CHAPTER 33

HVAC COMPUTATIONAL FLUID DYNAMICS

THE APPLICATION of computational fluid dynamics (CFD) to room air motion, as summarized by Haghighat et al. (1992), began with investigations into room airflow (Nielsen 1974) and natural convection in enclosed cavities (Catton 1978, Ostrach 1982, Markatos and Pericleous 1984, Lin and Nansteel 1987, Hadjisophocleous et al. 1988, Gadgil et al. 1984, Chen et al. 1990b). Lemaire (1987) applied the CHAMPHxN code (Pun and Spalding 1976) coupled with radiation to predict air movement and heat transfer in a room heated by a radiator.

Convective heat and mass tranfer has been analyzed using a commercial code by Holmes (1982), Markatos (1983), Jones and Sullivan (1985), and Chen and Van der Kooi (1988). Murakami et al. (1988), Horstman (1988), and Chen et al. (1990a) developed numerical models of ventilation with contaminant transport.

Partitioned models (multiple zones) have also been developed to predict room air motion. Natural convection was investigated by Chang et al. (1982) and Kelkar and Patankar (1985). Contaminant transport models were used in the ventilation models of Haghighat et al. (1989, 1990).

Theory

The basis for ventilation computational fluid dynamics analyses are the incompressible Navier-Stokes equations. These equations describe the motion of a viscous Newtonian fluid. For example, the following represent these equations in two dimensions:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \frac{1}{\rho}X - \frac{1}{\rho}\frac{\partial p}{\partial x} + v\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \quad (1)$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = \frac{1}{\rho}X - \frac{1}{\rho}\frac{\partial p}{\partial y} + v\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \quad (2)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (3)$$

Because these equations are complex, only a few exact solutions have been obtained for very simple flow conditions. The typical ventilation system is well beyond the realm of an exact solution. Forms of the Navier-Stokes equations have been developed using numerical methods that divide the flow field into finite volumes or elements. The elements typically assume uniform properties throughout and exchange pressure, momentum, and viscous dissipation information with one another.

The solution is obtained iteratively either by time step or by a steady flow updating of each element. The new value of pressure or velocity of an element at each iteration may be incorporated into the overall solution gradually using relaxation methods, which help stabilize the solution, giving time for information to travel between the elements and allowing each to assert its influence on the entire flow field. An example of this method is the finite difference approximation of the **stream function-vorticity formulation** of the Navier-Stokes equations in two dimensions:

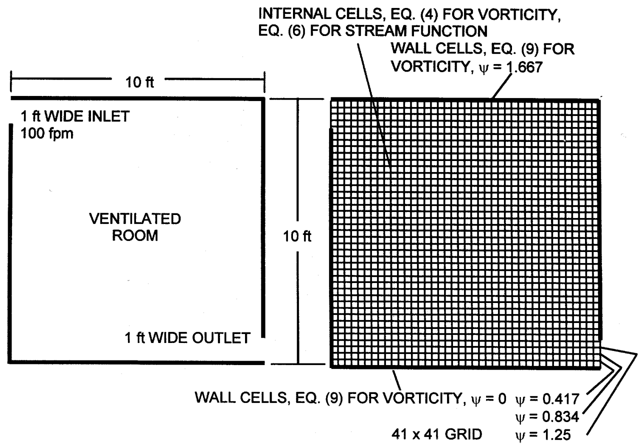

Fig. 1 Example Two-Dimensional Model

Example. A two-dimensional model of a ventilated room (Figure 1) is modeled using the stream function-vorticity formulation of the Navier-Stokes equations. The room is 10 ft by 10 ft. Air enters at 100 fpm through a 1 ft wide opening on the left wall near the ceiling and leaves through a 1 ft wide opening near the floor. The grid representing the room consists of 41 × 41 elements.

Solution:

The stream function-vorticity formulation of the Navier-Stokes equations begins with the **vorticity transport** equation:

$$u\frac{\partial \omega}{\partial x} + v\frac{\partial \omega}{\partial y} = v\left(\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2}\right) \quad (4)$$

The **stream function** is related to the velocity:

$$u = \frac{\partial \psi}{\partial y}, \qquad v = -\frac{\partial \psi}{\partial x} \quad (5)$$

Vorticity is analogous to rotation of the fluid:

$$\omega = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) = -\nabla^2\psi \quad (6)$$

Most CFD models have some sort of turbulence model to account for eddy viscosity when the Reynolds number exceeds about 2000. The simplest of the turbulence models uses the Prandtl mixing length:

$$v_t = \kappa L C_\mu^{1/4} K^{1/2} \quad (7)$$

The preparation of this chapter is assigned to TC 4.10, Indoor Environmental Modeling.

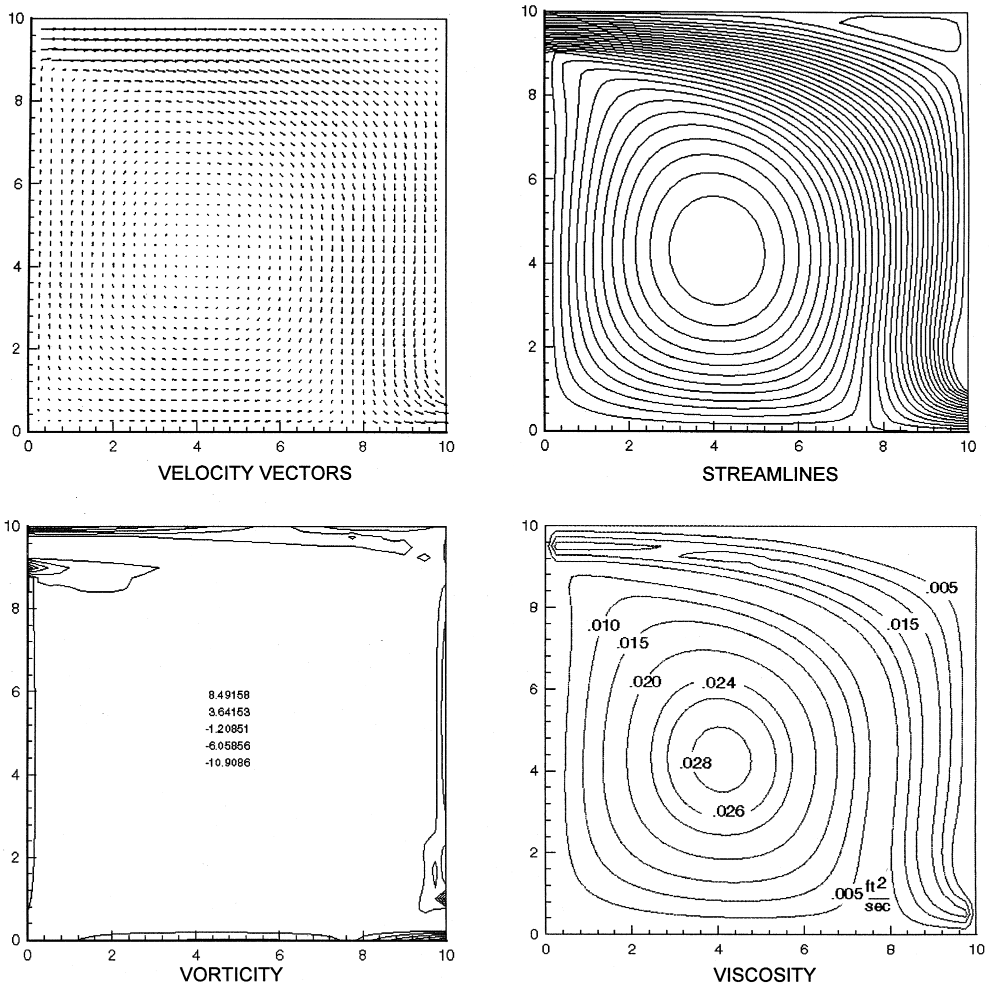

Fig. 2 Stream Function-Vorticity Solution for Ventilated Room (50,000 Iterations)

In recirculating flows, the length to the wall is difficult to measure and is flow-dependent. Equation (5) defines the length scale by the position of streamlines relative to the two boundary values of the stream function:

$$L = \left|\frac{\psi}{V}\right| \quad \text{or} \quad L = \frac{|\psi - \psi_B|}{|V|} \quad \text{whichever is smaller} \tag{8}$$

The turbulent kinetic energy K is a function of V', the magnitude of the fluctuating component of the velocity due to turbulence. V' can be assumed to be about 10% of V, the local average velocity, for typical room air models:

$$K = \frac{3}{2}(V')^2 = \frac{3}{2}(0.1V)^2 \tag{9}$$

Equations (7), (8), and (9) are combined to give a simple turbulence model based on the stream function alone:

$$v_t = \kappa|\psi| C_\mu^{1/4} \sqrt{\frac{3}{2}}(0.1)$$

$$\text{or} \quad v_t = \kappa|\psi - \psi_B| C_\mu^{1/4} \sqrt{\frac{3}{2}}(0.1) \quad \text{whichever is smaller} \tag{10}$$

Finite difference equations of the following form are used to approximate Equations (4) and (6):

$$\frac{\partial f}{\partial s} \approx \frac{f_{n+1} - f_{n-1}}{2\Delta S} \tag{11}$$

$$\frac{\partial^2 f}{\partial s^2} \approx \frac{f_{n+1} - 2f_n + f_{n-1}}{\Delta S^2} \tag{12}$$

The equations are rearranged so that the current value of the center cell or element is based on that of the surrounding cells. In addition, the vorticity transport Equation (4) is expanded for second-order accuracy as done by Dennis and Hudson (1978):

$$\omega_{i,j} = \frac{\begin{aligned}&\left(1 - \frac{\Delta x u_{i+1,j}}{2v_{i,j}} + \frac{\Delta x^2 u_{i+1,j}^2}{8v_{i,j}^2}\right)\omega_{i+1,j} \\ &+ \left(1 + \frac{\Delta x u_{i-1,j}}{2v_{i,j}} + \frac{\Delta x^2 u_{i-1,j}^2}{8v_{i,j}^2}\right)\omega_{i-1,j} + \dots \\ &\dots + \left(1 - \frac{\Delta x v_{i,j+1}}{2v_{i,j}} + \frac{\Delta x^2 v_{i,j+1}^2}{8v_{i,j}^2}\right)\omega_{i,j+1} \\ &+ \left(1 + \frac{\Delta x v_{i,j-1}}{2v_{i,j}} + \frac{\Delta x^2 v_{i,j-1}^2}{8v_{i,j}^2}\right)\omega_{i,j-1}\end{aligned}}{\left(4 + \frac{\Delta x^2 (u_{i,j}^2 + v_{i,j}^2)}{4v_{i,j}^2}\right)} \tag{13}$$

$$\psi_{i,j} = \frac{\psi_{i+1,j} + \psi_{i-1,j} + \psi_{i,j+1} + \psi_{i,j-1} + \Delta x^2 \omega_{i,j}}{4} \tag{14}$$

The subscripted velocities shown in Equation (13) are obtained from Equation (5) in the form of Equation (11). For example, the v velocity at location $i,j-1$ is

$$v_{i,j-1} = -\frac{(\psi_{i+1,j-1}) - (\psi_{i-1,j-1})}{2\Delta x}$$

or (15)

$$v_{i,j} = -\frac{(\psi_{i+1,j}) - (\psi_{i-1,j})}{2\Delta x}$$

Boundary conditions for vorticity are defined by the apparent rotation rate of the nearby fluid passing by:

$$\omega_{i,j}(\text{wall}) = -\frac{\psi_{adj} - \psi_{i,j(\text{wall})}}{\Delta x^2} \tag{16}$$

Relaxation parameters are applied to the vorticity and to the viscosity. The parameter δ for the vorticity is typically equal to 0.03, so that 3% of the new value and 97% of the old value is used. The viscosity is unstable, so a smaller relaxation parameter is required; 0.01 was used in this example. To illustrate this procedure, a new value of vorticity is calculated using Equation (13) or (16) (depending on the node type). The current value of vorticity is obtained by adding most of the old value to a fraction of the new value:

$$\omega_{i,j}(\text{current}) = \delta\omega_{i,j}(\text{new}) + (1-\delta)\omega_{i,j}(\text{old}) \tag{17}$$

The inlets and outlets have a fixed, uniform velocity. The stream function is set at each of the three nodes between the walls. Since the wall below the inlet has $\psi = 0$ and the wall above has $\psi = 1.667$, the stream function is divided evenly between the inlet nodes: $\psi_{1,38} = 0.25(1.667) = 0.417$; $\psi_{1,39} = 0.5(1.667) = 0.834$; $\psi_{1,40} = 0.75(1.667) = 1.25$. The same distribution is applied to the outlet nodes as shown in Figure 1. In a real room, the velocity would not be uniform across the outlet, but for an illustrative example, this simplification is appropriate.

The results of this analysis method are shown in Figure 2.

These results show the overall flow pattern of the room. Ventilation air travels along the ceiling, then attaches to the right wall before exiting. This generates a clockwise rotational flow where the eddy viscosity approaches two hundred times the molecular viscosity. At this point, the engineer would make a finer grid and run the model again to see if the solution has reached grid independence.

Finite Volume Formulations

Commercially available CFD programs are generally used to solve specific applications. Code development requires frequent validation and benchmarking and is best done by companies that specialize in that task. Most commercially available codes are based on finite volume formulations such as the SIMPLE algorithm (Patankar 1988). The finite volume method is distinguished from the finite element method in that the properties and flow conditions of the fluid are assumed uniform throughout the elemental volume and the exchange of momentum, energy, and pressure occurs at the faces.

Finite Element Formulations

Less common than the finite volume formulations, finite element formulations are usually available when the flow solver is packaged commercially with a stress analysis code. The finite element method may be differentiated from the finite volume method in the way the properties and flow conditions are point values and information travels between nodes, through the elements (Baker 1983). Until recently, the finite element method had the advantage of geometry adaptability.

Preprocessors

The first step in the CFD process is building the grid that mathematically represents the physical model. The preprocessor constructs the grid from such data as room dimensions, diffuser and duct dimensions, and internal features such as furniture or partitions that may affect flow. The preprocessor usually includes a computer aided design (CAD) package to allow the analyst to construct the required geometry. A good preprocessor usually includes a translator to other CAD packages so that geometry data is interchangeable.

The grid has an exact correspondence to the geometry; open areas, walls, diffusers, etc., are represented by the elements or volumes of computational fluid, boundaries, and blocked elements. The grid may be structured, where the element index or location is described by coordinates of $i(x)$, $j(y)$, and $k(z)$. Or, the grid may be unstructured, in which the elements are arbitrarily arranged and defined by associative identification.

Post Processors

After the flow solver has converged on the solution, the data is presented through the post processor. The typical post processor displays contour diagrams of equation variables such as pressure, temperature, velocity, turbulence, and contaminant concentration. Post processors are available that provide elaborate three-dimensional contour and vector plots with color scaling and animated output for time-dependent solutions.

Energy Equation

When the CFD application involves the exchange of heat, the energy equation is used. The energy equation is similar in structure to the Navier-Stokes equation:

$$\rho c\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{18}$$

The energy equation accommodates several HVAC situations. For example, heat in a room may be lost or gained through walls, windows, heaters, air conditioners, or occupants. Natural convection may dominate the flow pattern during heating or cooling and affect the comfort of the occupants, causing drafts or reduced air change effectiveness.

The energy equation is also important in compressible flow (high velocity). When the Mach number exceeds 0.2, the effects of compressibility begin to appear. The fluid density and velocity are determined in part by the energy equation. The CFD applications that require these high velocities are usually components (valves, ducting, ejectors, etc.).

A special class of problems requires the solution of the **conduction equation** (Laplace's equation). This type of analysis is called **conjugate heat transfer**. An example would be a room with an insulated wall of known thermal conductivity, where the solver determines both the temperature profile across the wall and the flow in the room.

Multispecies Flows

Often, a system with a mixture of gases must be evaluated. For example, a vent hood in a laboratory may be used to evacuate toxic gases from the room. A CFD model can provide an accurate assessment of the hood performance under a wide range of conditions and help to optimize the design for energy consumption.

If the concentration of the second gas is low, its viscous and momentum effects may be ignored. These types of problems may be solved using the single-species equation with the second species being convected along. The diffusion term is handled separately. If the concentration of the second gas is high enough to affect the properties of the mixture, such as density or viscosity, then the fully coupled form of the transport equations must be used.

Another class of multispecies flow problems involves the change of phase of one or more components. The humidity of a room with an evaporative water source would be an example of this.

Chemically reacting flows are often analyzed with CFD. Combustion models have been used extensively. These flows are complicated by the interaction of the various species, the phase changes, and the changes in properties and heat of formation of the combustion products.

Viscosity Models

In the classical Navier-Stokes equations, the viscosity is assumed constant. For low Reynolds numbers (i.e., Re < 2500), the laminar form is used. When higher Reynolds numbers are encountered, a means to adjust the viscosity (the **eddy viscosity**) is required. These types of equations are frequently called **turbulencemodels** because the basis for eddy viscosity is the turbulence level. Several turbulence models are available, but the most accepted and benchmarked model is the k-ε model. Chen (1995) provides some background for the k-ε application to room airflow. The standard k-ε model is a two-equation model. The first equation describes the turbulent kinetic energy transport and generation rate:

$$\rho u\frac{\partial k}{\partial x}+\rho v\frac{\partial k}{\partial y} = \frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial y}\right] + \mu_t\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2\right] - \rho\varepsilon - 2\mu\left[\left(\frac{\partial\sqrt{k}}{\partial x}\right)^2+\left(\frac{\partial\sqrt{k}}{\partial x}\right)^2\right] \quad (19)$$

Turbulent kinetic energy [Equation (9)] represents the fluctuating component of velocity.

The second equation in the k-ε model is the dissipation rate:

$$\rho u\frac{\partial \varepsilon}{\partial x}+\rho v\frac{\partial \varepsilon}{\partial y} = \frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial y}\right] + C_{\varepsilon 1}\mu_t\frac{\varepsilon}{k}\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2\right] - C_{\varepsilon 2}f_2\rho\frac{\varepsilon^2}{k}+E \quad (20)$$

where $$E \cong 2\frac{\mu_t}{\rho}\left[\left(\frac{\partial^2 u}{\partial x^2}\right)^2+\left(\frac{\partial^2 v}{\partial x^2}\right)^2\right]$$

Equation (20) is solved in conjunction with the Equation (19) to obtain the local eddy viscosity:

$$\mu_t = C_\mu f_\mu \rho\frac{k^2}{\varepsilon} \quad (21)$$

This two-equation model has a counterpart for high Reynolds number flows, but most room air motion studies use a lower Reynolds number model.

The near wall conditions for the turbulence model are often defined with wall functions that are based on the classical turbulent boundary layer. They may have two layers (the laminar sublayer and the turbulent layer) or they may have three layers (a laminar sublayer, a buffer layer, and a fully turbulent outer layer). The two-layer wall function is shown as follows:

$$\frac{u}{u_\tau} = y^+ \quad \text{for} \quad 0 < y^+ \le 30 \quad (22)$$

$$\frac{u}{u_\tau} = \frac{1}{\kappa}\ln(Ey^+) \quad \text{for} \quad 30 < y^+ \le 300 \quad (23)$$

where $$y^+ = \frac{\rho u_\tau y}{\mu}$$

and $$\mu\frac{\partial u}{\partial y}\bigg|_{\text{wall}} = \tau_w = \rho u_\tau^2 \quad (24)$$

Direct numerical simulation (DNS) is an active area of study of turbulence modeling. The basis for DNS turbulence modeling is that there is actually no turbulence model at all. Rather, the motion of the individual eddies within the bulk flow are modeled themselves. This model requires an extremely small grid size in which an eddy cannot exist due to molecular viscosity. Due to the huge demand on computing resources; only small flow fields, about shoe box size at the most, have been simulated using DNS.

Since DNS is really not yet practical for room air simulations, a compromise between it and conventional turbulence models has been developed. This method, **large eddy simulation** (LES), uses an eddy viscosity for the subgrid scale turbulence, but fully captures the time-dependent behavior of the larger energy bearing eddies. The application to room airflow is just beginning (Emmerich and McGrattan 1998).

CFD Applications

Currently, CFD has been applied to room air motion. The room might represent an auditorium, an aircraft cabin, or an automobile interior. Useful parameters such as velocity and temperature distribution, air change effectiveness, and humidity are predicted for the comfort of the occupant.

External flow models have used CFD to predict building infiltration, heat transfer rates for heating and cooling loads, and wind loads for structural design.

Finally, CFD is a useful tool for other internal flows, especially when nonstandard components are analyzed. Complex manifolds can be flow-balanced in one step, and important data such as pressure drop, aero/hydrodynamic loads, and heat transfer rates are available with the currently obtainable tools.

SYMBOLS

adj = subscript denoting adjacent node
c = specific heat, Btu/lb·°F or Btu/slug·°F based on units of ρ
$C_{\varepsilon 1}$ = constant = 1.45
$C_{\varepsilon 2}$ = constant = 1.92
C_μ = constant = 0.09
f = field variable such as velocity
f_2 = coefficient = $1 - 0.3\exp(-\text{Re}_t^2)$, where Re_t is local Reynolds number = $\rho K^2/\mu\varepsilon$
f_μ = coefficient = $\exp[-3.4/(1 + \text{Re}_t/50)^2]$
i = subscript index in x direction
j = subscript index in y direction
k = thermal conductivity, Btu/s·ft·°F
K = kinetic energy of turbulence, ft^2/s^2
L = turbulence length scale, ft
n = general subscript index
P = pressure, lb_f/ft^2
s = distance, ft
T = temperature, °R
u = fluid velocity in x direction, fps
u_τ = friction velocity, fps
v = fluid velocity in y direction, fps
V = local velocity magnitude, fps
V' = turbulent velocity fluctuation, fps
x = horizontal distance, ft
Δx = node spacing, ft
y = vertical distance, ft

y^+ = distance from wall in the universal velocity profile, ft
t = time, s
δ = relaxation parameter
ε = dissipation, ft^2/s^3
κ = von Karman constant = 0.41
μ = dynamic viscosity, slug/ft·s
μ_t = effective viscosity, slug/ft·s
ν = kinematic viscosity, ft^2/s
ν_t = effective viscosity, ft^2/s
ρ = density, $slug/ft^3$
σ_k = constant = 1.0
σ_ε = constant = 1.3
ψ = stream function, ft^2/s
ψ_B = boundary stream function, ft^2/s
ω = vorticity, 1/s

REFERENCES

Baker, A.J. 1983. *Finite element computational fluid mechanics*. Hemisphere Publishing, New York.

Catton, I. 1978. Natural convection in enclosures. *Heat Transfer*, 6.

Chang, L.C., J.R. Lloyd, and K.T. Yang. 1982. A finite difference study of natural convection in complex enclosures. *Proceedings of the 7th International Heat Transfer Conference* 2:183-88.

Chen, Q. and J. Van der Kooi. 1988. ACCURACY—A program for combined problems of energy analysis, indoor airflow, and air quality. *ASHRAE Transactions* 94(2):196-214.

Chen, Q. 1995. Comparison of different k-ε models for indoor air flow computations. *Numerical Heat Transfer*, Part B Fundamentals 28:353-69.

Chen, Q., A. Moser, and A. Huber. 1990a. Prediction of buoyant, turbulent flow by a low-Reynolds-number k-ε model. *ASHRAE Transactions* 96(1):564-73.

Chen, Q., A. Moser, and P. Suter. 1990b. Indoor air quality and thermal comfort under six kinds of air diffusion. *ASHRAE Transactions* 96(1).

Dennis, S.C.R. and J.D. Hudson. 1978. A difference method for solving the Navier-Stokes equations. *Numerical Methods in Laminar and Turbulent Flow*, pp. 69-80*.* Taylor Morgan Brebbia.

Emmerich, S.J. and K.B. McGrattan. 1998. Application of a large eddy simulation model to study room airflow. *ASHRAE Transactions* 104(1).

Gadgil, A., F. Bauman, E. Altmayer, and R.C. Kammerund. 1984. Verification of a natural simulation technique for natural convection. *ASME Journal of Solar Energy Engineering* 106:366-69.

Hadjisophocleous, G.V., A.C.M. Sousa, and J.E.S. Venart. 1988. Prediction of transient natural convection in enclosures of arbitrary geometry using a non orthogonal numerical model. *Numerical Heat Transfer* 13:373.

Haghighat, F., J.C.Y. Wang, and Z. Jiang. 1989. Natural convection and airflow pattern in a partitioned room with turbulent flow. *ASHRAE Transactions* 95(2):600-10.

Haghighat, F., J.C.Y. Wang, and Z. Jiang. 1990. Development of a three-dimensional numerical model to investigate the airflow and age distribution in a multizone enclosure. *Proceedings Fifth International Conference on Indoor Air Quality and Climate: Indoor Air '90* 4:183-88. International Conference on Indoor Air Quality and Climate, 2344 Haddinton Crescent, Ottawa, Ontario K1H 8J4, Canada.

Haghighat, F., J.C.Y. Wang, Z. Jiang, and F. Allard. 1992. Air movement in buildings using computational fluid dynamics. *Transactions of the ASME Journal of Solar Energy Engineering* 114:84-92.

Holmes, M.J. 1982. The application of fluid mechanics simulation program PHOENICS to a few typical HVAC problems. Ove Arup and Partners, London.

Horstman, R.H. 1988. Predicting velocity and contamination distribution in ventilated volumes using Navier-Stokes equations. *ASHRAE Conference, IAQ 88*, pp. 209-30.

Jones, P. and P.O. Sullivan. 1985. Modeling of air flow patterns in large single volume space. SERC Workshop: Development in Building Simulation Programs, Loughborough University.

Kelkar, K.M. and S.V. Patankar. 1985. Numerical prediction of natural convection in partitioned enclosures. *Numerical Heat Transfer*, pp. 63-71.

Lemaire, A.D. 1987. The numerical simulation of the air movement and heat transfer in a heated room resp. a ventilated atrium. *Proceedings of the International Conference on Air Distribution in Ventilated Space, ROOMVENT-87*. NORSKVVS, Oslo.

Lin, D.S. and N.W. Nansteel. 1987. Natural convection heat transfer in a square enclosure containing water near its density maximum. *International Journal of Heat and Mass Transfer* 30(11):2319-29.

Markatos, N.C. 1983. The computer analysis of building ventilation and heating problems. *Passive and Low Energy Architecture*, pp. 667-75.

Markatos, N.C. and K.A. Pericleous.1984. Laminar and turbulent natural convection in an enclosed cavity. *International Journal of Heat and Mass Transfer* 27(5):755-72.

Murakami, S., S. Kota, and Y. Suyama. 1988. Numerical and experimental study on turbulent diffusion fields in convection flow type clean rooms. *ASHRAE Transactions* 94(2):469-93.

Nielsen, P.V. 1974. Flow in air-conditioned rooms. *Ph.D. thesis*. Technical University of Denmark, Copenhagen.

Ostrach, S. 1982. Natural convection heat transfer in cavities and cells. *Proceedings of the International Heat Transfer Conference*. Hemisphere, Washington, D.C., pp. 365-79.

Patankar, S.V. 1988. Elliptic systems: Finite-difference method 1. *Handbook of numerical heat transfer*, Chap. 6, pp. 215-40. John Wiley and Sons, New York.

Pun, W.M., and D.B. Spalding. 1976. A general computer program for two-dimensional elliptic flows. HTS/76/2. Imperial College, London.

BIBLIOGRAPHY

Cebeci, T. 1988. Parabolic systems: Finite-difference method II. *Handbook of numerical heat transfer*. John Wiley and Sons, New York.

Chen, Q. and Z. Jiang. 1992. Significant questions in predicting room air motion. *ASHRAE Transactions* 98(1):929-39.

Lin, C.H., T. Han, and C.A. Koromilas. 1992. Effect of HVAC design parameters on passenger thermal comfort. SAE No. 920264. Society of Automotive Engineers, Warrendale, PA.