2005 WSEAS Int. Conf. on ENVIRONMENT, ECOSYSTEMS and DEVELOPMENT, Venice, Italy, November 2-4, 2005 (pp272-277)

Developing efficient tools to evaluate indoor environment issues: on-site measurements and numerical simulation of indoor air flow in a test room

M.GR. VRACHOPOULOS, M.K. KOUKOU, N.W. VLACHAKIS,
N.G. ORFANOUDAKIS, G. THANOS, A.E. FILIOS, AND J. KALDELIS

Mechanical Engineering Department, Renewable Laboratory
Faculty of Applied Sciences, Technological Educational Institution of Halkida, 344 00, Psachna,Evia
GREECE
mvrachop@teihal.gr http://www.teihal.gr

Abstract: Advance and efficient building design can be achieved by gathering information about air quality and the behavior of the airflow in the rooms of the building. An accurate understanding of indoor air distribution is crucial to the design of heating, ventilating, and air-conditioning installations that provide thermal comfort and indoor air quality. In this context, a full-scale test room was built with the purpose to conduct experiments under various test conditions to evaluate specific building issues, including different insulation materials and techniques, different heating/cooling/ventilation technologies, etc. The test room is located at the Technological Educational Institution of Halkida in the rural area of Psachna. The dimensions of the test room are 4m x 6m x 4m and its roof is covered with roman tiles and a radiant barrier reflective insulation system. The room is accessed from two doors located in the opposite walls and it is ventilated through these openings. The design and construction of the test room has been completed and the measurement plan has been organized to proceed with the measurements of the indoor air quality quantities in the test room. A three-dimensional steady state numerical model has been developed to describe indoor airflow in the room. Runs have already been conducted to study important issues including grid independence and application of the adequate turbulence model. Comparison of modeling predictions with preliminary experimental measurements shows that there is a sufficient agreement.

Key-Words: indoor air quality, CFD model, experiments

1 Introduction

Nowadays, indoor air quality and thermal comfort have become a crucial topic of concern. Indoor air quality affects human health and productivity in a significant way while proportions of respiratory illnesses and lung cancers may be caused by avoidable indoor pollution. Indoor air pollution may consist of compounds from different sources including environmental pollutants (tobacco smoke, radon), inorganic (CO_2, CO, etc.), organic (volatile organic compounds, VOCs), and biological agents (fungi, mites, etc.) [1]. Airflow within a room affects the emission rate at which contaminants emit into the air from sources within the room.

The options currently available to determine indoor airflow and quality are measurement and mathematical modeling [2-5]. A measurement plan should account for measurement of air velocity, temperature, humidity and pollutant concentrations. Simulation can determine and predict airflow and concentrations for all possible combinations of building and weather conditions. There are two main types of mathematical models: microscopic scale models (Computational Fluid Dynamics, CFD) that calculate the values of all relevant parameters at discrete points in the flow field with a high degree of resolution [5]; and macroscopic models (including multi-zone and zonal models) which assume that large zones of the building contain well mixed air and calculate flows between these zones [2-4]. Over the past decade there has been a substantial body of work completed using both methods to examine various aspects of indoor air flows, air quality and contaminant transport.

In the context of the above, the objectives and steps of this on-going research work are: a) to explore and summarise the options for measurement of indoor air quality parameters with respect to accuracy, sensitivity and applicability, b) to review the current capabilities in the numerical simulation of indoor air flow, c) to develop efficient simulation tools for the evaluation of indoor air quality in a test room developed for the research purposes, d) to perform systematic measurements in the test room according to the existing standards that can be used for comparison with simulations, and as a final stage e) to perform a cost-effective analysis to show the links between improvements in the indoor environment and the potential financial benefits.

An extended literature review has already been carried out to explore existing measurement methodologies and techniques to assess indoor air flow and quality as well as to review the situation in the related modeling area. The design and construction of the test room has been completed and the measurement plan has been organized to proceed with the measurements of the indoor air quality quantities in the test room. Preliminary measurements have been obtained for air velocity and temperature. A three-dimensional steady state numerical model has been developed to describe indoor airflow in the room. Various cases have been considered accounting for isothermal and non-isothermal conditions. Runs have been conducted to study important issues including grid independence and application of the adequate turbulence model. Comparison with preliminary experimental results shows that the model predicts with sufficient accuracy the air flow in the test room.

2 Experimental Part

2.1 Description of the test room

Aiming to the validation of the computational simulation and for the conduction of systematic experimental measurement plan the test room of Figure 1 has been designed and constructed at the premises of the Technological Educational Institution of Halkida located in the agricultural area of Psachna.

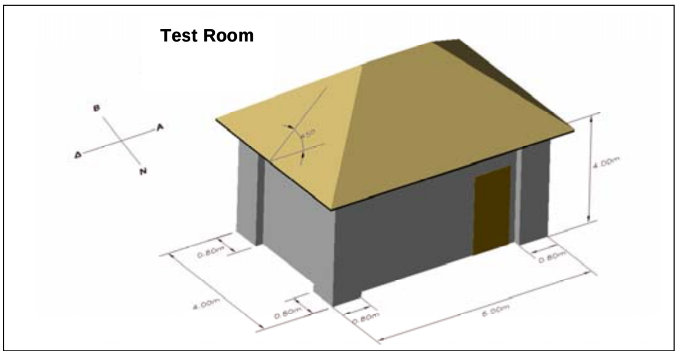

Fig. 1 Outside view of the test room.

The dimensions of the test room are 4m x 6m x 4m and its roof is covered with roman tiles and a radiant barrier reflective insulation system. The side walls are a two series brick construction with a bubble material laminated between layers of aluminum foil placed in the 20mm gap of the brick layers. The side wall total thickness is 210mm including a 20 mm thickness of the wall sheathing in each side. The selection of the above mentioned insulation materials used in residential construction in attics, walls, ceilings, and radiant floor heating applications is due to the requirement for an excellent thermal resistance regarding the shell of the test room. Two doors are located in the north and south-facing walls and the room is ventilated through these openings. A small partition with the height of 1m is located at the north wall.

2.2 Experimental plan

The experimental plan focuses on the measurement of air velocity, temperature and at a later stage pollutants concentration that describe the flow field characteristics and the contaminant level in the test room considered. Detailed measurements of indoor air flow are difficult to accomplish because similar to numerical simulations behavior at the boundaries of rooms (e.g. walls, internal partitions) can have a large effect on the fluid motion. The selection of the sampling method for use in conducting indoor air studies depend on the research objectives, the contaminants of concern and the required sampling duration which should be representative of occupants' exposure time. In the current phase of the implementation of the experimental plan certain air quality measurements that are indicative of common indoor air quality concerns, such as temperature and indoor air flow have been obtained. The air velocity and temperature were measured at several strategic locations in the test room, e.g. close to the inlet, close to the outlet, at the middle of the room and at other locations in the test room as recommended by the ASHRAE standard [6]. Air velocity has been measured with the KIMO VT 200 hot wire anemometer. Hot wire anemometry has been widely used to measure air velocities in buildings. The advantages are that it provides high resolution in time and space, and is able to measure velocity, and turbulence intensity in time and space. The accuracy of the measurements is estimated at ±3% of reading ±0.03m/s.

3 Model Development

3.1 Introduction

An extended literature review of indoor airflow models has been carried out to investigate their advantages and limitations. There are two main methods for simulating indoor airflow and pollutant levels, macro models and micro models. Macro models include multi-zone models and zonal models. In multi-zone models [4] uniform conditions inside a room (zone) are assumed and the airflow is modeled through links or flow paths between zones. The network of links is described by a set of equations that are solved simultaneously to provide a converged solution. In zonal models [3,5] the room interior is divided into a small number of zones or cells and the mass and heat balance equations are applied to them. The solution of the set of coupled equations provides the air flow and temperature distribution in the room. Micro models refer to Computational Fluid Dynamics (CFD) that was deployed for the simulation of indoor airflow after the development of the first code by Nielsen [7]. In this type of models the equations are discretised in order to solve the flow field numerically. Multi-zone models are considered easier, and cheaper to use than CFD models, however, as they assume the air inside a room is uniformly mixed, they cannot be used to predict local air velocities or concentrations variations inside rooms. In this context, in the present work the CFD approach has been adopted.

3.2 The physical problem and its mathematical formulation

3.2.1 The governing equations

For the case of isothermal conditions the simulation of the indoor airflow in the test room of Figure 1 the equations that describe the conservation of mass and momentum in three dimensions are solved. In the non-isothermal case the energy equation is also solved. The steady state conservation equations for the dependent variable φ (for continuity equation φ=1, for momentum equations φ=u,v,w, for energy equation φ=T) may be written in the general form of [8]:

$$\mathrm{div}\left(\rho\vec{u}\phi\right)=\mathrm{div}\left(\Gamma_\phi \mathrm{grad}\phi\right)+S_\phi \quad (1)$$

where ρ, is the mixture density, $\vec{u}$, is the velocity vector, Γ_ϕ, is the effective exchange coefficient and S_ϕ, is source rate per unit volume. These governing flow equations are highly non linear and to obtain their solution it is necessary to use numerical techniques.

Airflow in the test room is normally three-dimensional, recirculating and turbulent. Because the turbulent fluctuations affect the transport of momentum and energy they must be included in the formulation and solution of the Eqs. (1). Turbulence transport models account for the influence of turbulence on the time-mean motion. The job of the turbulence model is to calculate the distribution of the eddy viscosity (μ_t) throughout the flow domain. For the numerical simulation of the turbulent flow inside the test room four turbulence models have been tested: the standard k-ε model, the RNG k-ε model, the realizable k-ε model and the Spalart-Almaras model provided by Fluent®. The standard two-equation k-ε turbulence model involves the solution of two additional partial differential equations for the turbulent kinetic energy (k) and its dissipation rate (ε) [9-10]. The values of the constants C_μ, C_1, C_2, σ_κ and σ_ε applied are 0.09, 1.44, 1.92, 1.0 and 1.3 respectively [9-10]. The RNG k-ε model is essentially a variation of the standard k-ε model, with the used constants estimated rather through a statistical mechanics approach than from experimental data. The values of the constants C_μ, C_1 and C_2 applied are 0.0845, 1.42 and 1.68, respectively [11]. For the realizable model the term ``realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. The realizable k-ε model contains a new formulation for the turbulent viscosity. Also, a new transport equation for the dissipation rate, ε, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation [12].

Traditionally there are two approaches for modeling of the near-wall region. In the first approach, the viscosity-affected inner region is not resolved. Instead a wall function is used to bridge the viscosity-affected region with the fully turbulent region. In the second approach, the turbulence model is modified to enable the viscosity-affected region to be resolved with a mesh all the way to the wall (enhanced wall treatment). In Fluent® there are three different models to choose between for calculating the flow behavior near the wall: i) standard wall function, ii) non-equilibrium wall function and iii) the enhanced wall treatment. Models i) and iii) were tested.

In the isothermal case fluid properties are held constant with values corresponding to a room temperature of about 305.5K. The air density is 1.16kg/m^3 and the viscosity 1.79e-5 kg/(m.s). In the non-isothermal case effects of buoyancy were modeled based on the Boussinesq-approximation. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation. The model is valid as the temperature differences in the domain are not large.

3.2.2 Grid development

The applied coordinate system and the respective grid used in the simulations is shown in Figures 2a,b. A quite large number of grid sizes have been tested and it has been concluded that a grid independent solution is possible when the computational domain is discretised with approximately 86925 tetrahedral grid cells (Figures 2a,b and 3).

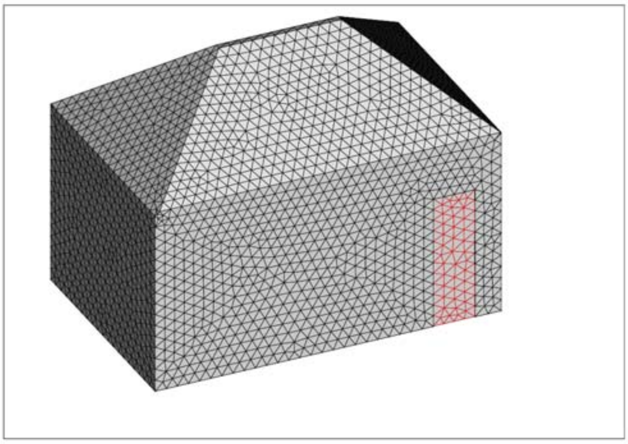

Fig. 2a View of the grid employed

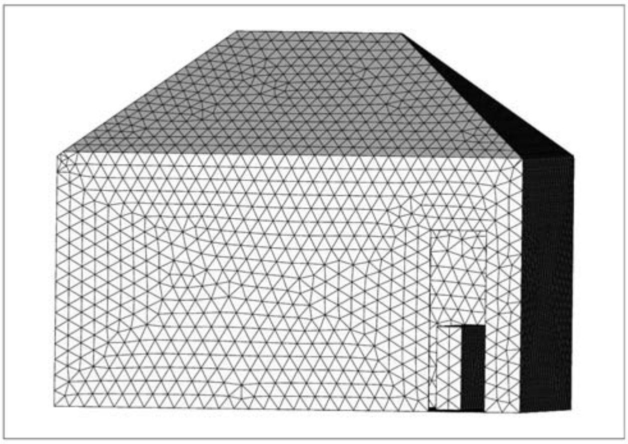

Fig. 2b View of the internal wall in the grid employed

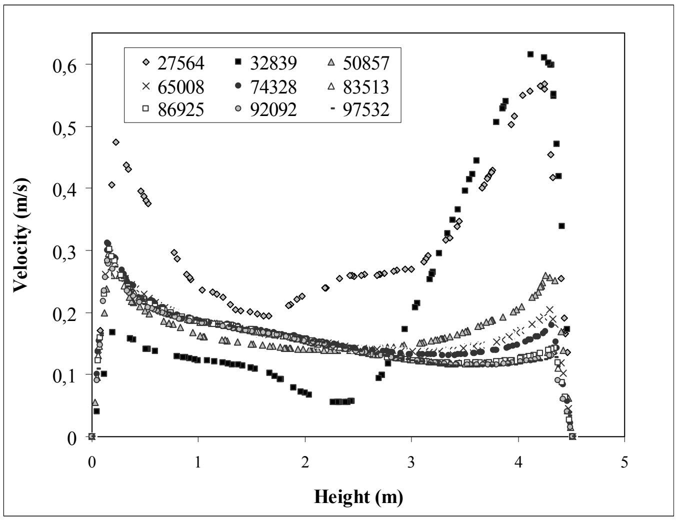

Fig.3 Velocity profile at the middle of the room (x=2.97, y=2.03) for different grid sizes. Inlet airflow rate=2.18kg/s. Isothermal conditions.

The type of grid used is ideally suited for the discretisation of complicated geometrical domains and allows an exact description of the test room geometry. As the flow field is a complex one, therefore there is no advantage in using a hexahedral (structured) mesh since the flow is not aligned with the mesh. Furthermore, the grid can be refined without adding unnecessary cells in the other parts of the domain as happens in the structured grid approach (hexahedral cells).

3.2.3 Boundary conditions

The equations are solved with boundary conditions at the air inlet and outlet, and on the internal surfaces and any obstructions in the test room.

Inlet-Outlet: Various inlet airflow rates assuming uniform inlet velocity profile have been applied (0.5-2.5kg/s) with inlet temperature T_{in}=305.5K. However, various factors, including a temperature difference between the room and supply air, may create velocity profiles at the inlet which are far from uniform. In the cases studied the two doors located in the opposite walls of the test room are considered open, thus air enters the room through the one door and leaves through the other. In all cases the inlet air direction was taken to be normal to the inlet. Values for the turbulent kinetic energy, k_{in} (m^2/s^2), and dissipation rate, ε_{in} (m^2/s^3) at the inlet were found from literature [5]. At the outlet, zero reference pressure has been specified. The kinetic energy of turbulence (k) and its dissipation rate (ε) at the outlet are not required due to the upwind computational scheme used.

Walls: As the walls are impermeable, the normal velocities are zero at the boundaries. Two options were checked: 1) The boundary conditions at the walls for velocity components and k-ε are specified using the logarithmic wall functions [10]. The standard wall function has been found to work reasonably well. 2) Enhanced wall treatment which is a near-wall modeling method [13] that combines a two-layer model with enhanced wall functions.

In the non-isothermal case a fixed temperature condition is applied at all the internal walls equal to T_w=302K.

3.2.4 Computational details

The solution of the set of the equations together with the boundary and internal conditions has been made with the segregated steady-state solver [12] embodied in Fluent® commercial software. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. In addition, the presence of internal wall in the flow may introduce an additional stability problem during the calculation. For pressure-

velocity coupling Fluent® provides three methods in the segregated solver: SIMPLE, SIMPLEC, and PISO. SIMPLE has been used in all cases studied.
Because of the non-linearity of the problem the solution process is controlled via relaxation factors that control the change of the variables as calculated at each iteration. The convergence is checked by several criteria (e.g. the conservation equations should be balanced; the residuals of the discretised conservation equations must steadily decrease).

4 Results and Discussion

In this section typical results of the research work presented above are shown and discussed. In Figures 4 and 5 the computed velocity profile assuming isothermal conditions is shown at the middle of the room and close to the west side using the three different turbulence models.

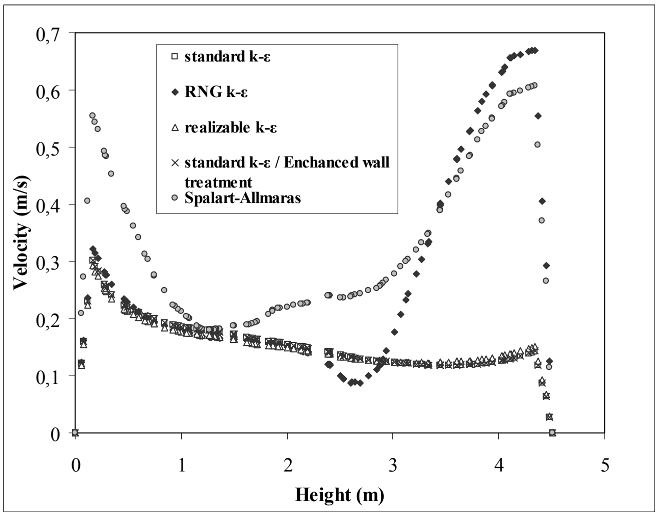

Fig.4 Velocity profile at the middle of the room (x=2.97, y=2.03) using various turbulence models. Inlet airflow rate=2.18kg/s. Isothermal conditions.

As it is shown there are important differences between the predicted results. The RNG k-ε and Spalart-Almaras models predict higher velocities in the roof area. The standard k-ε model with standard wall functions and with enhanced wall treatment, and the realizable k-ε model give almost the same results. The applicability of the turbulence model that will be used for the simulation of indoor airflow in the test room will be decided after comparison with systematic experimental measurements that have already been scheduled for the next steps of the work. First results show that the standard k-ε model with standard wall functions predicts sufficiently the air flow in the test room. In Figure 6 the computed velocity profile at the middle of the room (x=2.97, y=2.03) for various airflow rates is shown for isothermal conditions. In all cases studied, the velocity is higher closer to the floor level decreasing with height increase and slightly increasing in the roof area.

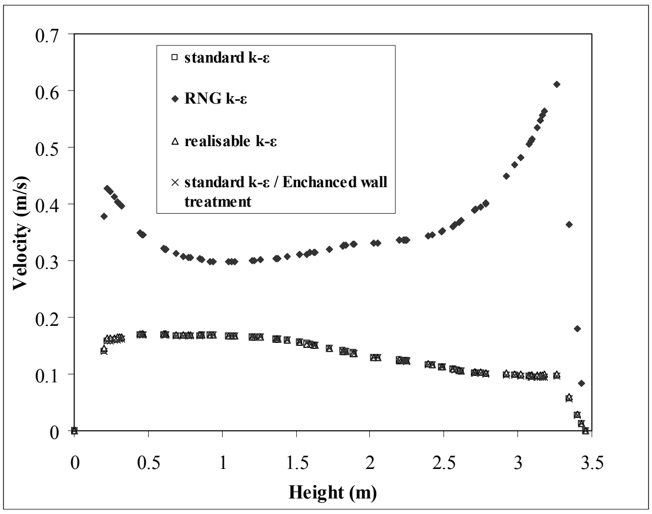

Fig.5 Velocity profile close to the west side (x=0.7,y=2.03) using various turbulence models. Inlet airflow rate=2.18kg/s. Isothermal conditions.

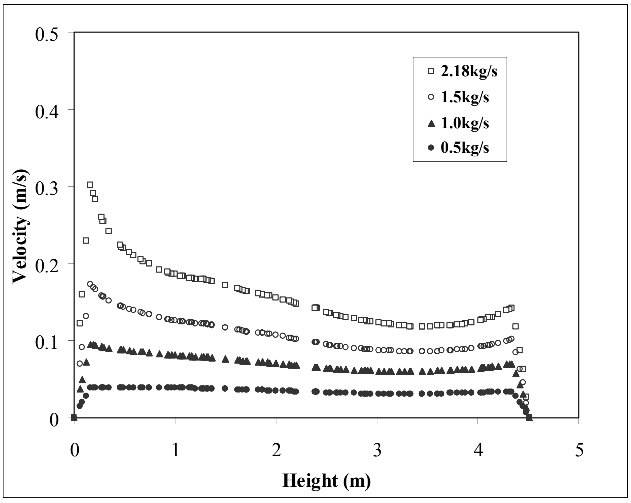

Fig.6 Velocity profile at the middle of the room (x=2.97, y=2.03) using various inlet feed rates. Isothermal conditions.

For the validation of the numerical predictions a number of measurements have been conducted. These measured values were used for setting the boundary conditions in the numerical predictions, in order to achieve comparable conditions to the experiments. As it is shown in Table 1 both models predict velocity magnitude with sufficient accuracy and the non-isothermal model seems to give slightly better predictions. This is due to the fact that the temperature difference inside the room is quite small thus its influence is small in the calculation of the flow field. In Table 2 typical values of temperature at various points inside the test room are presented.

The model predicts with sufficient accuracy temperature especially over the height of 1m.

Next steps of this work focus on further improvement of the model and additional validation of numerical predictions on indoor air flow. Also, as measuring and predicting the air velocity and temperature alone is not sufficient to assess the quality of air in the indoor environment it is planned to study and other parameters including pollutant concentrations and humidity.

Table 1 Model validation. Typical values of velocity magnitude at various points inside the test room. Inlet airflow rate=2.18kg/s. T_{in}=305.5K

1: Middle of the test room

2: Close to the west side (x=0.7,y=2.03)

	Distance from the floor (m)	Isothermal Model (m/s)	Non-Isothermal Model (m/s)	Measurement (m/s)
1	0.20	0.28	0.28	0.30
1	2.60	0.13	0.10	0.12
1	3.00	0.12	0.21	0.22
2	0.50	0.16	0.29	0.24
2	1.00	0.16	0.27	0.17
2	2.00	0.12	0.28	0.14

Table 2 Model validation. Typical values of temperature at various points inside the test room. Inlet airflow rate=2.18kg/s. T_{in}=305.5K

1: Middle of the test room

2: Close to the east side (x=5.3,y=2.03)

	Distance from the floor (m)	Non-Isothermal Model (K)	Measurement (K)
1	1.00	304.84	303.9
1	1.60	304.89	304.8
1	2.00	304.92	304.5
2	0.50	304.80	304.5
2	1.00	304.87	305.5
2	1.50	304.92	305.5

5 Conclusions

A full-scale test room was built with the purpose to conduct experiments under various test conditions to evaluate specific building issues. The measurement plan has been organized to proceed with the measurements of the indoor air quality quantities in the test room. Preliminary measurements have been obtained for air velocity and temperature. A three-dimensional steady state numerical model has been developed to describe indoor airflow in the room. Various cases have been considered accounting for isothermal and non-isothermal conditions. Comparison with preliminary experimental results shows that the model predicts with sufficient accuracy the air flow in the test room.

6 Acknowledgement

Authors would like to thank Hellenic Ministry of National Education and Religious Affairs and European Commission that financially supported this work under the auspices of the Operational Programme for Education and Initial Vocational Training, Archimedes Programme.

References:

[1] A.P. Jones, Indoor air quality and health, *Atmospheric Environment*, Vol. 33, 1999, pp. 4535-4564.

[2] S. Murakami, S. Kato, Numerical and experimental study on room airflow-3D predictions using the k-ε turbulence model, *Building and Environment*, Vol. 24 (1), 1989, pp. 85-97.

[3] F. Haghighat, Y. Li, A.C. Megri, Development and validation of a zonal model – POMA, *Building and Environment*, Vol. 36, 2001, pp. 1039-1047.

[4] H.E. Feustel, COMIS - An international multizone airflow and contaminant transport model, *Energy and Buildings*, Vol. 30, 1999, pp. 3-18.

[5] H.B. Awbi, Application of Computational Fluid Dynamics in Room Ventilation, *Building and Environment*, Vol. 24 (1), 1989, pp. 73-84.

[6] *Thermal Environmental Conditions for Human Occupancy*, ANSI/ASHRAE 55, 1992.

[7] P.V. Nielsen, *Flow in air conditioned rooms*, PhD thesis, Technical University of Denmark, 1974.

[8] S.V. Patankar, *Numerical heat transfer & fluid flow*, Hemisphere Publishing Corporation, Washington, 1980.

[9] B.E. Launder, D.B. Spalding, The Numerical Computation of Turbulent Flow, *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, 1974, pp. 269-289.

[10] W. Rodi, *Turbulence models and their application in hydraulics – A state of the art review*, Int. Association for Hydraulics Research, 1980.

[11] V. Yakhot, S.A. Orszag, Renormalization group analysis of turbulence. I. Basic theory, *Journal of Scientific Computing*, Vol. 1 (1), 1986, pp. 3-51.

[12] Fluent® 6.1 User's Guide 2003.

[13] B. Kader, Temperature and Concentration Profiles in Fully Turbulent Boundary Layers, *Int. J. Heat Mass Transfer*, Vol. 24(9), 1993, pp. 1541-1544.