is possible to predict the flows found in practice with the capacity of present computers. All the turbulence transport models are applied to the so-called "Reynolds equation," which is a statistically averaged Navier-Stokes equation. It can be written as

$$\frac{D\bar{V}_i}{Dt} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu_l\left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i}\right) - \overline{V'_iV'_j}\right] + \frac{\beta}{C_p}(\bar{H}_o - \bar{H})g_i \tag{1}$$

where $\bar{V}_i$ and V'_i are the mean and fluctuating velocities in i direction, respectively, ρ is density, and $\bar{p}$ is the mean pressure. The term $-\overline{V'_iV'_j}$ in Equation 1 can be considered as the effect of turbulence on the averaged flow. It is usually written in terms of Reynolds stress, τ, viz,

$$\tau = -\rho(\overline{V'_iV'_j}). \tag{2}$$

This term is a second-order tensor that makes the solution of the Navier-Stokes equation very complicated. Therefore, the starting point of all transport models is how to determine the Reynolds stress.

Turbulence transport models can be divided into two types (eddy-viscosity models and Reynolds stress models) according to whether or not the Boussinesq suggestion is used.

Eddy-Viscosity Models Boussinesq (1877) suggested that the Reynolds stress might be related to the rate of mean stream through an "eddy" viscosity. For the general Reynolds stress tensor in incompressible flow, the Boussinesq suggestion gives

$$\tau = -\rho\,\overline{V'_iV'_j} = \rho\nu_t\left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\rho k \tag{3}$$

where ν_t is the turbulent kinetic viscosity and is determined from turbulence transport models and δ_{ij} is Kronecker delta. The kinetic energy of turbulence k in the equation is defined as

$$k = \overline{V'_iV'_i}/2. \tag{4}$$

All the models using the Boussinesq suggestion are called eddy-viscosity models. The following presents several early and popular eddy-viscosity models.

Prandtl's (1926) mixing-length model was one of the first eddy-viscosity models that was proposed for two-dimensional boundary layers:

$$\nu_t = l^2\left|\frac{\partial V}{\partial y}\right| \tag{5}$$

where l is a "mixing length." It can be thought of as a transverse distance over which small eddies maintain their original momentum. The mixing length is somewhat on the order of a mean free path for the collision or mixing of globes of fluid. The product $l\left|\frac{\partial V}{\partial y}\right|$ can beinterpreted as the characteristic velocity of turbulence.

Kolmogorov (1942) proposed that turbulent-flow phenomena should be completed by solving two equations. The first of these is for the energy of the turbulent motion and the second is for its "frequency." The second equation normally is regarded as an auxiliary one. This model is a one-equation turbulence model.

In the one-equation models, a length parameter still needs to be prescribed. The length scale in the turbulence models should also depend upon the upstream "history" of the flow and not just local flow conditions. An obvious way to provide more complex dependence of l on the flow is to derive a transport equation for the variation of l. Thus, two-equation models were developed. An early two-equation model was that proposed by Harlow and Nakayama (1968). That model, after modification by Launder and Spalding (1974), has been widely used. It is often called the "standard" k-ϵ model, where ϵ is the dissipation rate of turbulent energy. Launder and Spalding (1974) have compared the k-ϵ model with the k-W ($W = \epsilon^2/(C_D k)^2$) model and the k-kl ($l = C_D k^{3/2}/\epsilon$) model, where C_D is a constant. They have found that when the constants of the turbulence models are used to calculate the turbulent Prandtl number, σ_t, the resulting values are -0.8 for the k-kl model, 2.9 for the k-W model, and 1.3 for the k-ϵ model. Only the latter value is of a magnitude that fits the experimental data of the various entities at locations far from walls. Numerous other two-equation models were suggested afterward, but it is very difficult to identify any other model superior to the standard k-ϵ model.

Reynolds Stress Models The eddy-viscosity models discussed above assume that the Boussinesq suggestion holds. If this assumption fails, then the models also fail. The other shortcoming of the models is the need to make assumptions in evaluating the various terms in the model transport equations, especially in evaluating the third-order turbulent correlations. In complex flow situations, as in recirculating flows, the mean turbulent energy models (which are based on the Boussinesq suggestion) are sometimes considered inadequate to represent the local state of turbulence. This deficiency can be overcome in Reynolds stress models, which explicity employ transport equations for the individual Reynolds stresses.

Before 1985, the applications of Reynolds stress models were mainly for thin shear flow, such as Hah and Lakshminarayana (1980), Gibson et al. (1981), and Hossain and Rodi (1982). Studies directed toward three-dimensional flows are still rare, as reviewed by Launder (1989). Direct applications in room airflow computation are those by Murakami et al. (1990) and Renz and Terhaag (1990). They computed airflow patterns in a room with jets. The results denote that the Reynolds stress model is superior to the standard k-ϵ model because anisotropic effects of turbulence are taken into account.

Based on a large number of applications for engineering flows, Leschziner (1990) concluded that Reynolds stress models have been found to be especially appropriate and beneficial when flow is dominated by a recirculation zone driven by a shear layer. In general, the applications with Reynolds stress models need more computing time than those with eddy-viscosity models because of greater algebraic complexity.

There are still defects and weaknesses in Reynolds stress models that needed to be solved (Launder 1989). The models deserve continuing study, improvement, and validation before they can be widely used for room airflow prediction.

The above review concludes that the standard k-ϵ model is still the most appropriate model for predicting room air motion. In addition to the detailed field models discussed above, there are also simplified models, such as the zonal model (Inard 1990) and a regional model (Zhang et al. 1990) that can be useful for practical applications.

CAN ROOM AIR MOTION BE PREDICTED BY THE k-ϵ MODEL?

The prediction of airflow in rooms started some 20 years ago. Encouraging results have been achieved by using the k-ϵ model for a number of problems concerning the airflow in a room, as reviewed by Whittle (1986), Nielsen (1989a), Rhodes (1989), and Kuehn (1990). The techniques have been applied to study the field distributions of air velocity, temperature, turbulence intensity, relative humidity, contaminant concentration, and air quality within a room.

In this section, some of the recent computational results, predicted by the programs with the k-ϵ model and validated by experimental data, will be presented briefly. In order to validate different numerical models, an international research team (IEA 1989) used the case proposed by Nielsen et al. (1978) for validation exercise. It is a two-dimensional isothermal airflow (forced convection) in a room shown in Figure 1a. The Reynolds number at the supply opening is 5,000. Detailed experimental data by an LDA system are available for validation (Nielsen et al. 1978). All the researchers reported having obtained good agreement with the experimental data (Chen 1991; Nady 1991; Yogl and Renz 1991). Figure 1 illustrates Chen's (1991) results where the computed velocity distribution and the comparison of mean velocity and turbulence intensity at section $x/L = 1/3$ are demonstrated in Figures 1b and 1c, respectively. In the simulation, the turbulence intensity in the y and z directions in the two-dimensional wall jet is given as $v'^2 = 0.6\ u'^2$ and $w'^2 = 0.8\ u'^2$. This means the intensity can be calculated from turbulent kinetic energy as

$$\sqrt{u'^2} = 0.9 k^{1/2}. \tag{6}$$

Allard et al. (1990) investigated the airflow in a room with natural convection. Awbi (1990) studied the diffusion of wall jets with and without wall obstructions. Borth (1990) conducted a study on the airflows in a room with a jet diffuser and a partition wall using three different computer programs. Choi et al. (1990) predicted the room airflow with an obstacle. Davidson and Fontaine (1989) calculated the flow in a ventilated room by a low-Reynolds-number k-ϵ model. Yamagishi et al. (1990) investigated the airflow in a high-ceilinged room with heating systems. All reported that the comparisons of predictions with corresponding experimental data show a reasonably good agreement.

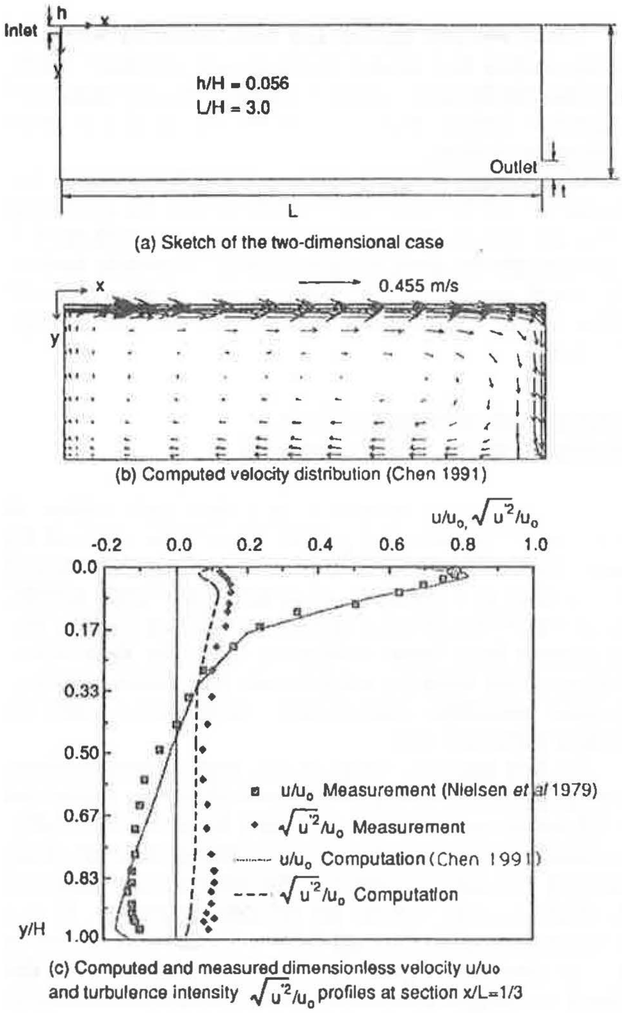


Figure 1 *Simulated and measured results of a two-dimensional case.*

From the above, it could be concluded that the k-ϵ model can be used to predict room air motion. However, there are many factors influencing the results predicted. Different results can be obtained by different users even with the same computer program. The accuracy of the results depends on the user's experience and skill in numerical simulation. On the other hand, the computer's capability is also a major concern. Baker and Kelso (1990) found that the accuracy of mathematically based discrete approximation lies strictly in the use of computational meshes with sufficient refinement to resolve local solution gradients. For a correct prediction, a large computer and a skillful researcher are essential.

It should also be noted that most of the above investigations were for fully turbulent flows or were based on the assumption that room airflows are fully turbulent. There is still argument about the characteristics of room air motion. As mentioned above, airflows in a room may be laminar unsteady, locally artificially induced turbulent, transitional, or fully developed turbulent flows. The success of applying the k-ϵ model to partially turbulent flows is doubtful. Unfortunately, few results on the subject are available. In addition, the concern with turbulence models clouds the issue of numerical stability in CFD simulation. Boundary conditions that give rise to unsteady flow are often "smoothed" over by the turbulence model. This is a very real physical phenomenon that often gets lost in turbulence modeling. It is an area that should receive more attention in the future.

CAN ROOM AIRFLOWS WITH MULTIPLE SOLUTIONS OR UNSTABLE AIRFLOWS BE PREDICTED?

Niu (1990) recently carried out an experiment (shown in Figure 2) in a room with a cold window and a hot radiator under the window. The airflow patterns observed are different in different measurements, although the boundary conditions in all the experiments were kept the same. Some flow patterns observed were closest to the numerical simulation shown in Figure 2a and others to that in Figure 2b. Of course, it is difficult to control the boundary conditions, such as window and radiator temperatures, to be constant. In most cases, the difference in observed airflow patterns would be explained as the results of a difference in boundary conditions. In fact, these observed airflow patterns can be repeated. Therefore, it is a flow with multiple solutions.

This type of airflow is difficult to predict. Niu (1990) tried to simulate the flow by a program with the standard k-ϵ model. However, it is not possible to obtain a stable solution, as shown in Figure 2. The computation first seems to converge to a solution after 350 iterations (Figure 2a) but it diverges afterward and leads to another possible solution at 1,100 iterations (Figure 2b) (the iteration is defined as the process for obtaining a converged result under a steady situation instead of the step used for a time-dependent computation). This process repeats during the computation, and it is not clear whether more iteration will result in a stable solution. Figure 3 shows the residuals of mass and energy continuity, which are acceptable at each possible solution.

The second example concerns the two-dimensional case used by the international research team (IEA 1989). The room length is three times as long as the height, and the inlet height is equal to 0.056 room height. The inlet conditions for the velocity are given by a Reynolds number of 5,000 with a 4% turbulence intensity. The Archimedes number is increased until a reduced penetration depth takes place. This is done by increasing the heat source on the floor and decreasing the supply air temperature while all the other surfaces were assumed to be adiabatic. More detailed information about the boundary conditions is given in Nielsen et al. (1978).

Different flow programs predict similar results. For example, Chen (1991) reported that the turning Archimedes number was 0.142, and Vogl and Renz (1991) showed a turning number of 0.15. However, Heikkinen (1991) reported that the turning Archimedes number depended on the iteration number, as shown in Figure 4. With an Archimedes number of 0.08, Figure 4a shows a counterclockwise circulation at the 500th iteration. The flow pattern changes during the computation and leads to a clockwise circulation at the 6,000th iteration. The

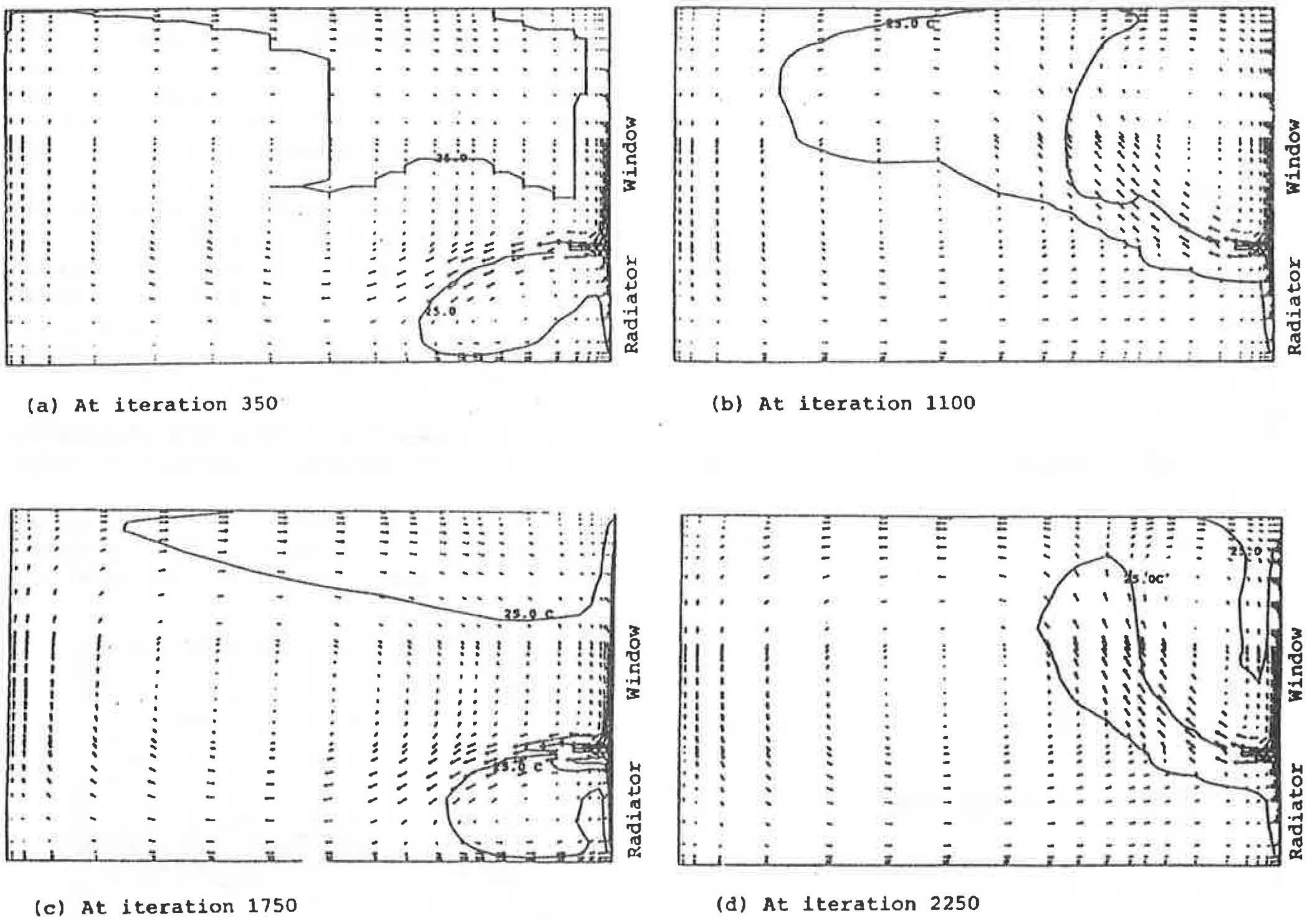
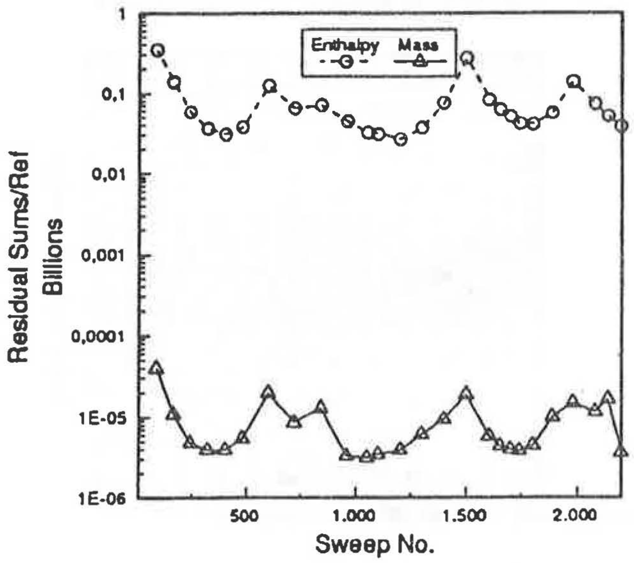


Figure 2 *Velocity and temperature distributions of the room with a cold window and a hot radiator surface with different numbers of iteration (Niu 1990).*



a). Residual Changing vs. Sweep No.

Figure 3 *Mass and energy residuals during the computation (Niu 1990).*

convergence residual at the 500th iteration is 1% mass inflow; at the 2,000th, 0.1%; at the 4,000th, 0.01%; and at the 6,000th, 0.005%. In such a situation, it is inadequate to do the computation with only 500 iterations, although the results seem converged. For a three-dimensional problem, it is more difficult to find the "true" solution. This is a case where the airflow is unstable.

The above two cases happen to be in a critical point where aiding force and counteraiding force are equivalent. A minor disturbance would have a significant impact on the airflow pattern. Therefore, they are difficult to compute. The experimental data obtained in those cases are not suitable for validation of computational results, and any validation exercise for airflows with multi-solutions or unstable airflows should be avoided.

Airflow with multiple solutions is also encountered in rooms with symmetrical boundary conditions. Asymmetry results are often found in both measurements and computations. The asymmetry phenomena are often interpolated as a result of the slight asymmetry in boundary conditions that may exist in an experiment. The international research team (IEA 1989) reported asymmetric results in experiments with symmetrical boundary conditions (Whittle and Clancy 1991).

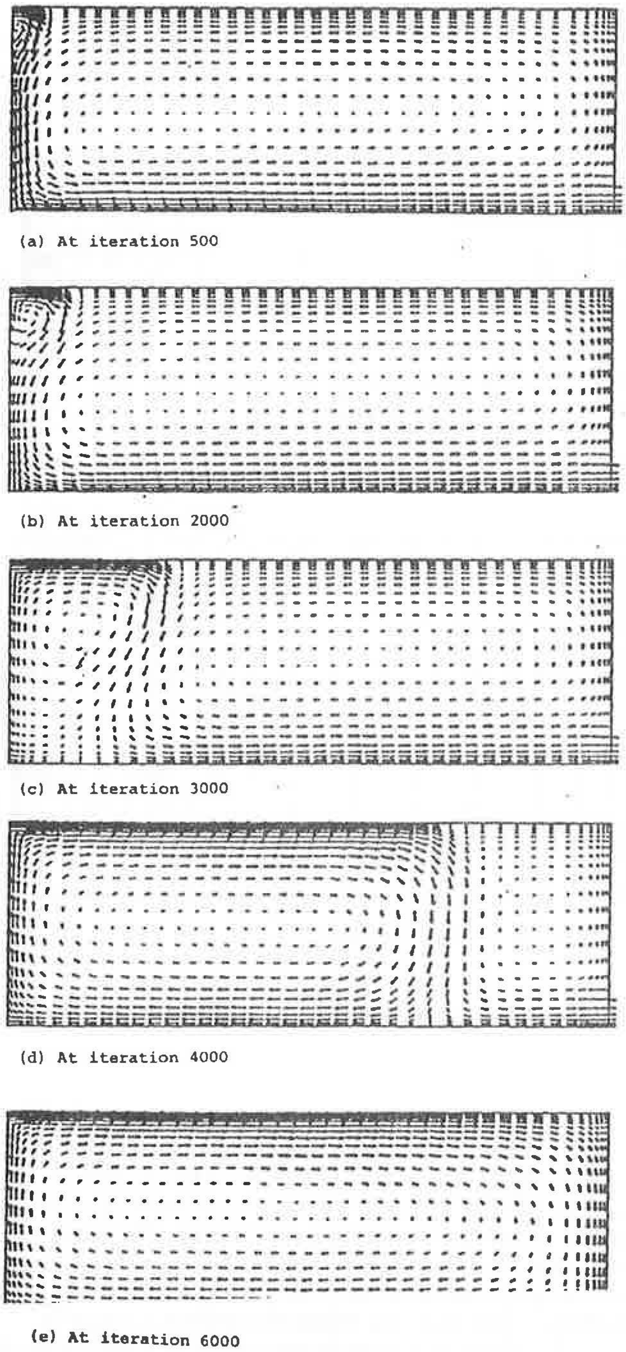


Figure 4 *Simulated velocity fields of the two-dimensional case with different iteration number (Heikkinen 1991).*

In numerical prediction, when a room is symmetrical in the mid-section, the airflow is computed only in half of the room for the sake of economy. The symmetry plane is often taken as a "zero-flux" boundary. However, if the computation is done for the whole room, asymmetric results may be obtained. Figure 5 illustrates the predicted air velocity distributions in a room with a supply opening on the rear wall near the ceiling. They are simulated by a flow program with the k-ϵ model. The boundary conditions for both cases are absolutely symmetrical. The only difference between the two cases is the effective area ratio of supply opening (effective area/gross area of the supply opening). Figure 5a shows a symmetrical velocity distribution but not Figure 5b. In numerical prediction, the asymmetric results are attributed mainly to the solving procedure, grid mesh distribution, and truncation error (numerical diffusion). Although the asymmetric results may be close to experimental data, it is difficult to identify the results as the true solutions.

The above examples demonstrate the difficulty in simulation of flows with multiple solutions or unstable airflows. It may be better to compute the flows as time-dependent ones. It would greatly increase the computing cost, but the results would be more reliable.

CAN HEAT TRANSFER AT WALLS BE CORRECTLY COMPUTED BY THE WALL FUNCTION METHOD?

The standard k-ϵ model is only suitable for high-Reynolds-number flow. In the near-wall region, where the local Reynolds number is very low, the model is not

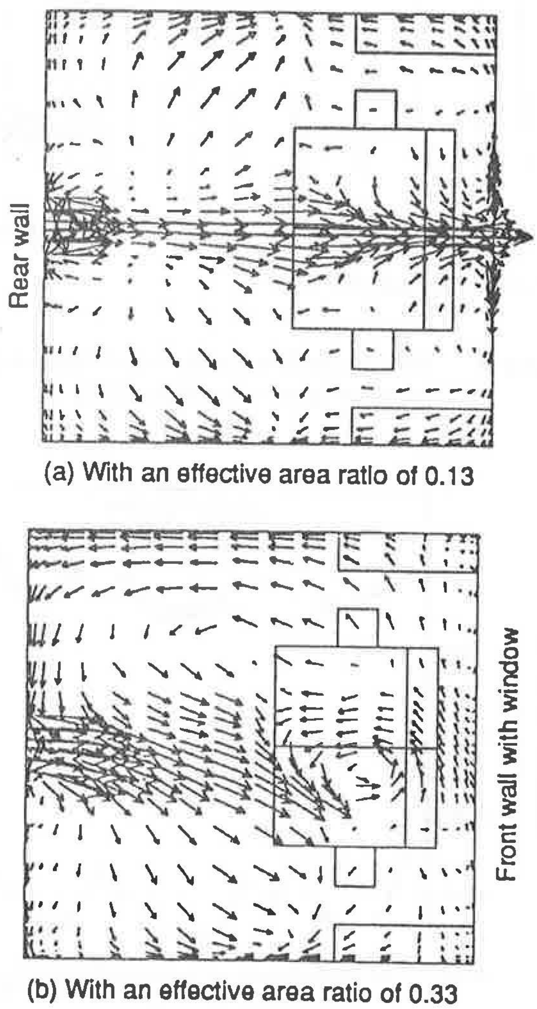


Figure 5 *Velocity distributions in an office with different effective area of supply opening (top view at the section via the supply opening near the ceiling).*

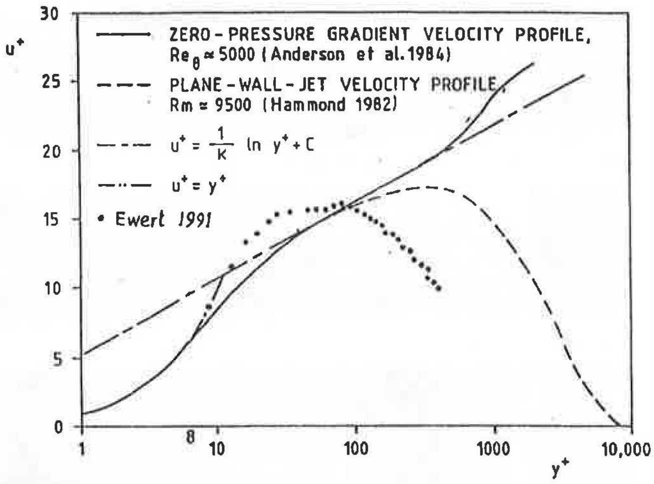


Figure 6 *Near-wall velocity profiles.*

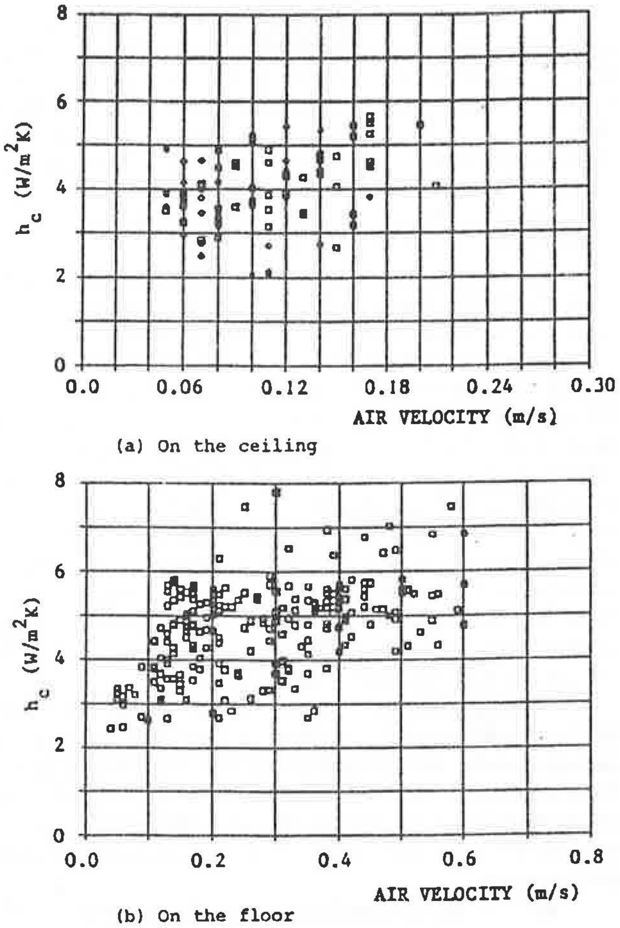


Figure 7 *Measured convective heat exchange coefficients for the ceiling and floor (Chen et al. 1989).*

valid. Hence, in the standard model, tne semi-empirical wall functions (Launder and Spalding 1974) are used to predict the heat transfer on a solid wall surface. A linear temperature and velocity distribution is often assumed for the inner region and a logarithmic one for the outer region. Figure 6 shows different measured near-wall velocity profiles and the velocity profile from the wall function. The one measured by Ewert and Zeller (1991) is from a room with a three-dimensional jet on the rear wall. It is seen that the wall function is not in agreement with the experimental data obtained by Ewert and Zeller (1991). For an office with a displacement ventilation system, Chen (1988) reported that, with the wall function, the computed heat exchange coefficient was between 1 and 3 W/m^2·K, while the measured data were 2 to 8 W/m^2·K (Chen et al. 1989), as shown in Figure 7.

One of the possible alternatives is to use low-Reynolds-number turbulence models that are also valid in the near-wall region. The models incorporate either a wall damping effect or a direct effect of molecular viscosity, or both, with the empirical constants and functions in the turbulence-transport equations devised originally for the high-Reynolds-number, fully turbulent flows remote from the walls. In the absence of reliable turbulence data in the immediate vicinity of a wall or at low Reynolds numbers, these modifications have been based largely upon comparisons between calculations and experiments in terms of global parameters. Patel et al. (1985) systematically evaluated the existing two-equation, low-Reynolds-number turbulence models. They found that most modifications to the high-Reynolds-number k-ϵ turbulence models lack a sound physical basis. The results of each of the models were compared for different flows, and it was not clear which of the many proposed models could be used with confidence. From an overall examination of the results, they concluded that the models of Launder and Sharma (1974), Chien (1982), and Lam and Bremhorst (1981), which are based on the k-ϵ model, and that of Wilcox and Rubesin (1980) yield comparable results and perform considerably better than the others. However, Patel et al. (1985) also suggest that these still need further refinement if they are to be used with confidence to calculate near-wall and low-Reynolds-number flows. Further, Betts and Dafa'Alla (1986) studied the buoyant, turbulent airflow in a tall rectangular cavity by the same low-Reynolds-number turbulence models used by Patel et al. (1985). Their results showed that only the models of Launder and Sharma (1974) and Hassid and Poreh (1978) were reasonably comparable with their experimental data, but none of them was very satisfactory. Chen et al. (1990) used the model of Lam and Bremhorst (1981) to compute airflow and heat transfer in a cavity with natural convection. It is found that the results are in better agreement with the experimental data. However, with the low-Reynolds-number models, an additional 20 to 30 grids are required for the near-wall region. This significantly increases the computing cost, which limits the practical applications of the models.

CAN COMPLEX DIFFUSERS BE SIMULATED?

The air diffusion in a room is dominated by diffuser type and the air supply parameters of the diffuser. It is difficult to compute the airflow around a diffuser because of the complex geometric configurations of diffusers used in practice. Without a correct description of the airflow around a diffuser, the simulations of air diffusion in rooms are not reliable. Hence, a suitable method for simulating diffusers is essential in predicting room air motion.

Several methods are applicable for simulating a complex diffuser. The box model used by Nielsen et al. (1978) is one of the earliest models. In principle, the model can be applied in the prediction of room air motion with any kind of diffuser. However, to use the box model, data must be obtained from either experiment or a more detailed computation.

Nielsen (1989b) imposed a formula for a jet diffuser based on the results from his experiment.* This formula has been applied to room airflow prediction with promising results (Lemaire 1990). Since the formula varies with the diffuser type, this method requires a large amount of time and effort.

Heikkinen (1990b) simulated a complex diffuser with 84 round nozzles, as shown in Figure 8a, by a so-called basic model. In the basic model, the diffuser is simulated simply by a rectangular slot that has the same effective flow area as the complex diffuser. The basic model was used to predict the airflow in a room with a complex diffuser under isothermal conditions. It gave a reasonable indication of the airflow pattern in the room, although there are discrepancies between the computations and measurements, but the method is not suitable for nonisothermal flow (Chen and Moser 1991).

The same diffuser shown in Figure 8a was simulated by Chen and Moser (1991) with two methods: the simple rectangular-slot method and the momentum method. In the former method, the 84 round nozzles were simulated by 84 rectangular slots (Figure 8b) with the same effective area. In the momentum method, the supply air momentum mV_{in} is set as

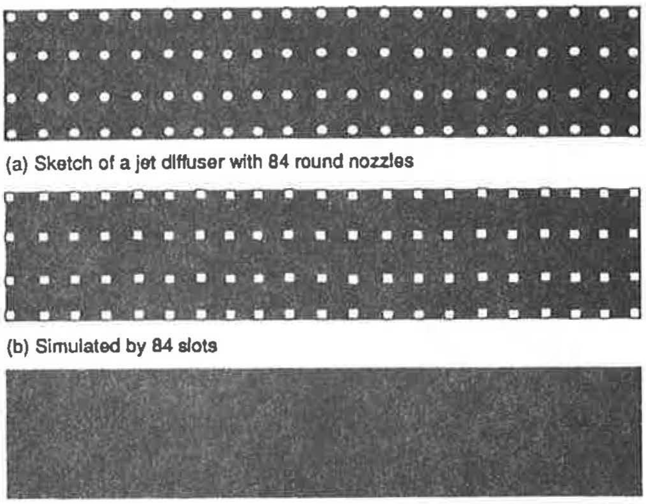
$$m\,V_{in} = m\,(\text{volume inflow rate}/\text{effective area}) \qquad (7)$$

where m is the mass inflow rate. In the numerical approach, the flow rate of the inlet is characterized with a fraction of the effective area over gross area of the diffuser. The fraction determines the portion of the grid cells of the inlet available for the supply air. By giving different kinds of supply momentum and its initial directions, different diffusers can be simulated. This method is equivalent to setting infinite nozzles/slots, as shown in Figure 8c. The computational results are compared with the experiments conducted in a full-scale room in Figure 9a. The computed results are in reasonable agreement with experimental data from Fossdal (1990) and Heikkinen (1990a), although there are discrepancies, especially in the region near the floor (Figures 9b and 9c). About five hours of CPU time in a supercomputer are required by the simple slot method because a large number of grid nodes are needed for presenting the 84 slots of the diffuser. Since the momentum method does not need many grids, the computing time can be reduced. Recently, Jiang et al. (1991) applied the momentum method to simulate the airflow around a complex vortex diffuser. The predicted results look reasonable.

CONCLUDING REMARKS

- It is impossible to use a direct simulation method to predict room air motion because of the limits of computer capacity. Large-eddy simulation is too expensive to be used for computing room airflows and at present cannot handle complex room geometry. Reynolds stress models give better results in room airflow simulation. However, the models need further refinement before they can be used widely. Among the eddy-viscosity models, the standard k-ϵ model is still the most appropriate one for computing turbulent airflows in rooms.
- The standard k-ϵ model has been applied to many room airflow problems, and encouraging results have been achieved. It may correctly predict turbulent air motion in a room when the thermal and flow boundary conditions are relatively simple.
- In many situations, room airflow has multiple solutions or is unstable. It is difficult to predict such an airflow, and prediction may vary markedly with the number of iterations used in the simulation.
- Most numerical predictions of heat transfer on a solid surface in the standard k-ϵ model use the semi-empirical wall function formulas. The wall function method is not suitable for room airflows. The low-Reynolds-number k-ϵ models could be an alternative, but too many grids are required in the near-wall region.
- A number of approximated methods have been imposed for simulation of complex diffusers. The momentum method is a simple one and may economically simulate a complex diffuser.

*His experiment is called the prescribed velocity method.



(a) Sketch of a jet diffuser with 84 round nozzles

(b) Simulated by 84 slots

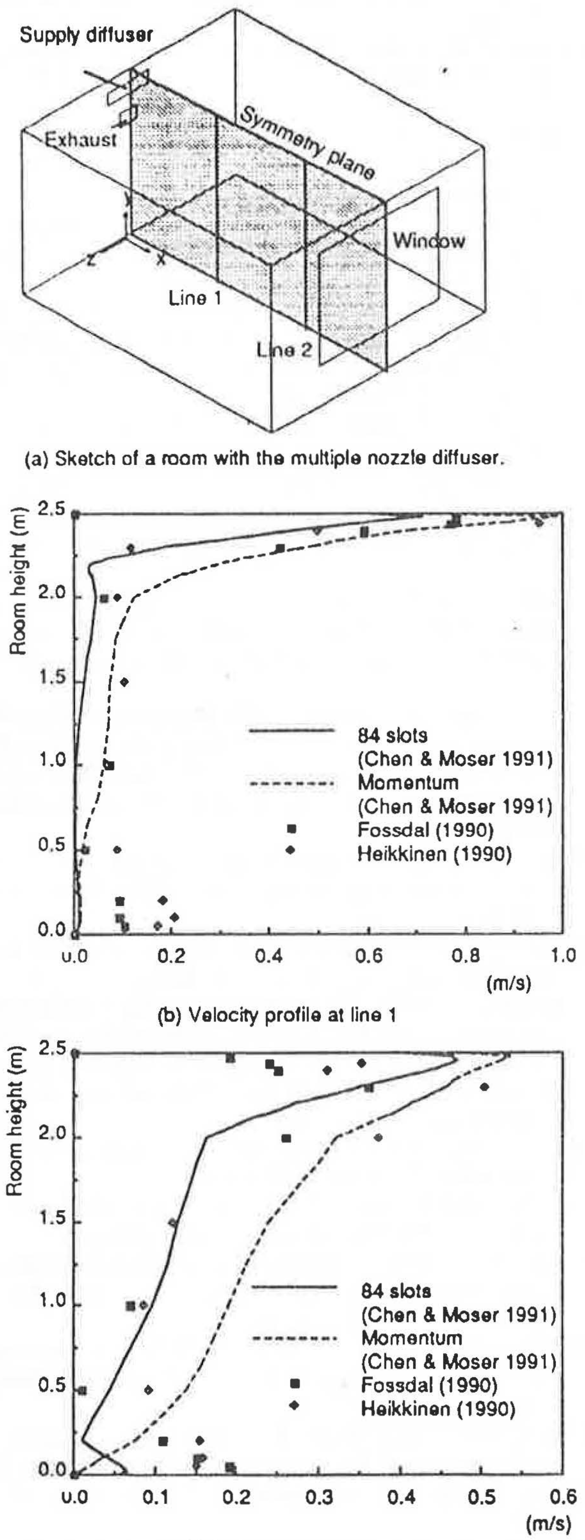
(c) Simulated by the momentum method (infinite slots)

Figure 8 Sketch of a jet diffuser and its simulation methods.

REFERENCES

Allard, F., C. Inard, and J.P. Simoneau. 1990. Experimental study and numerical simulation of natural convection in a room with heated ceiling or floor. *Proc. ROOMVENT '90*: 11.1-11.15. Oslo: NORSK VVS.

Anderson, D.A., J.C. Tannehill, and R.H. Pletcher. 1984. *Computational fluid mechanics and heat transfer*. Hemisphere Publishing Corp.



(a) Sketch of a room with the multiple nozzle diffuser.

(b) Velocity profile at line 1

(c) Velocity profile at line 2

Figure 9 *Comparison of velocity profiles between the computations and measurements in the mid-plane of the room.*

Awbi, H.B. 1990. The role of numerical solutions in room air distribution design. *Proc. ROOMVENT '90*: 2.1-2.17. Oslo: NORSK VVS.

Baker, A.J., and R.M. Kelso. 1990. On validation of computational fluid dynamics procedures for room air motion prediction. *ASHRAE Transactions* 96(1): 760-774.

Betts, P.L., and A.A. Dafa'Alla. 1986. Turbulent buoyant air flow in a tall rectangular cavity. *Significant Questions in Buoyancy Affected Enclosure or Cavity Flows*: 83-91. New York: American Society of Mechanical Engineers.

Borth, J. 1990. Numerical simulation of air flows in rooms. *European Research Community on Flow Turbulence and Combustion Bulletin* 5: 9-14.

Boussinesq, J. 1877. Essai sur la theorie des eaux courantes. *Mem. Presentes Acad. Sci.* 23: 46. Paris.

Chen, Q. 1988. Indoor airflow, air quality and energy consumption of buildings. Ph.D. thesis. Delft, the Netherlands: Delft University of Technology.

Chen, Q. 1991. Simulation of simple test cases. Inter national Energy Agency Annex 20, Report AN20.1-CH-91-ETHZ15.

Chen, Q., and A. Moser. 1991. Simulation of a multiple nozzle diffuser. Proc. 12th AIVC Conference. Coventry, UK: Air Infiltration and Ventilation Centre.

Chen, Q., C.A. Meyers, and J. van der Kooi. 1989. Convective heat transfer in rooms with mixed convection. *Proc. Int. Seminar on Air Flow Patterns in Ventilated Spaces*: 69-82. Liege: The National Fund of Scientific Research in Belgium.

Chen, Q., A. Moser, and A. Huber. 1990. Prediction of buoyant, turbulent flow by a low-Reynolds-number k-ϵ model. *ASHRAE Transactions* 96(1): 564-573.

Chien, K.Y. 1982. Predictions of channel and boundary-layer flows with a low-Reynolds number turbulence model. *AIAA J.* 20(1): 33-38.

Choi, H.L., L.D. Albright, and M.B. Timmons. 1990. An application of the k-ϵ turbulence model to predict how a rectangular obstacle in a slot-ventilated enclosure affects air flow. *Trans. in Agriculture* 33(1): 274-281.

Davidson, L., and J.R. Fontaine. 1989. Calculation of the flow in a ventilated room using different finite-difference schemes and different treatments of the walls. *Proc. CLIMA 2000* 3: 219-225. Sarajevo.

Deardorff, J.W. 1970. A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. *J. Fluid Mech.* 42: 453-480.

Ewert, M., and M. Zeller. 1991. Turbulence parameters at supply opening (measurements). Report R.I. 1.43, Int. Energy Agency Annex 20.

Fossdal, S. 1990. Measurement of testcase E (mixed convection, summer cooling) (preliminary). Report R.I. 1.17, Int. Energy Agency Annex 20.

Gibson, M.M., W.P. Jones, and B.A. Younis. 1981. Calculation of turbulent boundary layers on curved surfaces. *Phys. Fluids* 24: 386-395.

Hah, C., and B.J. Lakshminarayana. 1980. Prediction of two- and three-dimensional axisymmetrical turbulent wakes including curvature and rotation effects. *AIAA J.* 18: 1196-1204.

Hammond, G.P. 1982. Profile analysis of heat/mass transfer across the plane wall-jet. *Proc. 7th Int. Heat Transfer Conference* 3: 349-355. Munich.

Harlow, F.H., and P.I. Nakayama. 1968. Transport of turbulence energy decay rate. LA Report No. 3854. Los Alamos Science Lab, University of California.

Hassid, S., and M. Poreh. 1978. A turbulent energy dissipation model for flows with drag reduction. *ASME J. Fluids Eng.* 100: 107-112.

Heikkinen, J. 1990a. Measurements of case B2, B3, E2 and E3 (draft). Report R.I. 1.17, Int. Energy Agency Annex 20.

Heikkinen, J. 1990b. Modelling of the air inlet device. Report R.I. 1.24, Int. Energy Agency Annex 20.

Heikkinen, J. 1991. Simulation of simple test cases. Presented at 7th IEA Annex 20 Meeting, Aachen.

Hossain, M.S., and W. Rodi. 1982. A turbulence model for buoyant flow and its application to vertical buoyant jets. *Turbulent Shear Flows* I: 279-294. New York: Springer.

IEA. 1989. Air flow pattern within buildings. Flow Flash, No. 1. Int. Energy Agency Annex 20.

Inard, C. n.d. Simulation of testcase D with zonal model. Report R.I. 1.26, Int. Energy Agency Annex 20.

Jiang, Z., Q. Chen, and A. Moser. 1991. Comparison of displacement and mixing diffusers. Unpublished paper.

Kuehn, T.H. 1990. Heat and mass transfer in occupied buildings. *Proc. 9th Int. Heat Transfer Conf.* 1: 435-444. Jerusalem.

Lam, C.K.G., and K. Bremhorst. 1981. A modified form of the k-ϵ model for predicting wall turbulence. *ASME J. Fluids Eng.* 103: 456-460.

Launder, B.E. 1979. Stress-transport closures: Into the third generation. *Proc. 1st Symposium on Turbulent Shear Flows*. New York: Springer-Verlag.

Launder, B.E. 1989. Second-moment closure: Present . . . and future? *Int. J. Heat and Fluid Flow* 10: 282-300.

Launder, B.E., and B.I. Sharma. 1974. Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in *Heat and Mass Transfer* 1: 131-138.

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

Lemaire, A.D. 1990. Simulation of test case B. Presented at 6th IEA Annex 20 Expert Meeting, Nice.

Leschziner, M.A. 1990. Modelling engineering flows with Reynolds stress turbulence closure. *J. Wind Eng. and Industrial Aerodynamics* 35: 21-47.

Lipps, F.B. 1976. Numerical simulation of three dimensional Bénard convection in air. *J. Fluid Mech.* 75: 113-148.

Murakami, S., S. Kato, and Y. Kondo. 1990. Examining k-ϵ EVM by means of ASM for a 3-D horizontal buoyant jet in enclosed space. Int. Symp. on Eng. Turbulence Modelling and Measurements. Dubrovnik: ICHMT.

Nady, M.N. 1991. Simulation of a two-dimensional benchmark test case. Report No. AN20.1-CDN-90-IRC04, Int. Energy Agency Annex 20.

Nielsen, P.V. 1989a. Progress and trends in air infiltration and ventilation research. Proc. 10th AIVC Conf. Coventry: Air Infiltration and Ventilation Centre.

Nielsen, P.V. 1989b. Representation of boundary conditions at supply opening. Report R.I. 1.11, Int. Energy Agency Annex 20.

Nielsen, P.V., A. Restivo, and J.H. Whitelaw. 1978. The velocity characteristics of ventilated rooms. *J. Fluid Eng.* 100.

Nieuwstadt, F.T.M. 1990. Direct and large-eddy simulation of free convection. *Proc. 9th Int. Heat Transfer Conf.* 1: 37-47. Jerusalem.

Niu, J. 1990. Numerical simulation and experimental investigation of natural convection in an enclosure room. Report, Laboratory of Refrigeration, Delft University of Technology.

Patel, V.C., W. Rodi, and G. Scheuerer. 1985. Turbulence models for near-wall and low Reynolds number flows: A review. *AIAA J.* 23(9): 1308-1319.

Prandtl, L. 1926. Ueber die ausgebidete Turbulenz. *Proc. 2nd Int. Congress for Applied Mechanics*: 62-74. Zurich.

Prandtl, L. 1945. Uber ein neues Formelsystem fur die ausgebildete Turbulenz. *Nachr Akad. der Wissenschaft in Gottingen, van den Loeck und Ruprecht*: 6-19.

Renz, U., and U. Terhaag. 1990. Predictions of air flow pattern in a room ventilated by an air jet, the effect of turbulence model and wall function formulation. *Proc. ROOMVENT '90*: 18.1-18.15. Oslo: NORSK VVS.

Rhodes, N. 1989. Prediction of smoke movement: An overview of field models. *ASHRAE Transactions* 95(1).

Rodi, W. 1976. A new algebraic relation for calculating the Reynolds stress. *Z. Angew. Math.* 56: 219.

Schumann, U. 1991. Direct and large eddy simulation of turbulence—Summary of the state of the art 1991. Introduction to the Modelling of Turbulence, Lecture Series 1991-02. Rhode-Saint-Genese: von Karman Institute of Fluid Dynamics.

Tennekes, H., and J.L. Lumley. 1972. *A first course in turbulence*. London: MIT Press.

Vogl, N., and U. Renz. 1991. Simulation of simple test cases. Int. Energy Agency Annex 20 Report.

Whittle, G.E. 1986. Computation of air movement and convective heat transfer within buildings. *Int. J. Ambient Energy* 3: 151-164.

Whittle, G.E., and E.M. Clancy. 1991. Evaluation of cases B, D, E. Report D (R.I. 1.22). Int. Energy Agency Annex 20.

Wilcox, D.C., and W.M. Rubesin. 1980. Progress in turbulence modelling for complex flow fields including effects of compressibility. NASA Tech. Paper 1517.

Yamagishi, K., K. Kitagawa, K. Matsui, M. Itoh, and Y. Sakabe. 1990. Investigation of heating systems for a high-ceilinged room in a house. *ASHRAE Transactions* 96(2): 92-99.

Zhang, J.S., L.L. Christianson, and G.L. Riskowski. 1990. Regional airflow characteristics in a mechanically ventilated room under non-isothermal conditions. *ASHRAE Transactions* 96(1): 751-759.