CHARACTERISATION OF NATURAL CONVECTION IN A ROOM COMMUNICATING WITH THE OUTSIDE ENVIRONMENT THROUGH A DOOR

M G Carvalho, J E Ramos and R A Pitarma
Department of Mechanical Engineering
Thecnical University of Lisbon, 1096 Lisbon Codex; Portugal
e-mail: maria@navier.ist.utl.pt
Department of Mechanical Engineering
Polithecnic Institute of Leiria, 2404 Leiria Codex; Portugal
e-mail: jramos@estg.iplei.pt
Department of Mechanical Engineering
Polithecnic Institute of Guarda, 6300 Guarda; Portugal
e-mail: rpitarma@ipg.pt

ABSTRACT The present paper describes a numerical method for analysing three-dimensional natural convection in rooms connected to the outside through large openings. The calculations made use of a Computational Fluid Dynamics (CDF) procedure which solves the three-dimensional equations for the conservation of mass, momentum and thermal energy taking into account the effects of buoyancy, heat sources, thermal radiation heat transfer and air flow turbulence.
The CFD predictions agree with the experimentally observed features of the flow field, including the vertical air temperature stratification, with a reasonable degree of accuracy.

1 Introduction

The growing interest in the study of natural convection in geometrically complex enclosures with restricted communication has been stimulated by applications involving energy-efficient passive-solar buildings, cryogenic storage equipment, natural convection cooling of electronic equipment and room heating and ventilation. Airflow through doorways, windows and other large openings are significant ways in which air, pollutants and thermal energy are transferred from one zone of a building to another or to outside.

The nature and strength of airflow through doorways due to temperature differences between inside and outside has received a good deal of attention due to the new found popularity of natural ventilation as an alternative to air conditioning. Buoyancy-driven flow through a doorway and down a long a corridor has been studied by Lane-Serff (1989), amongst others.

Field tests on natural ventilation systems are difficult due to the uncertainty of various factors influencing their performance. The development of fast computers and accurate computer algorithms has presented a potential alternative to physical testing on the performance of ventilation systems. No code was able to solve airflow through doorways and windows problems other than by computational fluid dynamics (CFD) simulation. In the last decade or so, there have been a lot of applications of CFD in the simulation of air flow in rooms (Awbi 1989; Holmes *et al.* 1990; Niu and Kooi 1992; Said *et al.* 1995; Yaghoubi *et al.* 1995 and Ramos 1997).

2 Experimental setup and measurement procedures

The investigation has been carried out by means of physical measurement and numerical prediction of the environment in a naturally ventilated office room communicating with a 2.5 m high corridor via a 2 m high hinged wooden door opening. Fig.1 shows the office room tested in an schematic layout.

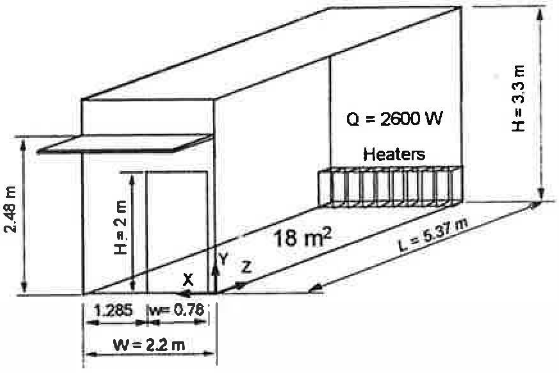

Fig.1 Schematic of the room configuration

Inside the room, near the external front wall, there are 4 electric air heater radiant units. Two of them were 500 W and the other were 800 W, each.

Experimental techniques are used to measure the air temperature distributions, to measure the velocity air flow pattern and to have flow visualizations.

2.1 Air temperature measurements

A vertical support with 6 Cu/Cu-Ni isolated thermocouples (0.5 mm wire diameter) was used for air temperature measurements inside the room and an other similar support with 8 thermocouples was used for the door opening section. Inside the room, the thermocouples were located from the floor to about 3 m in height. Thermocouple signals were acquired by a 12 bits Data Translation board (DT2811/DT756Y) programmed by an IBM microcomputer and were converted to temperature values by specific software with an accuracy of ± 1°C.

2.2 Air velocity components measurements

The air velocity components were measured using an *Airflow TA 400* temperature-compensated probe with a 0.06 m/s precision. The probe consists of two small spheres. One of them is electrically heated and maintained at a constant temperature. The probe was attached to a tripod and was displaced in a 7x10x11 measuring grid. In each position, for each velocity component, the air velocity was measured once per 4 seconds, over a 120 second measuring period.

2.3 Flow visualization

Injection of smoke in the air is probably the most common method of visualizing the air motion in a room. The smoke particles when illuminated cause scattering of light and their movement can then be traced and photographed. Because of low room air velocities it is essential that the smoke used for visualization has a density close to that of air, otherwise the effect of buoyancy could produce a false impression of the air motion. In this work flow visualization was carried out in the door opening section and inside the room.

3 Computational fluid dynamics algorithm

The equations that describe the flow of a fluid and heat transfer within an enclosure are all based on the conservation of mass, momentum and thermal energy within the enclosure. In most cases, the indoor airflows encountered in rooms are turbulent. The mathematical modeling of turbulent flow is within the capabilities of modem mathematical and numerical methods.

Among the turbulent models, the k-ε two-equation buoyancy-extended turbulence model, based on eddy-viscosity/eddy-diffusivity concepts (Launder and Spalding 1984), seems most widely used for airflow studies in rooms. The k stands for the kinetic energy of turbulence and ε for the dissipation rate of turbulence energy.

Since the temperature difference in room air is relatively small compared to the mean Kelvin temperature, it is a common practice to use the Boussinesq approximation. This approximation takes air density as constant and considers buoyancy influence on air movement in the momentum equation. Since the k and ε equations are derived from the momentum equation, the buoyancy term in the momentum equation is then changed into buoyancy production terms in the k and ε equations.

3.1 Airflow and heat transfer models

The density in the transport equations is represented by the equation of state and the governing equations are represented by the continuity equation, the incompressible Navier-Stokes equations, the energy conservation equation, the turbulence energy conservation and the dissipation rate of turbulence energy. The governing equations for the averaged steady flow can be written in the general elliptic form for an incompressible fluid as:

$$\frac{\partial(U_j\phi)}{\partial x_j} - \partial\left(\Gamma_\phi \frac{\partial\phi}{\partial x_j}\right)/\partial x_j = S_\phi \qquad (1)$$

Γ_ϕ and S_ϕ are identified for each governing equation in Table 1.

Table 1 Governing equations parameters

ϕ	Γ_ϕ	S_ϕ
1	0	0 (continuity)
U_i	$\nu+\nu_t$	$\left(-\frac{1}{\rho}\right)\frac{\partial P}{\partial x_i} - \beta g_i\theta$
θ	$\nu+\nu_t/\sigma_\theta$	$q/\rho c_p$
k	$\nu+\nu_t/\sigma_k$	$\nu_t+G-\varepsilon$
ε	$\nu+\nu_t/\sigma_\varepsilon$	$(C_1\nu_t S - C_2\varepsilon + C_3 G)\,\varepsilon/k$

with:

$$\nu_t = C_\mu k^2/\varepsilon$$

$$S = \left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}$$

$$G = \beta g_i \frac{\partial\theta}{\partial x_i}\nu_t/\sigma_\theta$$

For the empirical constants in the turbulence model, the standard values, recommended by Launder and Spalding (1984), were used: $C_\mu = 0.09$; $\sigma_k = 1.0$; $\sigma_\varepsilon = 1.3$; $\sigma_t = 0.9$; $C_{\varepsilon 1} = 1.44$; $C_{\varepsilon 2} = 1.92$; $C_{\varepsilon 3} = 1.00$. The effective viscosity concept $\mu_{eff} = (\mu + \mu_t)$ has been used to represent the combined molecular viscosity and turbulent (Reynolds) viscosity $\mu_t = \rho_0 C_\mu k^2 / \varepsilon$ and the effective diffusion coefficient is determined by $\Gamma_{eff} = (\mu/\sigma + \mu_t/\sigma_t)$, where σ is the Prandtl/Scmidt number.

3.2 Boundary conditions

The accuracy of the solution of the discretization equations presented in the previous section will depend on the accuracy of specifying the physical quantities at the boundary of the flow domain and on the method of linking these quantities to the bulk of the flow.

Wall Boundary:

Because of the damping effect of the wall, the transport equation for the turbulence quantities does not apply close to the wall. The alternative is to extend the Couette flow analysis and apply algebraic relations, the so-called logarithmic laws or wall functions for momentum and heat fluxes (Launder and Spalding 1984), for the calculation of the velocity parallel to the boundary components and the heat flux through the boundary.
The problem presents a conjugate heat transfer problem. Unlike the convection-conduction problem, the radiation-convection-conduction interactions at the air-wall interface give rise to complexities in the numerical method which requires special treatment. The air-wall interface is matched by the heat balance equation $q_{rad} + q_{conv} = q_w$, where q_w represents the heat conduction through the wall and q_{conv} represents the convection along the inside wall surface. The q_{rad} term represents the wall net radiant flux.

Free Boundary:

The room internal space is in communication with the corridor environment through a 2 m door opening. For economical reasons, the calculation domain does not extend all the corridor, outside the room. The flow is linked to the outdoor corridor conditions by an adequate treatment of the boundary conditions using a free boundary. A free boundary is not a physical boundary, but is a limit that imposes on the domain of calculation, a surface where the pressure is prescribed. At this free boundary, the fluid is entrained at an unknown rate. Thus, the boundary condition consists of prescribing the pressure equal to the atmospheric value sufficiently far from the door opening.

3.3 Numerical solution procedure

The general partial differential equation (1) has been discretized by means of a finite volume method, i.e., by integration of the equations over a control volume on a mesh to yield finite difference equations (Patankar 1980).
A staggered non-uniform grid with velocity nodes offset from scalar nodes is used in the present computations. A central difference scheme is used to discretize the diffusion terms and a hybrid upwind/central differencing scheme is used to discretize the convection terms. The continuity equation is rewritten into an equation for the pressure correction and the pressure-momentum linked equations are solved by the SIMPLE algorithm (Patankar and Spalding 1972). The resulting algebraic equations are solved in an iteration sequential manner by using the TDMA (tri-diagonal matrix algorithm) line-by-line method (Carnahan *et al.* 1969).

4 CFD validation and predictions

In this section, comparisons between the predicted and measured behavior of the flow field and the air heat transfer are presented and discussed. Experiments include measurements of mean air velocity and mean air temperature distribution, that allow to test the numerical model performance.
The Fig.2 shows the predicted steady-state longitudinal mean air velocity components profile on a longitudinal plane (x = 0.53 m). Fig.3 shows the comparisons between the predicted and its data, on the same plane, in the door opening section (z/L = 0).

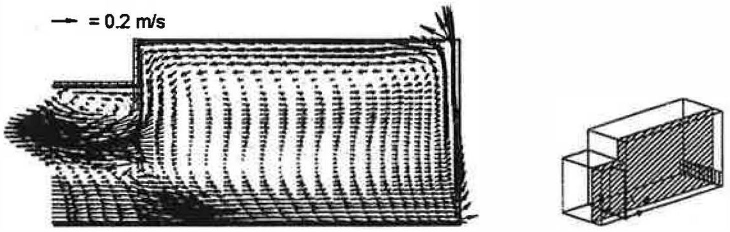

Fig.2 Predicted mean air velocity components on a longitudinal plane

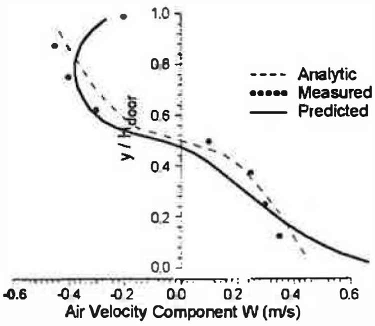

Fig.3 Comparison between the air velocity component prediction and its data in the door opening section

The comparisons between the measurements and the predictions indicated that the predicted flow pattern follows closely the experimental values. The incoming air flows from the corridor through the lower half-height door opening and then takes the form of a gravity current flowing horizontally along the floor in the direction of the air heaters. Beyond, due to buoyancy effect, the air goes up to the ceiling. The outgoing air flows to the doorway direction where it leaves the room to the corridor (cold zone), as a turbulent jet, by the upper half-height door opening.

The measurements of air temperature distribution are also consistent with the CFD predictions (Fig.4). The predicted vertical temperature stratification is generally in good agreement with the experiment. However, the CFD results show a cooler hot zone, near the air heaters, than that observed experimentally.

5 Concluding remarks

Natural convection in a heated office room connected to a corridor via a door opening has been predicted using a three-dimensional CFD method, and the results have been compared to experimental data obtained in full-scale.

The room was equipped with a vertical support with 6 Cu/Cu-Ni thermocouples connected to a data acquisition board and with a thermo-anemometer temperature compensated probe attached to a tripod. In each support position the air temperature and the air velocity was measured.

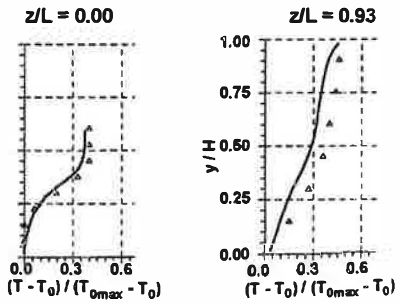

Fig.4 Comparison between the air temperature prediction and its data in two sections

The air flow pattern, the temperature distribution and the radiation heat transfer has been predicted solving the three-dimensional stationary equations for the conservation of mass, momentum and thermal energy, taking account of the effects of buoyancy, airflow turbulence, transfer of thermal radiation and thermal conduction through the walls, heat sources and obstacles in the room. The turbulence parameters had been modeled by the k-ε model. The validation of the ventilation model has show that 3-D CFD code is a useful tool to predict the air flow and the heat and mass transfer in naturally ventilated rooms provided that accurate measurements are made for dominant parameters such as air velocity components and air temperature distribution.

This work is to be extended for analyzing the influence of occupants and their distribution in the space on indoor air quality and comfort and assessing the ventilation effectiveness by appropriate arrangements of windows or door openings.

6 References

Awbi, H. B., J. A. (1989): Application of Computational Fluid Dynamics in Room Ventilation. *Building and Environment*, 24 (1), pp.73-84.

Carnahan, B.; Luther, H. A. and Wilkes, J. O (1969): *Applied Numerical Methods*.UK: John Wiley & Sons.

Gosman, A. D.; Pun, W. N; Runchal, A. K; Spalding, D. B; Wolfshtein, M. (1969): *Heat and Mass Transfer in Recirculating Flows*. London & New York: Ac. Press

Holmes, M. J.; Lam, J. K. W.; Ruddick, K. G. and Whittle, G. E. (1990): Computation of Conduction, Convection and Radiation in the Perimeter Zone of an Office Space.Oslo, Norway: *Roomvent '90*.

Lane-Serff, Gregory (1989): *Heat Flow and Air Movement in Buildings*.UK: Ph. D. Thesis of Philosophy, University of Cambridge

Launder, B. E. and Spalding, D. B. (1984): The Numerical Computation of Turbulent Flows .*Compt. Methods Applied Mech. & Eng .*; 3; pp. 269-289.

Niu, J. and Kooi, J. van der. (1992): Two-Dimensional Simulation of Airflow and Thermal Comfort in a Room with Open-Window and Indoor Cooling Systems. *Energy and Buildings*, 18, pp. 65-75.

Patankar, Suhas V. (1980): *Numerical Heat Transfer and Fluid Flow*. Washington: Hemisphere Publishing Corporation

Patankar, Suhas V. and SPALDING, D. B. (1972): A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows. *Int. Journal of Heat and Mass Transfer*, 15, pp. 1787-1806.

Ramos, João. (1997): *Modelação do Comportamento Térmico e Ambiental em Edifícios* Portugal: Ph. D. Thesis, Technical University of Lisbon