NISTIR 5997

Use of Computational Fluid Dynamics to Analyze Indoor Air Quality Issues

Steven J. Emmerich

Building and Fire Research Laboratory
Gaithersburg, MD 20899

United States Department of Commerce
Technology Administration
National Institute of Standards and Technology

NISTIR 5997

Use of Computational Fluid Dynamics to Analyze Indoor Air Quality Issues

Steven J. Emmerich

April 1997
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

Abstract

The potential for using a large eddy simulation (LES) computational fluid dynamics (CFD) model to analyze building indoor air quality (IAQ) and ventilation problems was investigated. The LES model was developed by the Fire Science Division of NIST to simulate the transport of smoke and hot gases during a fire in an enclosure. Based on an extensive literature review, the application of the LES model to a test case, and discussions with building industry contacts, it was determined that this model offers unique capabilities compared to other available CFD models and could be used to make a significant contribution in studying issues of current interest in the IAQ and ventilation field. Recommendations for future work include evaluation of the predictive accuracy of CFD, analysis of topics that take advantage of transient simulation capability of this model, and development of a strategy for U.S. industry to apply CFD in the design process.

Key Words: building technology, computational fluid dynamics, computer simulation, indoor air quality, large eddy simulation, room airflow modelling, turbulence, ventilation

BLANK PAGE

Acknowledgments

This work was funded by the National Institute of Standards and Technology. The author wishes to acknowledge the efforts of Kevin McGrattan and Andrew Persily in support of this project.

BLANK PAGE

Table of Contents

Introduction

There are two general types of computer simulation techniques for studying airflow and contaminant transport in buildings - multizone modelling and room airflow modelling. Multizone modelling takes a macroscopic view of indoor air quality (IAQ) by evaluating average pollutant concentrations in the different zones of a building as contaminants are transported through the building and its HVAC system. Room airflow modelling takes a microscopic view of IAQ by applying a computational fluid dynamics (CFD) program to examine the detailed flow fields and pollutant concentration distributions within a room or rooms. Each approach has strengths and limitations for studying different building ventilation and IAQ problems.

NIST has maintained a strong multizone modelling effort with the continuing development and application of the CONTAM program (Walton 1994). However, NIST has no active projects aimed at applying CFD to the study of building ventilation and IAQ. Several years ago, a CFD program called EXACT3 was developed for studying room airflows and was applied to the prediction of ventilation system performance in an open office space (Kurabuchi et al. 1990). EXACT3 employed a k-ε turbulence model which, at that time, was considered the only practical method for modelling turbulent airflows. Recently, a CFD program has been developed at NIST to simulate the transport of smoke and hot gases during a fire in an enclosure. This new CFD model employs highly efficient solution procedures and a technique, called large eddy simulation (LES), to eliminate the need for an empirical turbulence model. This program (referred to in this report as NIST-LES3D) may provide an opportunity to make a unique contribution to the analysis of IAQ and ventilation problems currently facing the building industry.

A project was undertaken at NIST to investigate options for using NIST-LES3D for studying building ventilation and indoor air quality. The first task of this effort was to work with the group developing NIST-LES3D in the Fire Science Division to learn the capabilities of the model. This task focused on learning unique features of the model that are either advantageous or disadvantageous to the study of building ventilation and IAQ. It also involved identifying additional features needed to study problems of interest. An additional aspect of this task was to perform some simulations for a test case. The second major task was to learn the current state of research in the area of CFD modelling of building ventilation and IAQ through a literature review, professional contacts, and technical conferences. An important aspect of this task was learning potential applications of interest in the building and IAQ industries.

This report is divided into five main sections. The first section contains a brief discussion on CFD theory and a summary of CFD terminology used throughout the report. The second section provides the context of the current work through a summary of CFD work done in the NIST Building Ventilation and IAQ Group in the past and a description of the NIST-LES3D program. The third section summarizes the literature review performed to learn the current state of CFD research in the building ventilation and IAQ fields. The fifth section discusses a number of CFD modelling issues based on the literature review. The final section summarizes the investigation and recommends options for using NIST-LES3D for studying building ventilation and indoor air quality.

CFD theory and terminology

This section contains a brief discussion on computational fluid dynamics (CFD) theory and summarizes some of the terminology used throughout the rest of the document.

Theory

This subsection provides a general introduction to CFD theory. A thorough treatment may be found in many textbooks on the subject such as Anderson et al. (1984) or in an earlier NIST report (Kurabuchi et al. 1990). CFD is the application of numerical techniques to solve the Navier-Stokes equations for fluid flow. The Navier-Stokes equations are derived by applying the principles of conservation of mass and momentum to a control volume of fluid. When applying CFD to the IAQ field, conservation of mass for a contaminant species and energy for thermal responses may also be applied. The Navier-Stokes equations are presented in many different forms depending on the assumptions included and the mathematical notation used. The equations below are a common presentation and incorporate the Boussinesq approximation (from Kurabuchi et al. 1990).

Conservation of momentum

$$\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{ \nu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right\} - \beta g_i \theta \tag{1}$$

Conservation of mass

$$\frac{\partial U_i}{\partial x_j} = 0 \tag{2}$$

Conservation of energy

$$\frac{\partial \theta}{\partial t} + \frac{\partial \theta U_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\kappa \frac{\partial \theta}{\partial x_j} \right) + H \tag{3}$$

Conservation of contaminant species

$$\frac{\partial C}{\partial t} + \frac{\partial C U_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(D \frac{\partial C}{\partial x_j} \right) + S \tag{4}$$

where C = instantaneous concentration of contaminant
D = molecular diffusion coefficient for the contaminant
g_i = gravitational acceleration in x_i direction
H = volumetric heat source generation rate
P = instantaneous static pressure difference
S = volumetric contaminant generation rate
t = time
U_i = instantaneous velocity component in x_i direction
x_i = Cartesian coordinates
β = volumetric coefficient of expansion
κ = thermal diffusivity
θ = instantaneous temperature difference

ρ = density of air
ν = kinematic viscosity of air.

In CFD, the Navier-Stokes equations are solved by discretizing the equations using either finite difference or finite element techniques. Direct numerical simulation (DNS) involves solving these equations directly with the problem discretized to a grid fine enough to capture the smallest possible turbulent eddy. The eddy size may be estimated from dimensional analysis to be of the order of the Kolmogorov length scale (ν^3/ε where ε is the dissipation rate of turbulence energy) and is typically 10^{-2} m to 10^{-3} m (Tennekes and Lumley 1972).

Performing a DNS for the airflow in a room is a formidable task even for modern computing power. Therefore, various other approaches have been used to model turbulence. The most widely applied turbulence modelling technique is the k-ε model (where k is the turbulent kinetic energy and ε is the dissipation rate of turbulent energy). This model is used in all available commercial CFD programs and in most research codes. As with the general theory of CFD, the details of the k-ε model are presented elsewhere (e.g. Kurabuchi 1990) and only a general description will be included here.

The k-ε model is derived by substituting the sum of an average term plus a fluctuating term for the instantaneous quantities in equations 1-4 (e.g., $U_i=u_i+u'_i$). The average terms are expected to vary less than the instantaneous quantities and can therefore be resolved over a coarser grid. However, this averaging procedure yields an additional unknown term called the Reynolds stress ($-\nu u'_i u'_i$). The additional unknowns are resolved by introducing the eddy viscosity concept which results in two additional conservation equations, one each for k and ε, and five empirical constants.

Although the k-ε model is an empirical formulation, it has been used successfully to model turbulent flow in many different situations including room airflow. However, there have also been problems applying the k-ε model to room airflow simulation as highlighted by Moser (1991) in a summary of International Energy Agency (IEA) Annex 20 "Airflow Patterns within Buildings". Moser concluded that the accurate simulation of room airflow with forced convection with a k-ε turbulence model was possible but problems included turbulence modelling at low Reynolds numbers and in the near-wall region and modelling natural and mixed convection. More detail on Moser's report is included in the literature review section of this report. In response to these problems, researchers have extended the k-ε model specifically for situations of low Reynolds numbers, near-wall regions, and natural convection. While they have had some success, the result is a more empirical and less general model.

An alternative to DNS and the k-ε turbulence model is large eddy simulation (Smagorinsky et al. 1965 and Deardorff 1970). The LES method involves solution of the time-dependent Navier-Stokes equations spatially filtered over the computational grid. The effect of subgrid scale (SGS) motion is approximated though a SGS eddy viscosity model where the viscosity of equation 1 is replaced with a sum of the viscosity and a SGS viscosity ($\nu=\nu+\nu_s$). The commonly used version of the LES model is presented in equation 5 (from Kurabuchi and Kusuda 1987).

$$\nu_s = (C_s\Delta)^2\sqrt{\frac{(\frac{\partial U_i}{\partial x_i}+\frac{\partial U_i}{\partial x_j})^2}{2}} \qquad (5)$$

where C_s = empirical constant
Δ = mean local grid size.

Kurabuchi and Kusuda cite the advantages of the LES model as being direct use of the filtered Navier-Stokes equations, only one empirical constant, and more universal and acceptable modelling for the SGS stresses than those required in other turbulence models. The primary disadvantage they mention is the greater computational difficulty which they see limiting the method to problems requiring very fundamental understanding of flow phenomena or verification of simpler models. An important fundamental difference between the LES and k-ε methods is the treatment of the time-dependency of the problem. The k-ε method results in a 'steady state' solution to an averaged version of the flow equations while the LES results in a transient solution to the actual Navier-Stokes equations. Any real turbulent flow situation is inherently transient. Therefore, LES methods have an advantage in modelling turbulent flow.

Terminology

Following are definitions of terms and acronyms used throughout this report.

air diffusion performance index (ADPI) - percentage of locations in an occupied zone that meet air movement and temperature specifications for comfort (Chapter 31 of ASHRAE 1993)
algebraic stress model (ASM) - empirical turbulence model that adds consideration of anisotropy of Reynolds stresses to the k-ε model through additional algebraic equations
computational fluid dynamics (CFD) - numerical solution of the Navier-Stokes equations
conjugate heat transfer (CHT) - processes that involve variations of temperature within solid walls and air due to thermal interactions between the walls and air
direct numerical simulation (DNS)- direct solution of the Navier-Stokes equations with grid fine enough to capture smallest possible turbulent eddy
differential stress model (DSM) - same as Reynolds stress model (see below)
eddy viscosity model (EVM) or k-ε model – common empirical turbulence model used with a time-averaged form of the Navier-Stokes equations
large eddy simulation (LES) - solution of time-dependent Navier-Stokes equations spatially filtered over the computational grid
predicted mean vote (PMV) - predicted mean vote one would expect to get by averaging the thermal sensation vote of a large group of people in a space
predicted percent dissatisfied (PPD) - predicted percentage of dissatisfied persons in a space due to draft and indoor air pollution
renormalization group (RNG) - a k-ε turbulence model in which small scale turbulence effects are represented by means of a random forcing function in the Navier-Stokes equations
Reynolds stress model (RSM) - also called second-order closure model, complex turbulence model used with time-averaged Navier-Stokes equations with differential equations to consider anisotropy of Reynolds stresses

Background

This section summarizes CFD work done by the NIST Ventilation and Indoor Air Quality (IAQ) Group in the past and describes the Large Eddy Simulation computational fluid dynamics (NIST-LES3D) code developed at NIST.

NIST Ventilation and IAQ Group

This section summarizes past applications of CFD to the study of building ventilation and IAQ at NIST. As mentioned earlier, Kurabuchi et al. (1990) describes the development of a public domain CFD code called EXACT3. The program employs finite difference approximations and can handle three-dimensional, nonisothermal, turbulent flows using the k-ε turbulence model. EXACT3 has a companion program called CONTAM3 which can be used to calculate the distribution of contaminants after a flow solution is obtained. Kurabuchi describes the history of CFD application to room airflow in Japan, basic theory for turbulent flows, the numerical method for solving the system of equations, comparison of predicted velocities and temperatures to experimental results for some simple geometries, and application to a practical problem. The limitations of the program discussed include: calculation domain must be rectangular, Reynolds number must be large enough so that viscous effects are negligible, and temperature gradients must be small enough so that buoyancy exerts little influence on the turbulence structure as it may exert a directional influence on turbulence for which the k-ε model cannot account.

EXACT3 has been used by the IAQ group at NIST to study several ventilation problems. For example, Fang and Grot (1990) applied EXACT3 to study ventilation system performance in a room. They compared temperatures, flow patterns, and Air Diffusion Performance Index (ADPI) to experimental results. Agreement for ADPI was good for supply flow rates up to 55 $m^3/h \cdot m^2$ (3 cfm/ft^2) but poor at higher flow rates.

Fang and Persily (1991) studied ventilation system performance in an open office space using EXACT3. They varied parameters including supply airflow rates, supply airflow discharge direction, and inclusion of partitions. Performance measures examined include ADPI, age of air and air change effectiveness (simulated via CONTAM3 with SF_6 as a tracer gas). The low supply airflow rates did not degrade either thermal comfort or ventilation effectiveness. The supply airflow discharge direction had no significant effect until it was over 45 degrees from the ceiling at which point it degraded thermal comfort. Locating a partition between the supply and return points disrupted the airflow pattern and degraded both thermal comfort and ventilation effectiveness. Providing clearance below the partition partially restored the airflow pattern. Items identified as needing additional work were specifying boundary conditions at the diffuser and verifying the predicted results with experimental data.

Fang and Persily (unpublished) used EXACT3 to predict air movement, thermal comfort and ventilation air distribution within an interior cubicle in an open office space. An objective was to study the applicability of CFD to the evaluation of thermal comfort and ventilation effectiveness in mechanically ventilated office space. Performance measures examined include ADPI, age of air and air change effectiveness (simulated via CONTAM3 with SF_6 as a tracer gas) for three supply airflow rates. Good thermal comfort and uniform air mixing were found.

EXACT3 has also been used to study building airflow by researchers outside of NIST. For example, Zhang et al. (1992) compared computations using EXACT3 and measurements of air velocity, turbulent kinetic energy and temperature in a full-scale ofice room under non-isothermal conditions. The numerical predictions achieved a 'reasonable agreement' with measurements for airflow pattern and distribution patterns of velocity, temperature and turbulent kinetic energy. However, the quantitative comparison was poor due to numerical predictions of slower jet decay, narrower jet spread, and delayed drop of jet compared to the measurements. Other explanations for differences included the lack of a radiation model and unavoidable approximation in the specification of boundary conditions. In another study, Lam et al. (1993) describe new pre- and post-processing software developed for EXACT3 and its application to an atrium with balconies. They found good qualitative agreement between simulations and scale model experiments. Also, Cooper (1993) describes use of EXACT3 to predict the pressure distribution around a sheltered building. Comparison with previous wind tunnel tests indicated that the predicted pressure coefficient values differed, but fairly good agreement was found with the trends in the tests. Problems were found with trying to restrict the problem to two dimensions.

NIST-LES3D Program Description

Recently, Howard Baum, Kevin McGrattan and others at NIST have developed a CFD code using the LES method to simulate the transport of smoke and hot gases during a fire in an enclosure (McGrattan et al. 1994). Their approach to CFD emphasizes the use of high spatial resolution and efficient flow solving techniques while avoiding empirical models. The NIST-LES3D model employs a weakly compressible form of the Navier-Stokes equations. The key to the approach is limiting the problem to regular rectangular, cylindrical or spherical geometry. The grid is required to be uniform in the horizontal direction but may be varied in the vertical direction. This allows the derivation of the flow equations in a manner which can be solved by a Poisson solver using Fast Fourier Transform (FFT) techniques instead of an iterative solution. The fast solution enables use of high grid resolution fine enough to capture all 'important eddies'. It is not possible to capture all eddies, but extremely small ones have little influence on bulk properties of most problems (temperature and contaminant distribution). The SGS model is simply a constant eddy viscosity (i.e., ν_s in equation 5 is constant). For fire-driven flows, this simple technique has led to good agreement with experimental data. For building ventilation and IAQ problems, the adequacy of a constant eddy viscosity will have to be reexamined. The current version of the derivation of the flow equations and a description of the numerical solution methods employed is presented by Baum et al. (1996).

Literature review

Although computational fluid dynamics has been applied to other fields for decades, its application to building ventilation and IAQ research is more recent. However, interest in applying CFD to ventilation and IAQ topics has grown tremendously in the last 10 years. This section summarizes a literature review performed to assess the current state of CFD research in the building ventilation and IAQ field including a description of many specific applications, a discussion of modelling and other issues, and a summary of studies employing LES.

General room airflow

This subsection describes a few major efforts that applied CFD to study room airflow in some detail and briefly summarizes reports covering a wide range of topics from cold-air distribution systems to the impact of ceiling-mounted obstacles on airflow from a supply jet. Other room airflow topics that have been studied by a significant number of researchers are covered in separate subsections.

One of the most important recent efforts was International Energy Agency (IEA) Annex 20 "Air Flow Patterns within Buildings" (Moser 1991). The objective of the Annex was to evaluate the performance of both CFD and multi-zone airflow simulation techniques and to establish their viability as design tools. Research under the Annex included simulation and measurement in the following areas: air supply device, room flow field, simplified methods, and evaluation. Moser's conclusions include:

- CFD simulations are useful when values of difficult-to-measure variables are needed in all points of the flow field.
- Simulations are useful to study trends (sensitivity of flow patterns to small changes).
- Simulations are useful to predict airflow patterns for critical projects, i.e. when neither similar experience nor measured data exist (such as large spaces, unconventional ventilating systems, strong buoyancy effects).

Several researchers have reported on long term efforts to develop and apply CFD programs to room airflow modelling. In an early report, Chen et al. (1988) applied a CFD program and a building energy load program to examine the airflow and temperature distribution patterns in a simple room with different ventilation systems and rates. The load program was used to supply boundary conditions for the CFD simulation. Techniques for using the CFD results in annual energy simulations are also discussed. Chen found high temperature and ventilation efficiencies for two of the systems examined, larger temperature and ventilation efficiencies at higher ventilation rates, and that a larger internal gain in the room lowered the ventilation efficiency but had little effect on the temperature efficiency. In later work, Chen et al. (1992) evaluated the performance of four ventilation system types in a classroom with a low ventilation rate. The ventilation options evaluated were displacement ventilation, well-mixed ventilation, and two low-wall-diffuser ventilation systems. The pupils and desks were represented as aerodynamic blockages generating heat and CO_2. The results showed that the secondary flow due to buoyancy dominated the airflow pattern which resulted in similar overall ventilation effectiveness and thermal comfort for all four cases except near the diffusers. The predicted percent dissatisfied (PPD) and percent dissatisfied due to draft (PDd) were less than 15% and 20% respectively for all cases. The average CO_2 concentration was similar in all cases as the expected benefit of the displacement ventilation system was negated by recirculation patterns caused by the strong buoyancy driven flow. The perceived air quality was not satisfactory but could be improved by increasing the outdoor airflow to 8 L/s (16 cfm) per person. More recently, Chen et al. (1995) applied CFD with both conjugate heat transfer (CHT) and radiation models to examine the thermal response of a room and issues related to sensor location. Only surface-to-surface radiation was included in this study.

In another major effort, Awbi et al. (1992) applied a CFD program with radiative heat exchange to predict thermal comfort in both mechanically and naturally ventilated offices. Thermal comfort is evaluated in terms of predicted mean vote (PMV), PPD, and mean radiative temperature. Predictions for an office module demonstrated the importance of taking into account radiative heat exchange for the prediction of thermal comfort when the surface temperature difference is large. PMV was predicted for a naturally ventilated office with a closed door, partially open window, and heat sources (occupant, radiator, and instruments) modeled as obstacles with heat flux from the surfaces. Awbi (1993) evaluated the effectiveness of several ventilation strategies in a two-dimensional room using the same CFD program. The strategies investigated included ceiling supply, floor displacement, upward displacement, and downward displacement. Local and occupied zone average effectiveness parameters calculated include the contaminant removal effectiveness (for a point source of CO_2) and the temperature effectiveness. Comparisons of laminar and turbulent cases showed that turbulence has a major influence on air movement in a room and reliable turbulence models and accurate boundary conditions must be used. The contaminant removal effectiveness was found to have a small dependence on air change rate but a much larger dependence on the air supply position and the room load. The upward displacement system performed well under a range of conditions. Additional room airflow cases reported include cooling and heating situations in an office (Awbi and Gan 1993), an air conditioned office module, a down-flow clean room and a naturally ventilated classroom (Awbi and Gan 1991).

Li (1992) describes the development of a CFD simulation code with an emphasis on the simulation of radiation heat transfer and its coupling with CFD. One application of the model was to examine the effects of radiation on the airflow in a room with displacement ventilation. It was found that radiation plays a considerable role in thermal stratification with displacement ventilation. Other topics explored include the development and application through CFD calculations of three methods for predicting age of air and air change efficiency in ventilated rooms. Problems studied more recently include development of a method to deal with complex geometries using non-body-fitted Cartesian grids (Li 1994), application of CFD simulations to evaluate several measures of ventilation system performance including contribution ratios, age of air, and residual life of air in a two-dimensional ventilated space (Li et al. 1996c), and an investigation of the flow patterns resulting from colliding free convection boundary layers such as occurs in the case of a cold window located above a radiator (Li et al. 1996d).

In the first part of a three-part paper, Baker et al. (1994) derives a new CFD analysis method, addresses issues involved in turbulence modelling for the room air motion problem class and states that LES may yield the superior turbulence closure model in the long term. In part two, Williams et al. (1994a) focuses on solution of the equations and also compares their CFD predictions with other predictions and measurements for several non-room airflow laminar benchmark cases. In part three, Williams et al. (1994b) continues the work with a comparison to a full-scale room air experiment. The problem modeled is a room with supply and exhaust with all rectangular geometry. A comparison to earlier measurements for air change rates in the test chamber of 15 h^{-1} and 30 h^{-1} was made with good results for the lower air change rate and poor results for the higher. In a follow-up study, Williams and Baker (1994) applied the CFD program to predict natural convection in adjoining hot and cold rooms.

Researchers have reported a number of miscellaneous applications of CFD to room airflow. For example, Knappmiller and Kirkpatrick (1995) compared the performance of conventional ventilation and cold-air distribution systems in a simple room. They examined ventilation effectiveness and air diffusion performance index (ADPI), and concluded that a system designed for acceptable ADPI will produce a ventilation effectiveness greater than 90% within 5 minutes after a source is introduced. Kirkpatrick and Knappmiller (1996) continued this work and examined the ADPI for a cold air jet with varied conditions of jet outlet size, temperature, momentum, Archimedes number, heat source location, and heat load. The results indicate that an optimum ADPI was obtained when the jet separation distance is approximately equal to the room characteristic length. Krafthefer and Shah (1995) investigated the implications on room air motion and temperature profiles during on/off cycling of a baseboard heater by a thermostat. The modeled room was a simple rectangular two-dimensional room with two adiabatic walls, two walls with known heat loss, and a simple heat input for the baseboard heater. The results demonstrate a lag between the heater operation and thermostat response depending on thermostat location but calls out the importance of accounting for other effects such as wall conduction, thermal mass, and radiation. Lee and Li (1994) studied the airflow in a floor of an office building consisting of both an open office area and individual offices. They studied the impact of door opening and the presence of cabinets on the airflow pattern for a heating condition and concluded that the cabinets only have a local effect and that the closed doors affected the airflow in the open plan area but not within the individual offices. Moshfegh and Sandberg (1996) predicted the thermal and airflow conditions in a room with a radiant cooling ceiling panel. Imano et al. (1996) reported on transient CFD simulations, employing coupled radiation and conduction heat transfer models, of the temperature rise in a telecommunications equipment room after cooling system breakdown. Transient airflows were calculated for the first 30 minutes of the total three-hour period considered, but only transient maximum temperatures were reported. Collineau et al. (1996) compared predictions and measurements of ventilation in a painting area. They present an uncertainty analysis for the predictions and conclude the CFD model leads to acceptable uncertainty (8% to 18%) for the cases considered. Svidt (1994) investigated the airflow in a ventilated livestock building. Plett et al. (1993) used a simple CFD model to predict air movement patterns in partitioned offices. They observed that the partitions distort airflow patterns and that it would be difficult to ventilate a row of partitioned offices with a single supply. Tang and Holmberg (1993) examined the performance of a horizontal flow ventilation supply diffuser with varying characteristics. Bergstrom (1994) also predicted the flow from a warm jet in a cool room using a CFD program with an ASM turbulence model. Results presented include isotherms at several time steps during the development of the jet. Christensen (1992) presented simulations of the airflow from a supply jet in a room with ceiling-mounted obstacles.

It was found that it was difficult to predict the critical size of the obstacle that would deflect the jet such that it would not re-attach to the ceiling. Murakami et al. (1994) compared the capability of three turbulence models at predicting the flow in a room with a horizontal, nonisothermal jet from a diffuser. Zhang et al. (1992) compared computations and measurements of air velocity, turbulent kinetic energy and temperature in a full-scale office room under non-isothermal conditions. Diffuser conditions were specified with uniform profiles for velocity, temperature, and turbulent kinetic energy and with velocity based on measured jet momentum.

Varying inlet/outlet arrangements

A common application of CFD simulation is to study the performance of ventilation systems with different diffusers and different inlet and outlet arrangements. In a detailed study, Murakami et al. (1989) analyzed airflow and contaminant diffusion in several types of clean rooms with different supply and exhaust diffuser arrangements. The cases modeled included three basic room geometries with variations in the number of supply outlets and exhaust inlets and in the height of the exhaust inlets. The performance of the ventilation systems are expressed by four measures based on the contaminant concentration distribution from a point source, spatial average concentration (also called first Scale of Ventilation Efficiency or SVE1), mean radius of diffusion (SVE2), and concentration distribution from uniform contaminant generation (SVE3). The following conclusions are reached: (1) numerical simulation is useful for parametrically analyzing changes in flow conditions for complex conditions, (2) supply outlets have a large influence on both flow fields and contaminant diffusion fields, (3) the arrangement of exhaust inlets has a small influence on flow fields but a large influence on contaminant diffusion fields, and (4) arrangement of supply outlets in a checkered pattern is superior to a linear pattern in terms of ventilation effectiveness. In a follow-up report, Kato et al. (1992) describe three additional measures of ventilation system performance determined from the calculated distribution of pollutant and present CFD predictions of these measures for a clean room. The new measures are contribution ratio of a supply opening (SVE4), contribution ratio of an exhaust opening (SVE5), and residual lifetime of air (SVE6). In another follow-up, Kato et al. (1994) describe three measures for assessing the contribution of heat sources and sinks to the temperature distribution in a room and demonstrates the scales with a CFD simulation.

Similarly, Chung and Lee (1996) examined air diffusion and thermal comfort in a room with different combinations of inlet and outlet diffusers. Predicted air movement, temperature contours, PPD, and ADPI were compared for a room with one window, a convective heat gain and cold air supply. All three inlet-outlet cases considered resulted in PPD of about 25% in part of the room. It was also found that both PPD and ADPI generally increased with inlet air velocity although the effect was not consistent. The use of ADPI alone to characterize air diffusion performance or draft risk is considered inadequate. The PPD decreased linearly with increasing inlet air temperature. Haghighat et al. (1991) analyzed ventilation effectiveness in a two-zone enclosure with the locations of the door between the zones and the supply and exhaust openings varied. While the inlet location and the relative position of door opening and inlet influenced the ventilation effectiveness as measured by age of air, the exhaust location had little effect. Mizutani et al. (1996) reported experiments and simulations of ventilation system performance in a room for both constant temperature and heating cases with different supply/return locations. They concluded that the locations had little impact on the ventilation performance for the constant temperature case but the combination of ceiling supply and ceiling return performed poorly for the heating case. Nho and Kim (1996) examined the effects of ventilation inlet/outlet and heat source location on the thermal environment in a telecommunications equipment room.

Diffusers

Moser (1991) identified modelling the airflow supplied from the diffuser as a critical point in applying CFD to room airflow. Several other recent studies have also focused on this issue. For example, Emvin and Davidson (1996) discuss four methods of representing a diffuser consisting of 84 jets. The four methods include detailed modelling of all jets, representing the diffuser as a single jet with an inlet area equal to the sum of the individual nozzles (basic model), representing the diffuser as a single jet with an inlet area equal to the total area of the diffuser with decoupled mass flux and momentum flux boundary conditions (momentum method), and 'box' model (described below as PV method). CFD results are presented for the detailed model and the basic model. They concluded that the detailed model is accurate but expensive, the basic model gives poor results for diffusers with small ratios of nozzle and diffuser areas, the momentum method is inconsistent and can only give rough estimates for coarse meshes, the box model should perform as well as the detailed model but requires measurements, and further work is needed to develop a simplified inlet model without performing any measurements or sacrificing accuracy.

In another recent study, Skovgard and Nielsen (1992) compared two techniques of modelling diffuser flow including modelling the diffuser directly and modelling the resulting flow pattern in a volume in front of the diffuser (PV method). They concluded that the PV method had the best performance but it depends on diffuser specific data. Chen and Jiang (1996) simulated the airflow from a curved surface diffuser using three different grid systems including cylindrical coordinates with small steps, body-fitted coordinates, and unstructured grids. They found that the body-fitted and unstructured grids produced flow patterns that agreed well with flow visualization techniques but required high labor costs for setting up the simulation. The cylinder coordinate could not predict the airflow pattern correctly because excessively high turbulence is calculated in the boundary layer with small steps. Huo et al. (1996) describes a new method to describe diffuser boundary conditions, called jet main region specification, that takes advantage of existing diffuser characteristic equations and manufacturer's data. They conclude that the method may be used to accurately specify diffuser boundary conditions without describing the complicated diffuser geometry and, therefore, save simulation time by using a coarser grid. Heikkinen and Piira (1994) compare several CFD models of circular ceiling diffusers in a rectangular grid. They conclude that at least 8 grid cells in both horizontal directions is necessary to produce a satisfactory result. A problem remained with the jet growth rate being too small which was attributed to either the Cartesian geometry or the k-ε turbulence model. Joubert et al. (1996) studied the impact of several inlet boundary condition variables including the values of k and ε and the profile (flat or parabolic) of the incoming jet. They found that k and ε did not significantly impact the flow pattern. The inlet profile affected the fluctuating component but not the average flow.

Occupant effects

Another topic of recent interest has been modelling the impact of occupants on room airflow conditions. Depecker et al. (1996) performed CFD simulations to compare alternative ventilation systems in an operating room with aerodynamic blockages representing the occupants. Brohus and Nielsen (1996) also present several CFD models of persons represented as blockages with heat supply at the surface. Comparison with wind tunnel experiments showed that the models were capable of simulating the important features of flow around a person and that the inclusion of 'legs' may be important for some cases. Nielsen et al. (1996) investigated the influence of obstacles in the occupied zone of a room with mixing ventilation and found that the flow in the entire room is affected and the maximum velocity is decreased. Kalzuka et al. (1992) predicted the steady-state thermal environment in a room heated with hot water floor panels using a CFD model with the CHT method and a radiative exchange model. Thermal comfort was analyzed through temperature, mean and vector radiant temperature, PMV, and standard effective temperature. Iwamoto (1996) extended this work to include a simple model of an occupant in the room. Schaelin and Kofoed (1994) simulated and measured the airflow and temperature distribution in ventilated rooms with thermal plumes due to both a point heat source and a cylindrical source designed to represent a human body. Tjelflaat and Knott (1996) also describe a method for CFD modelling of an occupant in a room.

Displacement ventilation

A specialized room airflow application simulated by many researchers is displacement ventilation systems. In a displacement ventilated room, air is supplied near or at the floor and exhausted near or at the ceiling. There is no attempt to mix the ventilation air uniformly throughout the room as is done with a conventional mixing ventilation system. In an early work, Mathisen (1989) examined thermal conditions resulting in a large auditorium with a displacement ventilation system through both CFD and measurement. He observed complex airflow patterns with no stratification of the air. CFD modelling showed that temperature gradients in a large space depended on where measurements were taken. Issues raised include local mixing evening out strong temperature gradients, measurement of efficiency under different conditions, details in airflow pattern with regard to thermal comfort, and comparison between displacement and complete mixing.

More recently, the effects of the outdoor environment on the flow and temperature fields and heat transfer in a displacement ventilated room were reported by Li et al. (1996a and 1996b). The study considered both heat conduction through walls and thermal radiation through windows and between surfaces. The authors found that the effects on the flow pattern were significant, in some cases adding to the displacement mechanism and, in others, weakening it. Also, they found that the outdoor environment can be more than 50% of the total cooling load and can also affect the spatial distribution of the heat gain.

Other recent displacement ventilation studies focusing on the thermal environment and/or ventilation effectiveness have been published. To confirm and improve the design of an underfloor air-conditioning system in the early design stage, Matsunawa et al. (1995) performed experiments and simulations to confirm that the system could remove heat generated by equipment without returning it to the air-conditioning unit. Alamdari et al. (1994) reported simulations of airflows and temperature distributions in an open-plan office space with a displacement ventilation system. They found that the secondary airflows resulting from infiltration and cold surfaces can adversely affect the ventilation performance and reduce thermal comfort. Jiang and Haghighat (1992) examined the effectiveness of a displacement ventilation system in a partitioned office with five different partition layouts. Contaminant concentration and average age of air results showed that the arrangement of the partitions was more important than the number of partitions and the advantage of displacement ventilation is diminished unless the partitions are carefully arranged. Regard et al. (1995) predicted airflows, tracer gas concentrations, and age of air in a classroom with displacement ventilation. Gan (1994) studied the thermal environment and ventilation system performance in an office with a displacement ventilation system and found that the optimal supply air conditions vary with the distance between the air diffuser and occupant and that increasing the airflow rate improves IAQ but may result in local thermal discomfort. Niu and Kooi (1993) studied displacement ventilation systems with and without chilled ceilings and found that the system with chilled ceilings provided good thermal comfort and ventilation effectiveness at a cooling load of 50 W/m^2 (4.6 W/ft^2). Cox and Elkhuizen (1993) also studied the performance of a displacement ventilation system in an office and concluded that the displacement system provided no improvement compared to a mixing ventilation system for the breathing zone. In a study with contradicting results, Awbi (1996) compared the performance of displacement and mixing ventilation systems for an office and concluded that the displacement system could provide similar air quality in the breathing zone with half the ventilation rate of the mixing system. Nakamura et al. (1996) compared the thermal comfort and energy performance of conventional, displacement and a hybrid system called task-ambient air conditioning which combines conventional ventilation with local displacement ventilation at a task area. The authors concluded that task-ambient system can effectively cool the local area where cooling loads are concentrated, reduce the thermal stratification of displacement ventilation, create comfortable local thermal environments, and save energy by increasing temperature in surrounding area. Manzoni et al. (1996) studied the influence of several parameters including heat release rate, supply flow rate, and presence of cold walls on the thermal environment in an enclosure with displacement ventilation.

A few displacement ventilation studies have focused on other issues. Jacobsen and Nielsen (1993) studied the thermal environment in a displacement-ventilated room with heat sources and examined the extension of the k-ε turbulence model with a buoyancy factor to account for turbulent viscosity dependency on vertical temperature gradients. Chen and Chao (1996) investigated the flow in a turbulent buoyant plume and in a displacement ventilated room with obstacles. In a slightly different displacement ventilation study, Aihara et al. (1996) applied a version of the k-ε model called 'depth average' to study the air distribution and supply flow rate of the underfloor chamber supplying the ventilation flow.

Large enclosures

Another specialized room airflow application simulated by many researchers is airflow within a large enclosure. In one detailed report, Murakami (1994) describes many of the unique characteristics of large enclosures (capacity of space, height of ceiling, and potentially small occupied zone), ventilation design principles, and prediction methods including both simple equations, scale model experiments, and CFD modelling. Important CFD modelling issues discussed include choice of turbulence models, grid discretization, and boundary conditions. Three case studies including an airport terminal lobby, an atrium between two office buildings, and a separate atrium are presented briefly, including temperature and airflow distribution results.

Atria have been the most commonly studied large enclosure, for example as was reported by Kato et al. (1995). This study investigated the temperature and flow fields in a partially air-conditioned atrium with a CFD program including consideration of both solar and infrared radiation. The 130 m (425 ft) high atrium is open to the adjoining office spaces located on the north and south sides. An external building energy program was used to estimate the heat sink or source rate due to the heat storage effect of the walls. Seven cases were analyzed including four cooling conditions and three heating conditions, using a variety of ambient temperatures. Two cases included roof top ventilation. Flow fields and temperature distributions are presented for several cases. The results show that, when the outdoor temperature is below 27 °C (80°F), rooftop ventilation is effective in exhausting the hot air accumulated below the ceiling and in reducing the cooling load of the upper bridges. In both heating and cooling cases, large-scale recirculation in the void space is promoted by an imbalance of heat transfer to the atrium. Also, in the winter, the large-scale recirculation results in a strong downwash at the bottom of the cooled side of the void space.

Another detailed study of an atrium is reported by Off et al. (1994), Moser et al. (1995) and Off et al. (1996). The authors describe the development and application of a model coupling CFD with heat transfer by convection, conduction, and radiation with thermal storage in an atrium. The heat transfer in walls was calculated through two methods - an off-line calculation with a simple program for conduction and radiative exchange and the CHT approach. A two-band model for surface-to-surface radiation with application to atria is addressed briefly. Off et al. (1996) describes the application of the model to perform transient simulations of the thermal conditions in a small experimental atrium.

Schild et al. (1995) provides guidance for modelling atria in CFD. The importance of and methods for accounting for solar radiation distribution are discussed. CFD boundary conditions are also reviewed including a new study to establish the most accurate way of defining natural convection boundary conditions. Five common methods of prescribing boundary conditions were investigated with two different grids. The results indicated three preferred methods. Optimal near-wall grid refinement was also studied. Another conclusion was that it is vital that CFD simulations of atria allow for surface-to-surface radiation exchange and that supply and exhaust openings are modeled accurately (methods of doing so are discussed). Schild also discusses modelling infiltration with the conclusion that the simplest approach is to estimate infiltration rates by hand and impose them in the code. A final topic discussed is transient aspects of CFD simulation including the possibility of linking a code with a building thermal analysis program. More recently, Schild (1996) reports the application of a CFD model employing CHT and long-wave and solar radiation models to predict the conditions in an experimental atrium.

Many other studies have applied CFD to the airflow in atria. Awbi and Baizhan (1994) recently modeled thermal and airflow performance in an atrium with solar gain calculated using a building thermal analysis program and thermal comfort analyzed using the PPD index. They concluded that acceptable thermal environment can be achieved in summer using outdoor air through large openings, but warm air from adjacent zones is required in winter. Takahashi et al. (1992) compares the results of simulations of the airflow and temperatures in a hotel atrium performed during the design with later measurements. Chikamoto et al. (1992) compared the results of simulations using a modified k-ε turbulence model for a simple atrium with measurements in a scale model. Ozeki et al. (1992) describes the development of a simulation method including solar radiation, radiative heat transfer, and flow distribution based on the k-ε turbulence model, and Sonda et al. (1992) reports on the application to simulation of temperature and flow field in an atrium. Ozeki et al. (1996) continued the CFD modelling of an atrium to examine the impact of various boundary conditions and other modelling choices. Kondo and Niwa (1992) applied both a CFD model and a macroscopic model to study airflows and temperatures in an atrium with general agreement between the models. Lemaire (1990) reports an early study of thermal comfort in large atria through CFD simulation. Lam et al. (1993) applied EXACT3 to an atrium with balconies. Guntermann (1994) describes both experiments and CFD simulation of airflow in an atrium. Alamdari et al. (1991) describe the application of a CFD program to evaluate the thermal and airflow performance of a naturally ventilated open atrium office building under both winter and summer conditions. Svidt et al. (1996) analyzed the application of natural ventilation in an atrium with both measurements and CFD simulations. They found that the system worked well under both winter and summer conditions. Renz and Vogl (1996) report predictions of the airflow in an atrium using a CFD model with a low Reynolds number extension to the k–ε turbulence model and a new wall boundary model proposed for mixed convection boundary layers.

Other types of large enclosures have also been the subject of CFD simulation study. Clancy et al. (1996) studied the application of natural ventilation to a large auditorium. Van der Maas and Schaelin (1995) studied the effect of using air curtains to prevent heat loss due to door openings in airplane hangars. Lefeuvre et al. (1995) examined the airflow and temperature conditions in an enclosure covering the street between buildings for four cases involving different ventilation systems and geometric design. Guthrie et al. (1992) found that predictions of airflow in an airport passenger terminal building were generally confirmed by a reduced-scale physical model. Ward and Wang (1994) studied the airflow and thermal environment in a large church. Borresen and Madsen (1992) simulated the thermal environment in a 350,000 m^3 (1.2×10^7 ft^3) speed skating rink. Chow and Fung (1992) found good agreement between CFD predictions and measured airflows, temperatures, and humidities for an air-conditioned gymnasium. Chow (1995) reported additional simulations for a waiting hall at a railway station and for a mechanically-ventilated underground parking garage. Grundmann et al. (1994) also reported both CFD predictions and measurements of airflows and temperatures in a gymnasium but did not compare them as they found more detailed measurements were needed. Nielsen (1995) applied CFD, scale model experimentation, and full scale measurement to a large enclosure. He concluded that CFD prediction is an important alternative to scale model experiments. Drangsholt (1993) predicted CO_2 concentrations in several occupied terraced auditoria with reasonable agreement with measurements. Borchiellini et al. (1994) also predicted CO_2 concentrations in an auditorium. Vazquez et al. (1991) presented CFD simulations of airflow patterns and temperatures in an auditorium with varied ventilation system parameters. Havet et al. (1994) applied coupled CFD and conductive, convective, and radiative heat transfer to study radiant panel heating in a large enclosure. Yoon et al. (1996) investigated the thermal environment of a railway station during the design stage of a project. Fontaine and Rapp (1996) report on CFD simulations of indoor air quality during the construction of an underground parking garage. Chow (1996) also presents the results of CFD simulations of airflow due to ventilation in an underground parking garage. Frydenlund et al. (1996) report transient CFD simulations with a model employing CHT of conditions in an underground ice hockey stadium. Fukoyo et al. (1996) report CFD simulations of the airflow in a subway station. Kondo et al. (1996) describe simulations of the temperature and humidity distribution in an indoor skating rink. In another unique application, Xu et al. (1996) studied the flow pattern of a ventilation system used during repair work inside the hull of a large ship. Buchmann et al. (1994) present predictions of airflow patterns in a heated and cooled large enclosure. Yau and Whittle (1991) report both scale-model measurements and CFD simulations of airflow in a large airport terminal building.

Natural Ventilation

In addition to the application to large enclosures mentioned above, many researchers have studied various approaches to natural ventilation of buildings using CFD simulation. For example, Tsutumi et al. (1992) applied CFD to study cross-ventilation in a single family house. The cross-ventilation system involves large openings (open windows) on opposite sides of the building with wind blowing through the openings. The airflow around the building and the indoor airflow were modeled simultaneously. Cases examined included both a single room and two rooms divided by an interior partition with an open door and different calculation meshes. The predicted ventilation rates were about 20 to 30% larger than those calculated by a simple equation. Although no comparisons are made to measurements, it was concluded that cross-ventilation can be simulated well if the indoor and outdoor airflow is calculated continuously. In a related study, Iwamoto et al. (1992) examined cross-ventilation in a four-story residential building. In addition to airflows and temperatures, PMV was calculated to evaluate the thermal comfort in the building. He et al. (1996) followed-up by examining the effect of wall thickness and bay windows on cross-ventilation. Both were found to result in the airflow passing more directly through the indoor space. The cross-ventilation rate initially decreases with thicker walls but increases as they become bay windows.

Gan et al. (1991) examined thermal comfort in a naturally-ventilated classroom. Various combinations of open windows were simulated, however the simulation did not include the wind on the outside of the building. Occupants were simulated as obstacles generating 100 W. The results indicate that, in a mild climate, adequate thermal comfort can be achieved by appropriate arrangement of windows and doors but additional cooling may be required in a hot climate. Also, they found that the airflow patterns and temperature distribution were greatly influenced by the occupants and their distribution.

Other CFD studies of natural ventilation have been reported recently. Satwiko and Donn (1995) modeled an 'Indonesian roof chimney', a system designed such that solar radiation can efficiently amplify the buoyancy of air in an attic to induce indoor ventilation. They modeled four different configurations and also performed physical experiments. Barozzi et al. (1991) also studied a solar driven passive ventilation system through both simulations and experiments. Similarly, Wang

(1995) used both CFD simulation and wind tunnel tests to study cross-ventilation through an apartment. The difference in predicted and measured air change rates for various cases studied was about 10%. Wang (1996) recently reported CFD simulations of cross-ventilation rates in six apartments with different internal configurations. Jones et al. (1991) studied the infiltration rates in a naturally ventilated industrial building using both multizone and CFD simulations. Kornaat and Lemaire (1994) examined the CO distribution due to automobiles in a parking garage with natural ventilation and determined that a mixing fan was needed to prevent unacceptably high local concentrations. Smith et al. (1992) studied natural ventilation in an office building using a single-cell ventilation model, a dynamic thermal model, and a CFD model. The CFD model was used to calculate the detailed internal conditions of air temperature and velocity after airflows at openings were obtained from the results of the single-cell modelling. Shao and Riffat (1996) simulated the application of heat pipes for heat recovery in natural ventilation stacks. Li and Teh (1996) investigated the natural ventilation flow in a room through a single large opening.

Contaminant transport

Many studies have reported CFD simulations for steady state cases of airflow and inert contaminant transport. For example, Nagano and Mimi (1992) examined airflows and pollutant concentrations in a rectangular, two-dimensional space with different combinations of floor and ceiling supplies and exhausts and Reynolds numbers from 5 to 10,000. The pollutant was generated from a source distributed along the floor. They found that for most cases the ceiling supply offered better ventilation efficiency than the floor supply. Similarly, Han (1992) used a calculated ventilation effectiveness in a two-dimensional, isothermal room. Results are presented as concentration patterns and local decay rates and local mean age of air. Chaturvedi and Mohieldin (1989) presented CO_2 concentrations from a line source in the floor of a two-dimensional, isothermal room. Rota et al. (1994) reported measurements and simulations of a tracer gas in room. One case involved a large partition which created a small, unventilated region of high pollutant concentration. They concluded that the predictions were in reasonably good agreement with measurements, however the CFD model overpredicted the concentrations in the unventilated region. Ohira and Omori (1996) reported measurements and simulations of exhaust gas concentration due to gas water heaters located in large stairwells in the center of Japanese buildings. Kato et al. (1996) report on simulations of contaminant distribution in a room with a displacement ventilation system and one occupant. Contaminant source cases considered include generation throughout room, from ceiling, from source, and from occupant surface. Tjelflaat and Bell (1996) present CFD simulations performed for pollutant concentrations in a fiberglass factory. The simulations were used to evaluate the addition of local exhaust to the existing ventilation system. Teng et al. (1996) simulated the CO_2 concentrations in a bedroom ventilated by opened windows. Limited results are presented as airflow patterns, temperatures and concentrations. Beghein et al. (1994) present simulations of airflow patterns and pollutant concentrations in a ventilated enclosure with an emphasis on examining the influence of pollutant diffusion coefficients. Several reports already discussed above as room airflow case studies also calculated steady state pollutant concentrations including Jiang and Hahgighat (1992) for an office with displacement ventilation, Regard et al. (1995) for a classroom, Knappmiller and Kirkpatrick (1995) for a room with a cold air distribution system, Murakami et al. (1989) and Kato et al. (1992) for a clean room, and Horstman (1988) and Mizuno and Warfield (1992) for an aircraft cabin.

Very few studies have reported CFD simulations of transient cases of both airflow and contaminant transport. In one such study, Cafaro et al. (1992) studied contaminant transport in a simple room to gain insight into the issue of natural gas leaks. The situations analyzed included both pollutant decay from an initial uniform concentration and pollutant buildup due to a source. For both situations, ventilation inlet and exhaust location, type, and supply flow rate were varied.

The more common approach taken for transient contaminant transport studies involves using a steady-state flow solution to solve for transient pollutant concentrations generated by a source. An example of this approach is Roy et al. (1993) which reports on contaminant transport in a simple room with different air change rates. Contaminant distributions resulting from contaminant injected in the supply vent are presented at different points in time up to 25 seconds. Suyama and Aoyama (1992) also used the steady flow solution method in combination with measurements to examine airflows, temperatures and pollutant concentrations in a real office. Two cases with contaminant sources and a third case with an initial uniform pollutant concentration but no source were examined. They concluded that the use of CFD simulation combined with limited field measurements for verification is an effective method for studying the indoor air in a building with the least labor and least disturbance of the occupants. Haghighat et al. (1994) used a similar method to study the impact of ventilation rate and partition layout on the volatile organic compound (VOC) emission rate from a source. The VOC source model used allowed for VOC emissions dependent on the resulting nonuniform contaminant concentration distribution. Riffat and Shao (1994) also used this method to examine the impact of tracer gas species on the mixing of tracer gas in situations of both increasing and decaying concentrations.

While the reports described above involved the convection and diffusion of inert pollutants, a few researchers have studied the more difficult problem of particle distribution including the issue of particle deposition. Lu and Howarth (1996) describe the development of a method to predict particle deposition using CFD and application to a two-zone chamber. The particle tracking model employed a Lagrangian approach with assumptions of no heat and mass transfer between air and particles, no particle rebound from surfaces, spherical solid particles, and motion governed by Newton's second law. The particles were divided into 10 groups from a diameter of 1 μm to 10 μm and had a density of 850 kg/m^3. They conclude that deposition and migration are mostly influenced by airflow patterns and particle properties, the majority of particle migrations occur within the first 10 min of particle tracking time, large particles deposit much faster than small, and small particles can remain in suspension for longer than two hours. In other work examining particle concentrations in indoor air, Goddard et al. (1996) and Byrne et al. (1995) report on the development of CFD modelling capabilities to account for particle deposition processes. The reports describe both experimental work and important issues in simulating deposition such as turbulent transport of particles close to the boundary layer. Sample simulations are presented. Similarly, Fontaine et al. (1994) report on both CFD simulations with aerosol transport using a Lagrangian model and water scale model experiments. They found good agreement between predicted and measured results. Cheong et al. (1995) also studied particle movement through both chamber tests and CFD simulations. A comparison is made between transient particle concentrations for a case with an initial uniform concentration. The results indicated a quicker concentration decay in the experiment. It was concluded that better agreement, in terms of particle deposition and resuspension, could be obtained with improved simulation boundary conditions.

Several researchers have reported on the modelling of moisture as a contaminant in CFD simulations. For example, Kolokotroni and Littler (1993) modelled moisture as an inert contaminant (no adsorption or condensation), thus the air movement was the only determining factor in the distribution of the moisture. Several cases were analyzed with the locations of the ventilation supply, exhaust and moisture source varied. Similarly, Kondo et al. (1996) simulated the moisture distribution in an indoor skating rink, and Vitale et al. (1996) described a method to simulate moisture distribution in conjunction with a CFD simulation.

Other Applications

Many researchers have applied CFD modelling to a variety of other building IAQ and ventilation related topics including exhaust ventilation systems, flow around buildings, air curtains, flow in vehicle enclosures, and flow in duct systems.

Several researchers have reported on the application of CFD modelling to exhaust ventilation systems. Schaub et al. (1995), Cardinale et al. (1993), and Li et al. (1996) modelled kitchen exhaust systems. While Schaub presented mainly flow field results, Cardinale and Li focused on prediction of kitchen exhaust hood capture efficiency. Whittle and Lam (1992) modelled the ventilation performance in a pharmaceutical laboratory containing exhaust hoods and presented sample results including air velocity vectors and speed contours. Fontaine and Rapp (1994) studied the performance of a local exhaust ventilation system in a painting workshop.

CFD simulations have recently been applied to study the pressures caused by wind flow around a building. For example, Stathopolous and Zhou (1995) and Zhou and Stathopoulos (1996) compared simulations and measurements of wind-induced pressures on the roof of a building with a focus on the performance of different versions of the k-ε turbulence model. Selvam (1996) compared predictions of the k-ε model with different solution procedures and grid refinements. Shao et al. (1993) and Walker et al. (1993) examined a practical application of natural ventilation in building courtyards to determine the conditions under which good courtyard ventilation is ensured. Vanderheyden and Schuyler (1994) describe three alternative methods, including CFD simulation, of evaluating the impact of cooling tower emissions on indoor air quality. Only very limited results from a single two-dimensional simulation are presented. Ong (1989) applied CFD to study the dilution of stack emissions from a factory with simulation results used to reassure local environment authorities that pollution levels would be within acceptable limits. Cooper (1993) compared CFD predictions to wind tunnel tests of the pressure distribution around a sheltered building and found that the pressure coefficient values differed but trends were predicted fairly well.

Another application reported on by several researchers is flow within a vehicle enclosure. Horstman (1988) describes the development of a CFD program and its application to airflow in an aircraft cabin. Airflow pattern and velocity distribution predictions were compared with measurements in a test section with excellent agreement for the pattern and good agreement for the velocities. Mizuno and Warfield (1992) also applied CFD to study the airflow and CO_2

concentration distributions within an aircraft cabin. Predictions were compared with measurements in a full-size test and it was concluded that overall results show a good correlation between predictions and measurements, but some aspects need improvement. Xuejun et al. (1996) simulated the airflow and temperature conditions inside an automobile passenger compartment and concluded that the location of both inlet and outlet vents and the angle of the inlet vent have a great effect on the airflow velocity and temperature distribution in the compartment.

Another application of CFD reported by several researchers is air curtain performance. Lam et al. (1990) used CFD simulations to evaluate the performance of an air curtain at the door of a heated shopping mall. Airflow, temperature and heat loss results were found to be plausible and conformed well to physical interpretation, however, concerns about mesh independence and numerical discretization errors were expressed. As discussed earlier, Van der Maas and Schaelin (1995) applied a CFD to evaluate the effect of using air curtains to prevent heat loss due to door openings in airplane hangars. Ho and Goodfellow (1994) performed two-dimensional CFD simulations to investigate the use of both air curtains and an exhaust hood to control contaminants in a polymer manufacturing facility and concluded that a combination of both provided satisfactory contaminant control. Oliveeira et al. (1996) reported simulations of the performance of air curtains on an open display refrigerated cabinet.

A variety of other IAQ and ventilation related CFD applications have been reported by other researchers. For example, Shao and Riffat (1994) predicted the flow field in ducts to devise experiments to improve measurements of pressure loss downstream of fittings. Sharples and Palmer (1994) describe both experiments and CFD simulations used to determine equations for flow through cracks subject to fluctuating pressures. Similarly, Kohal et al. (1994) used both CFD simulations and experiments to examine buoyancy driven airflow through large horizontal openings. Baskaran (1994) applied CFD simulations to predict the performance of Pressure Equalized Rainscreen wall assemblies. They investigated the impact of model parameters (grid refinement, time step, etc.) on model predictions and analyzed the performance of wall assemblies with airtight and leaky barriers. Awbi and Gan (1992) and Onishi et al. (1996) simulated solar induced ventilation systems. Awbi and Gan compared CFD simulations of the airflow and heat transfer in a Trombe wall and a solar chimney to calculations with heat transfer and fluid flow equations for a closed channel with 'generally good' agreement. Onishi simulated the transient airflows and thermal environment in a passive solar room with a Trombe wall for a winter day.

Other researchers have reported studies aimed at either combining CFD with other types of building modelling programs, developing simplified models, or modelling an entire building. For example, Schaelin et al. (1993) discuss the coupling of a CFD program and a multizone airflow program through a method referred to as detailed flow path values (DFPV). The potential advantages of such a coupling are to provide greater detail for an important zone of a building through the CFD model and to use the multizone model to provide boundary conditions for the CFD model. Several simple examples are used to demonstrate the potential importance of such a coupling. Taking coupling a step further, Clarke et al. (1995) describe the coupling of CFD, multizone airflow, and building energy simulation implemented by maintaining each method's separate solution algorithm. An example application of the coupled method to a simple four-zone house is discussed. Similarly, Fischer and Rosler (1996) describe the coupling of a CFD model and building thermal model with an application to an experimental atrium and an office room with different heating systems discussed briefly. Simons et al. (1995) described a method to combine cells within a CFD model of a space to create a model with a small number of zones representing a single room. They conclude that it is possible to produce reliable contour diagrams of ventilation parameter variations from a small number of properly defined large zones which could be used both as a guide to locating tracer gas sampling points and as the basis of a simplified model for design calculations. Depecker et al. (1995) applied CFD to model the infiltration and interzone flows in a model of an eight-room apartment.

Summary

An extensive literature review on the application of CFD to building ventilation and IAQ problems was performed. Topics discussed in the literature include room airflow case studies involving calculation of airflow patterns, temperatures, ventilation system performance and thermal comfort for various ventilation systems, strategies and room configurations; flow from diffusers; modelling occupants; exhaust ventilation system performance; wind pressure distribution for flow around buildings; thermal and airflow performance in large enclosures; pollutant transport including particles and moisture; air curtains; pressure loss in ducts; coupling of CFD programs with multizone airflow models and/or building energy simulation models.

CFD modelling issues

This section discusses a number of issues identified in the published literature related to the application of CFD modelling to building ventilation and IAQ problems. Most of the studies mentioned below are also discussed in the previous section on applications. These are complex issues, and the discussion below presents just a sampling rather than a complete analysis of each issue.

Prediction accuracy

Prediction accuracy is certainly an important issue to all modelers. Many of the reports reviewed comment on this issue at least briefly although few examine it in detail. Some of the specific reasons for inaccurate predictions are addressed below as separate topics. This section incorporates only selected findings reported in the literature as this topic is worthy of an entire report unto itself.

Many researchers have reported good quantitative agreement between CFD predictions and measurements. In a study including both CHT and surface-to-surface radiative heat transfer, Chen et al. (1995) reported excellent agreement between predictions and measurements for air and surface temperatures. Chen also performed simulations with prescribed wall temperatures from a cooling load program replacing the CHT model, which resulted in reduced but still good agreement. Awbi et al. (1992) found velocity and temperature distributions predicted with a CFD program including radiative heat exchange in good agreement with the measured values in an office module. Horstman (1988) compared airflow pattern and velocity distribution predictions for airflow in an aircraft cabin with measurements in a test section and found excellent agreement for the pattern and good agreement for the velocities. Sonda et al. (1992) compared numerical and experimental results for an atrium and found good agreement when radiative heat transfer was coupled with convective heat transport. Chow and Fung (1992) found good agreement between CFD predictions and measured airflows, temperatures, and humidities for an air-conditioned gymnasium. Off et al. (1996) modeled an atrium including conductive and radiative effects and reported good agreement for both steady state and transient predictions of the thermal conditions. Matsunawa et al. (1995) found field measurements of temperatures and temperature effectiveness corresponded to CFD predictions for an underfloor air-conditioning system. In another displacement ventilation study, Cox and Elkhuizen (1993) compared calculations and measurements for airflow patterns, temperature effectiveness and ventilation effectiveness with 'fair' to 'good' agreement found.

Other researchers have reported poor quantitative results but satisfactory qualitative results or prediction of trends. For example, in a kitchen exhaust ventilation study, Cardinale et al. (1993) found the same trend of hood capture efficiency vs. hood flow rate as measurements, but poor quantitative results of the predictions. Lam et al. (1993) found good qualitative agreement between simulations and scale model experiments for the flow in an atrium with balconies. Similarly, Satwiko and Donn (1995) found CFD simulation results to be qualitatively consistent with scale model measurements in a natural ventilation system study. Zhang et al. (1992) compared computations and measurements of air velocity, turbulent kinetic energy and temperature in a full-scale ofice room under non-isothermal conditions. They reported a ‘reasonable agreement’ for airflow pattern and distribution patterns of velocity, temperature and turbulent kinetic energy. However, the quantitative comparison was poor due to numerical predictions of slower jet decay, narrower jet spread, and delayed drop of jet compared to the measurements. Other explanations for differences included the lack of a radiation model and unavoidable approximation in the specification of boundary conditions.

Still other researchers have reported problems with prediction accuracy. Depecker et al. (1995) found that predicted flows into and between zones of a model of an 8-room apartment compared poorly with measured airflows. Two reasons were cited for the differences including errors in the modelling process due to a poor fit between the models and physical reality and errors from input to the code due to poorly known boundary conditions. Clancy et al. (1996) reported large discrepancies between predicted and measured temperatures in study of natural ventilation in a large auditorium. Vogel et al. (1993) found discrepancies between predictions and measurements of the behaviour of the jet from the inlets in a simple test room. This problem was amplified for low air change rates. They also reported that calculated and measured heat transfer coefficients showed similar patterns but need further improvement. In an application to natural convection in adjoining hot and cold rooms with a dividing partition, Williams and Baker (1994) found good agreement for most of the experimentally observed features of the flow field but measured stratification in the hot cell was not well predicted.

Turbulence modelling

In a summary of IEA Annex 20 research, Moser (1991) identified turbulence modelling as one of five major technical problems encountered (only the k-ε turbulence model was used). Problems with the standard k-ε turbulence model have been identified by others and have led to research on modified versions and advanced turbulence models. Studies considering the LES model are discussed in a separate section below. Murakami et al. (1994) compared the capability of three turbulence models, including k-ε EVM, ASM, and DSM, at predicting the flow in a room with a horizontal, nonisothermal jet from a diffuser. Measurements and predictions indicated that the DSM model predicted the velocity distribution best, both DSM and ASM predicted temperatures better than k-ε, and the k-ε could not predict the anisotropic nature of the Reynolds stress in the jet. Xu et al. (1994) simulated isothermal airflow in a room using the standard k-ε model and three different low Reynolds number k-ε. All four models were in good agreement for predictions of the velocity field. The low Reynolds number models overpredicted the turbulence kinetic energy in the boundary layers. Stathopoulos and Zhou (1995) and Zhou and Stathopoulos (1996) compared predictions of the wind-induced pressure on the roof of a building for various versions of the k-ε turbulence model to measurements. They reported poor performance with the standard model but better predictions with a “two-layer” method combining the k-ε model with either a one-equation model in the near-wall area. Buchmann et al. (1994) found that a low Reynolds turbulence model improved predictions compared to the standard k-ε model for a large enclosure. In a comparison of several turbulence models, Chen and Chao (1996) found that a Reynolds stress model performed best for a turbulent buoyant plume case, while a RNG k-ε model performed best for a displacement ventilation room case. In another displacement ventilation study, Manzoni et al. (1996) attributed difficulties in predicting temperatures mainly to shortcomings of the k-ε turbulence model which may be improved through use of a low-Reynolds model. Takemasa et al. (1992) evaluated the accuracy of different wall functions in conjunction with the k-ε turbulence model at predicting convective wall heat flows in both heated and cooled rooms. They concluded that a wall function that depends on turbulence energy at wall-adjacent nodes and takes into account viscous sublayer thickness produces the most satisfactory results.

Large eddy simulation

Recently, a few studies have applied CFD models with LES turbulence modelling to building ventilation and IAQ research topics. All of these studies could be categorized with various topics in the Applications section above but are included here because of their use of LES. In an early report, Sakamoto and Matsuo (1980) compared predictions of isothermal flow in a simple ventilated room using both the k-ε turbulence model and large eddy simulation model to measurements. They concluded that both methods showed 'good' agreement with the experimental results without a noticeable difference between the two methods with regard to mean velocity. However, both methods had difficulty and the LES model corresponded better with the experimental results around the supply outlet.

Davidson and Nielsen (1996) reports a recent simulation effort employing LES modelling of airflow in a three-dimensional ventilated room. Simulations of a test case were performed at two levels of grid refinement and with two LES subgrid scale models (Smagorinsky and dynamic). Some flow results are presented for a simple test case and compared to measurement. The authors concluded that the Smagorinsky subgrid model was inadequate but the results obtained with the dynamic subgrid model were in good agreement with measurements.

Bennetsen (1996) also studied the application of LES to a three-dimensional room airflow test case. Simulations were performed with Smagorisnky, mixed-scale and dynamic subgrid model and the results were compared to CFD simulations employing the standard k-ε model, the low-Reynolds number k-ε model, a renormalization group k-ε model and experimental measurements. The authors found that the LES with a dynamic model agreed 'quite well' with measurements, but the LES with Smagorinsky and mixed-scale models suffered from inadequate grid resolution.

Murakami et al. (1996) compare predictions for flow around a building using CFD with four turbulence models (k-ε EVM, ASM, DSM, and LES) to measurements from wind tunnel tests. Comparisons are presented for mean velocity vectors, turbulent kinetic energy, mean surface pressure distribution, turbulence energy production, and normal stress components. Conclusions include that the LES agrees well with experimental measurements, the k-ε EVM included several serious discrepancies due to isotropic eddy viscosity hypothesis, some inaccuracies still exist in results of ASM, and DSM improved on some aspects of ASM but was worse on others.

Grid dependence

Another modelling issue commented on by several researchers is grid dependence. If the computational grid used in a simulation is too coarse, it may influence the solution. Lam et al. (1990) expressed concerns about mesh dependence and numerical discretization errors in simulations to evaluate the performance of an air curtain at the door of a heated shopping mall. Ward and Wang (1994) described difficulties in obtaining the necessary grid refinement in the geometry of a large irregular enclosure. Murakami (1994) also mentions the coarseness of the grid discretization as the main cause of difficulties in applying CFD to large enclosures. This concern was echoed more recently by Schild (1996). Moser 1991 identified problems with the computational grid and difficulty to reach grid independent solutions as one of the major problems encountered during the IEA Annex 20 project. Cafaro et al. (1992) identified lack of a grid-independent solution as a main difficulty in a study of the contaminant transport in a simple room. Borth and Suter (1994) investigated the influence of mesh refinement on room airflow predictions and proposed a dimensionless number to indicate the sufficiency of the grid refinement.

Other modelling issues

A number of other issues have been commented on by a variety of researchers. In a study of the application of CFD to an atrium, Moser et al. (1995) reports on many issues such as shortcomings of typical building energy simulation programs for coupling, complexities of the CHT approach, different time scales involved in the problem, coupling of radiation, convection and building dynamics, iterative solution of the coupled system, and solution strategies. Svidt et al. (1996) found that a CFD model was a useful tool for analyzing natural ventilation in an atrium but warned that boundary conditions have to be treated very carefully to obtain reliable results. Similarly, Clancy et al. (1996) warned about the importance of accurate representation of boundary in a study of natural ventilation in a large auditorium. In a study of the application of displacement ventilation to a classroom, Regard et al. (1995) also found that simulation results were extremely sensitive to the boundary conditions at the supply and a more detailed modelling of supply conditions may lead to better agreement with experimental results. In another study of displacement ventilation, Jacobsen and Nielsen (1993) echoed the significance of inlet boundary conditions and also discussed the importance of including wall and radiative heat transfer and describing the thermal plume above a heat source in detail. Cooper (1993) and other researchers have reported problems with attempts to restrict CFD simulations to two dimensions for a variety of problems.

Application to design

During the past 5 years, many studies have emphasized the potential for the application of CFD modelling to design problems. For example, Murakami (1994) concluded that it is highly desirable that a preliminary assessment of the indoor climate of a large enclosure be made before construction and that CFD simulation is promising for such assessments. Schild et al. (1995) suggests that CFD modelling should be used at a late stage in the design process for atria to check for satisfactory thermal comfort and indoor air quality by calculating PPD, mean radiant temperature draft risk, and ventilation efficiency indices. The degree of accuracy required in the simulation depends on the design tolerance in these indices. Whittle and Lam (1992) concluded that CFD has the potential to investigate the interactions between the many important factors at the design stage to ensure that the laboratory exhaust hood systems operate in the manner required. Lefeuvre et al. (1995) concluded that currently available tools are a starting point from which to provide useful design tools for large glazed spaces. Nielsen (1995) concluded that CFD prediction is an important alternative to scale model experiments and that it may be efficient to include two-dimensional CFD at an early stage in design phase for large enclosures. Alamdari et al. (1991) concluded that CFD models are capable of modelling the complex flow patterns in atriums but that it is desirable to have the CFD modeller and environmental engineering designer working together in an iterative manner.

Not all comments on the application of CFD to design have been completely positive. For example, Barozzi et al. (1991) found that CFD predictions needed further refinement but provided a useful tool in evaluating and guiding future design of a proposed solar driven passive ventilation system concept. Satwiko and Donn (1995) discussed some of the barriers, such as a lack of teaching physical sciences and advanced numerics to students, in getting architects to apply CFD modelling during the design process. Onishi et al. (1996) concluded that CFD modelling is a useful design tool for passive solar systems but cautioned that quantitative validation, computation time, and calculation instability for natural convection were important issues to be pursued.

Summary

The literature review provides insight into a number of CFD modelling issues including prediction accuracy, turbulence modelling, and grid independence. A significant development in the literature is a growing number of reports applying CFD to the design stage of projects. Also significant are recent studies which found that LES results agree better with experimental results for situations examined. These reports indicate that the research community is moving toward the application of LES methods, and that the potential exists for successful application of CFD methods to building design.

Discussion

One of the original motivations for this effort was to determine the potential for applying NIST-LES3D to the study of building ventilation and IAQ problems of current interest. As discussed earlier in this report, this model has certain capabilities that may make it uniquely suitable to the study of some problems. The discussion of NIST-LES3D that follows includes its application to a test case as a demonstration of its capabilities, advantages and disadvantages of the code compared to conventional k-ε CFD models, and a number of topics to which the code could be applied. The development of a strategy for industrial application of CFD is also discussed.

Test case

Simulations of a realistic test case were performed with the program NIST-LES3D both to become familiar with the program and as a way to evaluate the program's appropriateness for the types of problems being considered. The test case was based loosely on an experimental case described by Shaw et al. (1993a) and (1993b). Shaw performed ventilation system performance tests for a partitioned workstation in a test chamber. This problem was selected as a test case because it represents a realistic geometry that might be used for building ventilation and IAQ simulations.

The room simulated was about 100 m^3 (3500 ft^3) and was divided into about 100,000 grid cells. The geometry included airflow blockages due to partitions, a desk, and a table. The case was non-isothermal with two internal heat sources and cool air supplied via ceiling diffusers. Also, species transport was simulated with an initial uniform concentration but no source in the space. The input to NIST-LES3D for this case is an ASCII file of about half of a page of text which was set up in less than an hour. The test case was simulated on an SGI Power Onyx RE2 and required about 8 hours of CPU time to simulate a 30 minute test case. The results indicate that it is feasible to do simulations of transient events of up to a couple hours in an overnight simulation. Simulation results were not directly compared to the reported test case due to a lack of both detailed input and results for the experimental case, however the simulated results were qualitatively acceptable and quantitatively reasonable.

NIST-LES3D advantages and disadvantages

NIST-LES3D is a simulation tool that could be applied to any of the research topics discussed in the literature review or mentioned during the building industry discussions. However, the program has a few unique aspects compared to a general CFD code that lead to some advantages and disadvantages.

NIST-LES3D has two main potential advantages over conventional CFD codes. The first obvious one is the potential for more accurate simulation of flow details and their impact on bulk fluid properties (i.e. temperature and contaminant concentration) due to the use of the LES method instead of the common k-ε empirical turbulence model. This advantage may be largest for flow situations where the k-ε model has been reported to have difficulties such as natural or mixed convection, ventilation supply flow and near-wall flow.

The second advantage is that NIST-LES3D simulations provide a true transient solution to the Navier-Stokes equations. This is dramatically different than the typical CFD solution which is a steady state approximation, and where a 'transient' solution involves repeatedly achieving a steady-state solution as the program steps through time. Since all turbulent airflow situations are truly transient problems, NIST-LES3D could have a tremendous advantage in the study of problems where transient effects are of particular interest.

The main limitation of NIST-LES3D is the restriction to regular geometries. This restriction and the lack of graphical input capability (such as through a CAD interface) would make the application of NIST-LES3D to certain problems with complicated geometry difficult.

Potential research topics

Based on the unique capabilities of NIST-LES3D, there are a number of topics in the field of building ventilation and IAQ that could be studied. As part of the current effort, a number of these topics were considered based on the results of the literature review, discussions with a number of individuals in the building industry, and consideration of current research issues in ventilation and IAQ.

The literature review has already been discussed, and several topics were identified for the application of NIST-LES3D. In addition, a number of topics were identified through discussions with building industry contacts. These include distribution of ventilation air in a room, CO detector location, backdrafting of vented appliances, validation of NIST-LES3D, comparison of accuracy to other CFD codes, time-dependent thermal comfort determination, pulsing of VAV systems, modification of VAV systems to improve ventilation effectiveness, impact of solar loading or other changes in thermal boundaries during the day on contaminant transport, using CFD for ventilation supply and outlet location during building design, and VOC diffusion from point sources under different ventilation conditions.

Several of these topics (e.g., modelling of ventilation distribution and ventilation effectiveness) would be straightforward applications of NIST-LES3D and could take advantage of its unique aspects by focusing on transient issues such as impacts of varying thermal loads or cycling VAV boxes on ventilation effectiveness. Other topics would require some model development. An example of such a topic is modelling of VOC emissions and adsorption/desorption processes. Models of the adsorption/desorption process would need to be developed and incorporated in the NIST-LES3D program. VOCs are a contaminant of great current interest in IAQ research and such a study could provide insight into the processes involved in contributing to elevated concentrations in working and living environments.

An important first step before pursuing the application of NIST-LES3D is to examine the accuracy of its prediction capabilities to room airflow situations. Although there is reason to believe that simulations using NIST-LES3D will be more accurate than with a CFD program employing a k-ε turbulence model as discussed above, this model was developed for and has only been used for studying the transport of smoke and hot gasses in fire situations. As stated previously, the test case simulations were not performed to determine the accuracy of the model but merely provided familiarization with the program and indicated the potential as a tool for studying IAQ and ventilation problems. An effort needs to be made to compare NIST-LES3D predictions to data measured or simulated by other CFD programs. Such data may be reported in the literature or may be available through other research contacts.

Application of CFD modelling to design

The literature review and industry feedback revealed a lack of application of CFD to building ventilation and IAQ problems in the U.S. compared to Europe and Asia. For example, in the last three RoomVent conferences (a leading biannual international conference on room airflow research with a growing emphasis on CFD modeling), only 18 of 326 papers were written by individuals associated with U.S. organizations. This disparity seems even greater with respect to application of these tools at the design level. While the international papers have an increasing emphasis on application to design problems, there is little evidence of any application of CFD to building ventilation design in the U.S. It is possible that U.S. industry may be at a competitive disadvantage in the future due to a void of knowledge in this area.

Another effort of potential interest associated with NIST-LES3D would be to work with the building design community to develop a strategy to apply CFD modeling to the building design process. This strategy could take the form of a building ventilation design equivalent to the National Fire Simulation Facility (NFSF) which is being developed in the Building and Fire Research Laboratory (BFRL) at NIST. The NFSF is being developed to address a similar capability gap in the area of fire scenario simulation. This facility will consist of professional staff, computer and graphics hardware, and software (including NIST-LES3D) for use by NIST staff and closely associated industrial partners. It is expected that U.S. industry will use the unique facility to gain insight needed for the design and development of new products that will be internationally competitive. It is intended to be used by partners from U.S. industry selected through the normal CRADA process who would send personnel to BFRL for the project duration. This model is only one possible plan that may be developed for the building ventilation industry and exploration of the possibilities with industry contacts may identify a better strategy.

Summary

Based on an extensive literature review, the application of the LES model to a test case, and discussions with industry contacts, it was determined that this model offers unique capabilities compared to other available CFD models and could be used to make a significant contribution in studying issues of current interest in the IAQ and ventilation field. Specific recommendations to be pursued include evaluation of prediction accuracy, exploration of topics that take advantage of transient simulation capability, and development of a strategy for U.S. industry to apply CFD in the design process.

References

Aihara M, Kurabuchi T and Tanigawa M. "Depth average version of the k-e model applied to air distribution and supply flow rate of plenum chamber of raised flow air-conditioning systems" (1996) Indoor Air '96 Vol. 2.

Alamdari F, Bennett KM and Rose PM. "Airflow and temperature distribution within an open-plan building space using a displacement ventilation system" (1994) Roomvent '94.

Alamdari F, Edwards SC and Hammond SP. "Microclimate performance of an open atrium office building: a case study in thermo-fluid modelling" (1991) Proceedings of *Computational Fluid Dynamics - Tool or Toy?* Institution of Mechanical Engineers.

Anderson DA, Tannehill JC, and Pletcher RH. *Computational Fluid Mechanics and Heat Transfer*. (1984) Hemisphere Publishing Company, New York.

ASHRAE. *1993 Handbook of Fundamentals* (1993) American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.

Awbi HB. "Numerical assessment of room air distribution strategies" (1993) Proceedings of the 14th AIVC Conference, Air Infiltration and Ventilation Centre.

Awbi HB. "A CFD study of the air quality at the breathing zone" (1996) Indoor Air Vol. 2.

Awbi HB and Baizhan L. "Predicting the thermal and airflow performance of large spaces " (1994) Roomvent '94.

Awbi HB, Croome DJ and Gan G. "Prediction of airflow and thermal comfort in offices" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Awbi HB and Gan G. "Computational fluid dynamics in ventilation" (1991) Proceedings of Institution of Mechanical Engineers Conference.

Awbi HB and Gan G. "Simulation of solar-induced ventilation" (1992) Proceedings of 2nd World Renewable Energy Congress.

Awbi HB and Gan G. "Evaluation of the overall performance of room distribution" (1993) Proceedings of Indoor Air 93, Vol. 5.

Baker AJ, Williams PT, and Kelso RM. "Numerical calculation of room air motion - Part 1: Math, Physics, and CFD Modeling" (1994) ASHRAE Transactions Vol. 100.1.

Barozzi GS, Imbabi MS, Nobile E, and Sousa ACM. "Scale models and CFD for the analysis of air flow in passively ventilated buildings" (1991) Building Simulation 91.

Baskaran A. “A numerical model to evaluate the performance of pressure equalized rainscreen walls” (1994) Building and Environment 29:159-171.

Baum HR, McGrattan KB, and Rehm RG. ”Large Eddy Simulations of Smoke Movement in Three Dimensions” (1996) Interflam `96, Proceedings of the 7th International Fire Science and Engineering Conference.

Beghein C, Allard F and Limam K. “Numerical study of the influence of inlet velocity and thermal and solutal diffusivities on air flow pattern in a ventilated enclosure” (1994) Roomvent ‘94.

Bennetsen J, Sorensen JN, Sogaard HT, and Christiansen PL. “Numerical simulation of turbulent airflow in a livestock building” (1996) Roomvent ‘96 Vol. 2.

Bergstrom DJ. “Numerical prediction of a turbulent fountain in a room” (1994) ASHRAE Transactions Vol. 100.1

Borchiellini R, Fracastoro GV and Perino M. “Analysis of IAQ in a university auditorium” (1994) Roomvent ‘94.

Borresen BA and Mansen N. “Climatization of indoor speed skating rink” (1992) Roomvent ‘92.

Borth J and Suter P. “Influence of mesh refinement on the numerical prediction of turbulent air flow in rooms” (1994) Roomvent ‘94.

Brohus H and Nielsen PV. “CFD models of persons evaluated by full-scale wind channel experiments” (1996) Roomvent ‘96 Vol. 2.

Buchmann P, Riberon J, Millet JR and Lauriat G. “Numerical predictions of airflow patterns in large enclosures with supplied air jet system” (1994) Roomvent ‘94.

Byrne MA, Goddard AJH, Lockwood FC, and Nasrullah M. “Particulate deposition on indoor surfaces - its role, with ventilation, in indoor air quality prediction” (1995) 16th AIVC Conference, Vol. 2.

Cafaro E, Cardinale N, Fracastoro GV, Nino E, and di Tommaso RM. “Simulation of gas leaks in ventilated rooms” (1992) 13th AIVC Conference.

Cardinale N, Di Tommaso RM, Fracastoro GV, Nino E, and Perino M. “Theoretical and experimental simulation of exhaust hoods” (1993) Proceedings of the 14th AIVC Conference.

Chaturvedi SK and Mohieldin TO. “A CFD analysis of effects of vent location on pollutant concentration in rooms” (1989) Building Systems: Room Air and Air Contaminant Distribution, ASHRAE.

Chen Q and Chao N-T. "Prediction of buoyant plume and displacement ventilation with different turbulence models" (1996) Indoor Air Vol. 1.

Chen Q and Jiang Z. "Evaluation of air supply method in a classroom with a low ventilation rate" (1992) Proceedings of the Jacques Cartier Conference.

Chen Q and Jiang Z. "Simulation of a complex air diffuser with CFD technique" (1996) Roomvent '96 Vol. 1.

Chen Q, Peng X and van Paassen AHC. "Prediction of room thermal response by CFD technique with conjugate heat transfer and radiation models" (1995) ASHRAE Transactions Vol. 101.2.

Chen Q, van der Kooi J, and Meyers A. "Measurements and computations of ventilation efficiency and temperature efficiency in a ventilated room" (1988) Energy and Buildings Vol. 12:85-99.

Cheong KW, Adam N, Riffat SB and Shao L. "Migration and deposition of aerosol particles in buildings" (1995) Proceedings of IAQ, Ventilation and Energy Conservation in Buildings, Vol. 1. Centre for Building Studies, Concordia University.

Chikamoto T, Murakami S and Kato S. "Numerical simulation of velocity and temperature fields within atrium based on modified k-ε model incorporating damping effect due to thermal stratification" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Chow WK. "Assessment of the fire protection and ventilation systems in an enclosed car park" (1996) Indoor Air '96 Vol. 2.

Chow WK and Fung WY. "Indoor aerodynamics and ventilation design in large enclosed spaces" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Christensen KS. "Numerical prediction of airflow in a room with ceiling-mounted obstacles" (1992) Roomvent '92.

Chung KC and Lee CY. "Predicting air flow and thermal comfort in an indoor environment under different air diffusion models" (1996) Building and Environment Vol. 31:21-40.

Clancy EM, Scholzen F and Howarth A. "Comparison of measured and calculated environmental conditions for a naturally ventilated auditorium" (1996) Roomvent '96 Vol. 3.

Clarke JA, Dempster WM, and Negrao C. "The implementation of a computational fluid dynamics algorithm within the ESP-r system" (1995) Building Simulation '95.

Collimeau S, Serieys J-C, Fontaine J-R, and Aubertin G. "Experimental and numerical simulation of ventilation in a painting area" (1996) Roomvent '96 Vol. 2.

Cooper A. “Using CFD techniques to evaluate wind pressure distribution for air infiltration analysis” (1993) Air Infiltration Review Vol. 14, No. 2.

Cox CWJ and Elkhuizen PA. “Displacement ventilation: calculated versus measured data” (1993) CLIMA 2000.

Davidson L and Nielsen PV. “Large eddy simulation of the flow in a three-dimensional ventilated room” (1996) Roomvent ‘96. Vol. 2.

Deardorff JW. “A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers” (1970) J. Fluid Mech., Vol. 41.

Depecker P, Rusaouen G and Inard C. “Study and comparison of two types of air flow in operating rooms using a CFD code” (1996) Indoor Air ‘96 Vol. 1.

Depecker P, Virgone J, and Rusaouen G. “Numerical modelling of air flows in building and design of a data base of experiments” (1995) Building Simulation ‘95.

Drangsholt F. “Air flow patterns and pollutant transmission in auditoria - measurements and CFD-simulations” (1993) Indoor Air 93 Vol. 5.

Emvin P and Davidson L. “A numerical comparison of three inlet approximations of the diffuser in case E1 Annex20” (1996) Roomvent ‘96 Vol. 1.

Fang JB and Grot RA. "Numerical simulation of the performance of building ventilation systems" (1990) ASHRAE Transactions Vol. 96.

Fang JB and Persily AK. "Numerical prediction of airflow patterns and ventilation effectiveness in an open office environment" (unpublished).

Fang JB and Persily AK. "Numerical prediction of ventilation system performance in an open office space" (1991) 12th AIVC Conference.

Fischer V and Rosler M. “Investigation and application of coupled building and room air flow simulation” (1996) Roomvent ‘96 Vol. 2.

Fontaine JR and Rapp R. “Design of air supply systems of workshops equipped with pollutant removal devices - a CFD approach” (1994) Roomvent ‘94.

Fontaine JR and Rapp R. “The design of ventilation systems of large enclosures with unconfined pollutant sources” (1996) Roomvent ‘96 Vol. 3.

Fontaine JR, Rapp R, Serieys JC and Aubertin G. “Aerosol transport in room turbulent air flows experimental” (1994) Roomvent ‘94 Vol. 2.

Frydenlund F, Mathisen HM, Tjelflaat PO and Schild P. "Indoor climate CFD-simulation of an underground icehockey stadium: strategy and results" (1996) Roomvent '96 Vol. 2.

Fukuyo K, Shimoda Y, Mizuno M, Onishi J and Kaga A. "Prediction of airflow in subway stations" (1996) Indoor Air '96 Vol. 4.

Gan G. "Numerical assessment of thermal comfort and air quality in an office with displacement ventilation" (1994) 15th AIVC Conference.

Gan G, Awbi HB, and Croome DJ. "Airflow and thermal comfort in naturally ventilated classrooms" (1991) 12th AIVC Conference.

Grundmann R, Richter E, Vogel P, and Rosler M. "Measurements and computations in a gymnasium-type large enclosure" (1994) Roomvent '94.

Goddard AJH, Byrne MA, Lockwood FC, Nasrullah M and Lai CK. "Aerosol deposition on internal building surfaces: experimental validation of a computational model" (1996) Roomvent '96 Vol. 2.

Guntermann K. "Experimental and numerical study on natural ventilation of atrium buildings" (1994) Roomvent '94.

Guthrie A, Ikezawa H, Otaka K, and Yau RMH. "Airflow studies in larger spaces: a case study of an airport passenger terminal building" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Haghighat F, Jiang Z, and Wang J. "A CFD analysis of ventilation effectiveness in a partitioned room" (1991) Indoor Air 1:606-615.

Haghighat F, Jiang Z, and Wang J. "The impact of ventilation rate and partition layout on the VOC emission rate: time-dependent contaminant removal" (1994) Indoor Air 4:276-283.

Han H. "Calculation of ventilation effectiveness using steady-state concentration distributions and turbulent airflow patterns in a half scale office building" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Havet M, Blay D, and Veyrat O. "Numerical study of coupled convective, radiative and conductive heat transfer in a large enclosure application to radiative heating" (1994) Roomvent '94.

He P, Katayama T, Hayashi T, Tsutsumi J, Tanimoto J, and Hosooka I. "Examination of the effects of wall thickness and bay windows on cross-ventilation by CFD" (1996) Roomvent '96.

Heikkinen J and Piira K. "CFD computation of jets from circular ceiling diffusers" (1994) Roomvent '94.

Ho FCM and Goodfellow HD. "The application of computational fluid dynamics to predict contaminant concentration in a polymer manufacturing facility" (1994) 4th International Symposium on Ventilation for Contaminant Control.

Horstman RH. "Predicting velocity and contamination distribution in ventilated volumes using Navier-Stokes equations" (1988) Proceedings of ASHRAE IAQ 88.

Huo Y, Zhang J, Shaw C and Haghighat F. "A new method to describe the diffuser boundary conditions in CFD simulation" (1996) Roomvent '96 Vol. 2.

Imano H, Kurabuchi T, Kamata M and Hayama H. "A long term transient analysis of thermal flow field of a telecommunications equipment room in case of air-cooling system breakdown" (1996) Roomvent '96 Vol. 3.

Iwamoto S. "A study on numerical prediction method of thermal environment around occupants" (1996) Indoor Air '96 Vol. 1.

Iwamoto S, Ishii A, Katayama T, and Tsutsumi J. "Numerical prediction of indoor airflow by cross-ventilation" (1992) Proc. of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Jacobsen TV and Nielsen PV. "Numerical modelling of thermal environment in a displacement-ventilated room" (1993) Indoor Air 93, Vol. 5.

Jiang Z and Haghighat F. "Ventilation effectiveness in a partitioned office with displacement ventilation determined by computer simulation" (1992) Indoor Environment 2:365-373.

Jones PJ, Alexander DK, and Powell G. "The simulation of infiltration rates and air movement in a naturally ventilated industrial building" (1991) 12th AIVC Conference, Vol. 1.

Jones P and Waters R. "The practical application of indoor airflow modeling" (1993) Modelling of Indoor Air Quality and Exposure, ASTM STP 1205, American Society for Testing and Materials.

Joubert P, Sandu A, Beghein C, and Allard F. "Numerical study of the influence of inlet boundary conditions on the air movement in a ventilated enclosure" (1996) Roomvent '96.

Kalzuka M, Iwamoto S, Ishii A, and Sakai K. "A numerical prediction of a thermal environment in a room heated with floor panels" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Kato S, Murakami S, and Kobayashi H. "New scales for evaluating ventilation efficiency as affected by supply and exhaust openings based on spatial distribution of contaminant" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Kato S, Murakami S, Shoya S, Hanyu F, and Zeng J. "CFD analysis of flow and temperature fields in atrium with ceiling height of 130 m" (1995) ASHRAE Transactions Vol. 101.2.

Kato S, Murakami S and Zeng J. "Numerical analysis of contaminant distribution around a human body" (1996) Roomvent '96 Vol. 2.

Kirkpatrick AT and Knappmiller KD. "The ADPI of cold air jets in an enclosure" (1996) ASHRAE Transactions Vol. 102.1.

Knappmiller KD and Kirkpatrick AT. "A numerical study comparing ADPI and ventilation effectiveness for conventional and cold-air distribution systems" (1995) ASHRAE IAQ 95.

Kohal JS, Riffat SB and Shao L. "An experimental and theoretical investigation of airflow through large horizontal openings" (1994) 15th AIVC Conference.

Kolokotroni M and Littler J. "Airborne moisture and its effects on condensation risks in dwellings" (1993) CLIMA 2000.

Kondo Y, Aoki S, Yuzawa H, and Hayashi K. "Numerical simulation on distributions of air temperature and humidity in large enclosure" (1996) Indoor Air '96 Vol. 3.

Kondo Y and Niwa H. "Numerical study of an atrium by means of a macroscopic model and k-ε turbulence model" (1992) Proc. of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Kornaat W and Lemaire AD. "Natural ventilation of parking garages ; dimensioning of ventilation units with the assistance of air flow models" (1994) Air Infiltration Review Vol. 15, No. 2.

Krafthefer B and Shah DJ. "Implications of room air motion on control of thermal comfort in rooms with natural convection heat sources" (1995) Building Simulation '95.

Kurabuchi T, Fang JB and Grot RA. *A Numerical Method for Calculating Indoor Airflows Using a Turbulence Model* (1990) NISTIR 89-4211, National Institute of Standards and Technology.

Kurabuchi T and Kusuda T. "Numerical prediction for indoor air movement" (December 1987) ASHRAE Journal.

Lam JK-W, Ruddick KG, and Whittle GE. "Air curtains for infiltration control - a computational fluid dynamics analysis" (1990) 11th AIVC Conference, Vol. 1.

Lam JC, Yuen RKK, and Lau TM. “Improvements to User-friendliness of a Computational Fluid Dynamics (CFD) Code for Simulation of Air Movement in Buildings” (1993) Building Simulation 93.

Lee WC and Li F. “An investigation of air flow in buildings using FLOW3D” (1994) *Proceedings of Building Environmental Performance - Facing the Future*, Building Research Establishment.

Lefeuvre M, Groleau D, and Marenne C. “Airflow pattern and temperature field simulation in a large glazed space” (1995) Building Simulation ‘95.

Lemaire AD. “A numerical study of the air movement and temperatures in large atria and sunspaces” (1990) Roomvent 90.

Li K and Teh SL. “Two-dimensional study of airflow through large openings” (1996) Indoor Air ‘96 Vol. 2.

Li Y. *Simulation of Flow and Heat Transfer in Ventilated Rooms.* (1992) Royal Institute of Technology; Stockholm, Sweden.

Li Y. “A simple assembly approach for representing complex geometry on non-body-fitted Cartesian grids” (1994) Roomvent ‘94.

Li Y, Moller S and Symons J. “Effects of outdoor thermal environment in displacement ventilation - Part 1 flow and temperature fields” (1996a) Indoor Air Vol. 1.

Li Y, Moller S and Symons J. “Effects of outdoor thermal environment in displacement ventilation - Part 2 heat transfer analysis” (1996b) Indoor Air Vol. 1.

Li Y, Symons J and Holmberg S. “Contribution ratio and sojourn times of air in a ventilation flow system with multiple openings” (1996c) Roomvent ‘96 Vol. 3.

Li Y, Vidakovic S and Symons J. “Some numerical observations of colliding free convection boundary layers” (1996d) Roomvent ‘96 Vol. 1.

Lu W and Howarth AT. “Numerical analysis of indoor aerosol particle deposition and distribution in two-zone ventilation system” (1996) Building and Environment Vol. 31:41-50.

Manzoni D, Vialle P and Blay D. “Convective phenomena involved in a displacement ventilation system” (1996) Indoor Air ‘96 Vol. 1.

Mathisen HM. "Case studies of displacement ventilation in public halls" (1989) ASHRAE Transactions Vol. 95.

Matsunawa K, Iizuka H and Tanabe S. "Development and application of an underfloor air-conditioning system with improved outlets for a 'smart' building in Tokyo" (1995) ASHRAE Transactions Vol. 101.2.

McGrattan K, Rehm RG and Baum HR. "Fire-driven flows in enclosures" (1994) Journal of Computational Physics 110:285-291.

Mizuno T and Warfield MJ. "Development of three-dimensional thermal airflow analysis computer program and verification test" (1992) ASHRAE Transactions Vol. 98.2.

Mizutani K, Okazaki T, Nakamura S and Takizawa T. "Model experiment and numerical simulation of air change efficiency in a room using different air conditioning configurations" (1996) Roomvent '96 Vol. 3.

Moser A. "The Message of Annex 20: Air Flow Patterns within Buildings" (1991) 12th AIVC Conference.

Moser A, Off F, Schalin A, Yuan X. "Numerical modelling of heat transfer by radiation and convection in an atrium with thermal inertia" (1995) ASHRAE Transactions Vol. 101.2.

Moshfegh B and Sandberg M. "Influence from surface area of radiant cooling ceiling panel on air movement" (1996) Roomvent '96 Vol. 3.

Murakami S. "Prediction, analysis and design for indoor climate in large enclosures" (1992) Roomvent '92.

Murakami S, Kato S, and Ooka R. "Comparison of numerical predictions of horizontal nonisothermal jet in a room with three turbulence models - k-ε ESM, ASM, and DSM" (1994) ASHRAE Transactions Vol. 100.2.

Murakami S, Kato S, and Suyama Y. "Numerical study on diffusion field as affected by arrangement of supply and exhaust openings in conventional flow type clean room" (1989) ASHRAE Transactions Vol. 95.2.

Murakami S, Mochida A, Ooka R, Kato S, and Iizuka S. "Numerical prediction of flow around a building with various turbulence models: comparison of k-epsilon EVM, ASM, DSM, and LES with wind tunnel tests" (1996) ASHRAE Transactions Vol. 102.1.

Nagano S and Mimi T. "Ventilation efficiency in a two-dimensional enclosure with a supply outlet in the ceiling or in the floor" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Nakamura Y, Mizuno M, Sekimoto Y, Akagi K, Kunimatu Y, Otaka K, and Kohyama M. "Study on thermal comfort and energy conservation of task-ambient air conditioning system" (1996) Roomvent '96 Vol. 1.

Nho HG and Kim WT. "Numerical study on the air-flows system with heat sources in an indoor telecommunication room" (1996) Indoor Air '96 Vol. 2.

Nielsen JR, Nielsen PV and Svidt K. "Obstacles in the occupied zone of a room with mixing ventilation" (1996) Roomvent '96 Vol. 3.

Nielsen PV. "Airflow in a world exposition pavilion studied by scale-model experiments and computational fluid dynamics" (1995) ASHRAE Transactions Vol. 101.2.

Niu J and Kooi Jvd. "Numerical investigation of thermal comfort and indoor contaminant distributions in a room with cooled ceiling systems" (1993) Indoor Air 93, Vol. 5.

Off F, Moser A and Suter P. "Transient numerical modelling of heat transfer by radiation and convection in atrium with thermal inertia" (1996) Roomvent '96 Vol. 3.

Off F, Schaelin A and Moser A. "Numerical simulation of air flow and temperature in large enclosures with surface radiation exchange"(1994) Roomvent '94.

Ohira N and Omori T. "Ventilation behavior in a void space furnished with gas water-heaters" (1996) Indoor Air '96 Vol. 1.

Oliveeira LA, Penot F, and Costa JJ. "Aerodynamic sealing through a double jet curtain: a parametric numerical study" (1996) Roomvent '96 Vol. 1.

Ong I. "Modelling wind flow over a factory with CFD" (1989) Building Systems: Room Air and Air Contaminant Distribution, ASHRAE.

Onishi J, Shirahama S, and Mizuno M. "Study on CFD simulation of airflows and thermal environment in a passive solar room with a Trombe wall" (1996) Roomvent '96 Vol. 1.

Ozeki Y, Higuchi S, Saito T, Ohgaki S and Sonda Y. "Simulation of temperature and flow field in an atrium Part 1: Computation of solar radiation, radiative heat transfer, airflow and temperature" (1992) Proc. of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Ozeki Y, Kato S and Murakami S. "Numerical analysis on flow and temperature fields in experimental real scale atrium" (1996) Roomvent '96 Vol. 3.

Plett EG, Soultogiannis AA, and Jouini DB. "Numerical simulation of ventilation air movement in partitioned offices" (1993) Indoor Air 3:26-33.

Regard M, Carrie FR, Voeltzel A, and Richalet V. "Measurement and CFD modelling of IAQ indices" (1995) 16th AIVC Conference.

Renz U and Vogl N. "Numerical predictions of air flow patterns in large enclosures" (1996) Roomvent '96 Vol. 3.

Riffat SB and Shao L. "Investigation of effect of tracer species on tracer mixing using CFD" (1994) 15th AIVC Conference.

Rota R, Canu P, Carra S and Nano G. "Ventilated enclosures with obstacles: experiments and CFD simulations" (1994) 4th International Symposium on Ventilation for Contaminant Control.

Roy S, Baker AJ, and Kelso RM. "Airborne contaminant CFD modelling studies for two practical 3-D room air flow fields" (1993) Proceedings of Indoor Air 93, Vol. 5.

Sakamoto Y and Matsuo Y. "Numerical predictions of three-dimensional flow in a ventilated room using turbulence models" (February 1980) Appl. Math. Modelling, Vol. 4.

Satwiko P and Donn M. "With what confidence can an architect use a CFD code package as a building environmental design tool? Case study of the search for an Indonesian roof chimney" (1995) Building Simulation '95.

Schaelin A, Dorer V, van der Maas J, and Moser A. "Application of a new method for improved multizone model predictions" (1993) 14th AIVC Conference.

Schaelin A and Kofoed P. "Numerical simulation of thermal plumes in rooms" (1994) Roomvent '94.

Schaelin A, van der Maas J, and Moser A. "Simulation of airflow through large openings in buildings" (1992) ASHRAE Transactions Vol. 98.2

Schaub EG, Baker AJ, Burk ND, Gordon EB, and Carswell PG. "On development of a CFD platform for prediction of commercial kitchen ventilation flow fields" (1995) ASHRAE Transactions Vol. 101.2.

Schild PG, Tjelflaat PO and Aiulfi D. "Guidelines for CFD modelling of atria" (1995) ASHRAE Transactions Vol 101.2.

Schild PG. "CFD analysis of an atrium, using a conjugate heat transfer model incorporating long-wave and solar radiation" (1996) Roomvent '96 Vol. 2.

Selvam RP. "Numerical simulation of flow and pressure around a building" (1996) ASHRAE Transactions Vol. 102.1.

Shao L and Riffat SB. "CFD for improvement of k-factor accuracy in HVAC systems" (1994) Proceedings of the 4th International Symposium on Ventilation for Contaminant Control.

Shao L, Walker RR, and Woolliscroft M. "Natural ventilation via courtyards: the application of CFD" (1993) 14th AIVC Conference.

Sharples S and Palmer RG. "Modelling fluctuating air flows through building cracks" (1994) 15th AIVC Conference, Vol. 1.

Shaw CY, Zhang JS, Said MN, Vaculik F, and Magee RJ. "Effect of air diffuser layout on the ventilation conditions of a workstation - Part I: air distribution patterns" (1993a) ASHRAE Transactions.

Shaw CY, Zhang JS, Said MN, Vaculik F, and Magee RJ. "Effect of air diffuser layout on the ventilation conditions of a workstation - Part II: air change efficiency and ventilation efficiency" (1993b) ASHRAE Transactions.

Skovgaard M and Nielsen PV. "Modelling complex inlet geometries in CFD - applied to air flow in ventilated rooms" (1991) 12th AIVC Conference Vol. 3.

Simons MW, Waters JR and Leppard J. "The combined use of CFD and zonal modelling techniques to aid the prediction and measurement of ventilation effectiveness parameters" (1995) 16th AIVC Conference.

Smagorinsky J, Manabe S, and Holloway JL. "Numerical results from nine-level general circulation model of the atmosphere" (1965) Mon. Weath. Rev., Vol. 93.

Smith MG, Walker RR, and Perera MDAES. "Prediction of natural ventilation air flows in a non-urban office" (1992) Roomvent '92.

Sonda Y, Higuchi S, Saito T, Ohgaki S and Ozeki Y. "Simulation of temperature and flow field in an atrium Part 2: Comparison and results from experiments and numerical analysis, and applications" (1992) Proc. of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Stankovic S and Setrakian A. "Thermal and CFD modelling vs. wind tunnel in natural ventilation studies" (1993) Building Simulation 93.

Stathopoulos T and Zhou YS. "Evaluation of wind pressures on flat roofs" (1995) Building and Environment Vol. 30.

Svidt K. "Investigation of inlet boundary conditions for numerical prediction of air flow in livestock buildings" (1994) Roomvent '94 Vol. 2.

Svidt K, Heiselberg P, and Hendriksen OJ. "Natural ventilation in atria - a case study" (1996) Roomvent '96 Vol. 3.

Suyama Y and Aoyama M. "Numerical simulation of indoor air environment in an office building" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Takahashi N, Iwata E, and Funatsu M. "Simulation and evaluation of the environment of a large-scale atrium" (1992) Proc. of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Takemasa Y, Kurabuchi T, and Kamata M. "Numerical simulation of indoor temperature and wall heat flow distribution of a heated and cooled room" (1992) Proceedings of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Tang Y-Q and Holmberg S. "Efficient 'horizontal flow' ventilation: influence of supply inlet designs" (1993) 14th AIVC Conference, Air Infiltration and Ventilation Centre.

Teng J, Chiang C-M, Chao N-T and Wu J-S. "The design of window-transom locations to improve the indoor air quality of bedrooms by outdoor air in the winter - computational fluid dynamics aided architectural design" (1996) Indoor Air '96 Vol. 3.

Tennekes H and Lumley JL. *A first course in turbulence* (1972) MIT press.

Tjelflaat PO and Bell D. "Work environment CFD-simulation of a fiberglass factory" (1996) Roomvent '96.

Tjelflaat PO and Knott R. "A simulation model for thermal comfort of a person in a large enclosure" (1996) Indoor Air '96 Vol. 2.

Tsutsumi J, Katayama T, Hayashi T, and He P. "Numerical simulation of cross-ventilation in a single-unit house" (1992) Proc. of Room Air Convection and Ventilation Effectiveness, ASHRAE.

Van der Maas J and Schaelin A. "Application of air flow models to aircraft hangars with very large openings" (1995) 16th AIVC Conference.

Vanderheyden MD and Schuyler GD. "Evaluation and quantification of the impact of cooling tower emissions on indoor air quality" (1994) ASHRAE Transactions Vol. 100.2.

Vazquez B, Samano D and Yianneskis M. "The effect of air inlet location on the ventilation of an auditorium" (1991) *Computational fluid dynamics - tool or toy?* Institution of Mechanical Engineers.

Vitale S, Alabiso M, and Castellano L. "Development of a numerical submodel for simulating moisture distribution inside indoor spaces by a multidimensional IAQ computer code" (1996) Indoor Air Vol. 3.

Vogel P, Richter E and Rosler M. "The effect of various inlet conditions on the flow pattern in ventilated rooms - measurements and computations" (1993) Proceedings of the 14th AIVC Conference, Air Infiltration and Ventilation Centre.

Walker R, Shao L, and Woolliscroft M. "Natural Ventilation via Courtyards: Theory & Measurements" (1993) 14th AIVC Conference, Air Infiltration and Ventilation Centre.

Walton GN. *CONTAM93 - User Manual* (1994) NISTIR 5385.

Wang Z. "Comparison of airflow within six apartments: cross-ventilation study with Computational Fluid Dynamics" (1996) Roomvent '96.

Wang Z. "Study of cross-ventilation in apartment with large openings - comparison between wind tunnel tests and simulations" (1995) Proceedings of 2nd Indoor Air Quality, Ventilation and Energy Conservation in Buildings Conference, Vol. 1.

Ward IC and Wang F. "Air movement studies in a large parish church building" (1994) 15th AIVC Conference.

Whittle G and Lam J. "A fluid situation" (August 1992) Building Services.

Williams PT and Baker AJ. "CFD characterization of natural convection in a two-cell enclosure with a 'door'" (1994) ASHRAE Transactions Vol. 100.2.

Williams PT, Baker AJ, and Kelso RM. "Numerical calculation of room air motion - Part 2: The Continuity Constraint Finite Element Method for Three-dimensional Incompressible Thermal Flows" (1994a) ASHRAE Transactions Vol. 100.1.

Williams PT, Baker AJ, and Kelso RM. "Numerical calculation of room air motion - Part 3: Three-dimensional CFD simulation of a full-scale room air experiment" (1994b) ASHRAE Transactions Vol. 100.1

Xu J, Liang H and Kuehn TH. "Comparison of numerical predictions and experimental measurements of ventilation in a room" (1994) Roomvent '94 Vol. 2.

Xu WQ, Sun H, Sun QQ, Ng KC, Kong HC, Koh CN and Lim W. "Investigation of air flow pattern of the ventilation system in the repair process of VLCC" (1996) Indoor Air Vol. 2.

Xuejun S, Zhijiu C and Xongcai Q. "Numerical computation of airflow in air-conditioning automobile compartment" (1996) Roomvent '96 Vol. 2.

Yau RH and Whittle GE. "Air flow analysis for large spaces in an airport terminal building: computational fluid dynamics and reduced-scale physical model tests" (1991) Proceedings of "Computational fluid dynamics - tool or toy?", Institution of Mechanical Engineers.

Yoon J, Kim BS, and Lee K. "A simulated indoor thermal environment of the CHONAN railway station" (1996) Roomvent '96 Vol. 1.

Zhang JS, Christianson LL, Wu GJ, and Zhang RH. "An experimental evaluation of a numerical simulation model for predicting room air motion" (1992) Proceedings of the Jacques Cartier Conference.

Zhou Y and Stathopoulos. "Application of two-layer methods for the evaluation of wind effects on a cubic building" (1996) ASHRAE Transactions Vol. 102.1.