#7530

Building Serv. Eng. Res. Technol. **14**(4) 169–178 (1993) Printed in Great Britain

Summary A numerical study of natural and forced convection inside an auditorium of the National University of Mexico was performed. Two-dimensional steady-state simulations of the heat transfer and fluid flow processes within a cross-section of the auditorium were carried out using a CFD program. The boundary conditions were varied to obtain Rayleigh numbers in the range 10^3 to 1.85×10^{10} for natural convection and Reynolds numbers of 10^3 to 10^5 for the forced convection flows. The κ–ε model of turbulence was employed to predict turbulent forced convection. The aim of the study was to determine the air flow patterns and temperature field and their effect on thermal comfort conditions inside the auditorium.

Natural and forced convection in an auditorium

B Vazquez and **M Yianneskis**
Centre for Heat Transfer and Fluid Flow Measurement, School of Physical Sciences and Engineering, King's College London, Strand, London WC2R 2LS, UK

Received 20 October 1992, in final form 4 June 1993

1 Introduction

A primary objective in building design is to provide appropriate thermal comfort conditions. The comfort of the persons inside a building is affected directly by the air speed and temperature gradients due to the convective processes within the building. It is therefore very important to predict the flow patterns and temperature distribution accurately in order to optimise the design of the equipment employed, the location of heating elements and/or ventilation openings etc. It is also important to minimise the energy expended for heating or cooling and to establish appropiate strategies for saving energy[1,2]. However, due to the complexity of the governing equations used in calculations of convective problems, analytical solutions are often not possible and the understanding of convection flows has therefore been restricted. The governing equations can, however, be solved using numerical methods built into computational fluid dynamics (CFD) programs. The main objective of the programme of research, of which only a small part is presented here, is to evaluate in detail the flow pattern inside an auditorium at the University of Mexico, and to estimate the effect of changes in the boundary conditions on the velocity and temperature distributions and on the comfort of the occupants. The first stage of the work, concerned with the effect of the location of the air inlets, has been reported in Reference 3. A large number of experimental and predictive investigations of ventilation flows have already been reported, and a representative sample is referred to below. Numerical studies have been carried out by Hertager and Magnussen[4], Timmons *et al.*[5] and Etheridge and Nolan[6]; it was reported that the near-wall zone, including the boundary layers, is often not modelled adequately, partly due to the size of the mesh used in that region and partly due to the limitations of the models. Full-scale chambers have been used for experimental studies under well controlled conditions, for example by Chen *et al.*[7], Bauman *et al.*[8], Gadgil[9] and Jones and O'Sullivan[10]. Chen *et al.*[7] and Timmons *et al.*[5] found that when mechanical air conditioning systems are used, the flow patterns depend strongly on the size, shape and location of the air inlets and outlets. Bauman *et al.*[8] found that the transition from laminar to turbulent flow could be delayed; in their work, laminar flow was maintained for Rayleigh numbers up to 6.75×10^9. Markatos and Pericleous[11] used a finite-volume two-dimensional method to solve the conservation equations for mass, momentum and energy as well as the two equations for the turbulence energy and its rate of dissipation in the κ–ε turbulence model, for natural convection flows in an square cavity. The cavity modelled had two differentially heated walls and two adiabatic horizontal surfaces corresponding to the floor and ceiling. The results were compared with published experimental data for laminar as well as turbulent convection. Markatos *et al.*[12] calculated the two-dimensional steady-state flow of a fire spreading in a room. Subsequently, Markatos *et al.*[13] calculated the corresponding three-dimensional flow under both steady-state and transient conditions, and included compressibility effects. The aim of the study was to model the fire development within a shopping mall and to predict smoke concentration levels. Only partial validation was possible due to the lack of experimental data. Recently[14,15] the application of CFD modelling has been extended to the modelling of two-phase fire events, using the Eulerian–Eulerian approach, particularly to predict fire-sprinkler interaction. The results obtained were compared with experimental data and showed good qualitative agreement. These studies highlighted the necessity for parallel-architecture computers to reduce cpu time usage for the simulations, and the urgent need for more detailed experimental data. Other studies that have made use of CFD methods include those of Chen *et al.*[16] and McGuirk and Whittle[17]. The latter paper emphasised that although CFD programs have proved to be a powerful tool for building design, they must be used with care in order to obtain the best results from them. McGuirk and Whittle suggested that a numerical benchmark test case should be used to evaluate the performance of various CFD programs for building design under different boundary conditions.

2 Flow configuration and numerical method

In the present study the steady-state air flow patterns, heat transfer coefficients and temperature distribution within a University of Mexico auditorium were calculated across a cross-section simulating the second row of seats. The thermophysical characteristics of air and the boundary conditions were identical to those measured in the auditorium. The section simulated is located under three ventilation fans: these are aeolic (wind-driven) turbines. The simulations were carried out for two-dimensional, steady-state natural and forced convection. The Boussinesq approximation was employed, so that all physical properties of the fluid remain constant except for the density in the body force term, which has been shown to be valid for temperature gradients inside a room of less than 28.6°C[18]. A two-dimensional configuration was chosen for this first stage of the work so as to minimise computational time costs and to allow an understanding of the relevant flow processes before the complexity of the flows in the auditorium was increased by the inclusion of three-dimensional and transient effects. The rectangular cross-section of the auditorium simulated can be seen in Figure 1. The vertical boundaries represent the east and west walls and the horizontal boundaries represent the floor and ceiling. One inlet is located in the lower part of each of the side walls and three outlets are located on the ceiling. The boundary at the bottom includes a central zone in which the 13 seats occupied by persons are located, while the areas adjacent to the walls represent the aisles. The heat generated by the seated occupants was modelled by employing a value of 33°C as the skin temperature and a body surface area of 1 m² per person. The Phoenics finite-volume base computer code[20] incorporating the κ–ε turbulence model was used for the predictions presented here for forced convection. For natural convection a laminar version of the same CFD code was used. The equations governing the forced and natural convection phenomena inside the auditorium, namely the conservation equations for mass, momentum and energy can be represented by the general form:

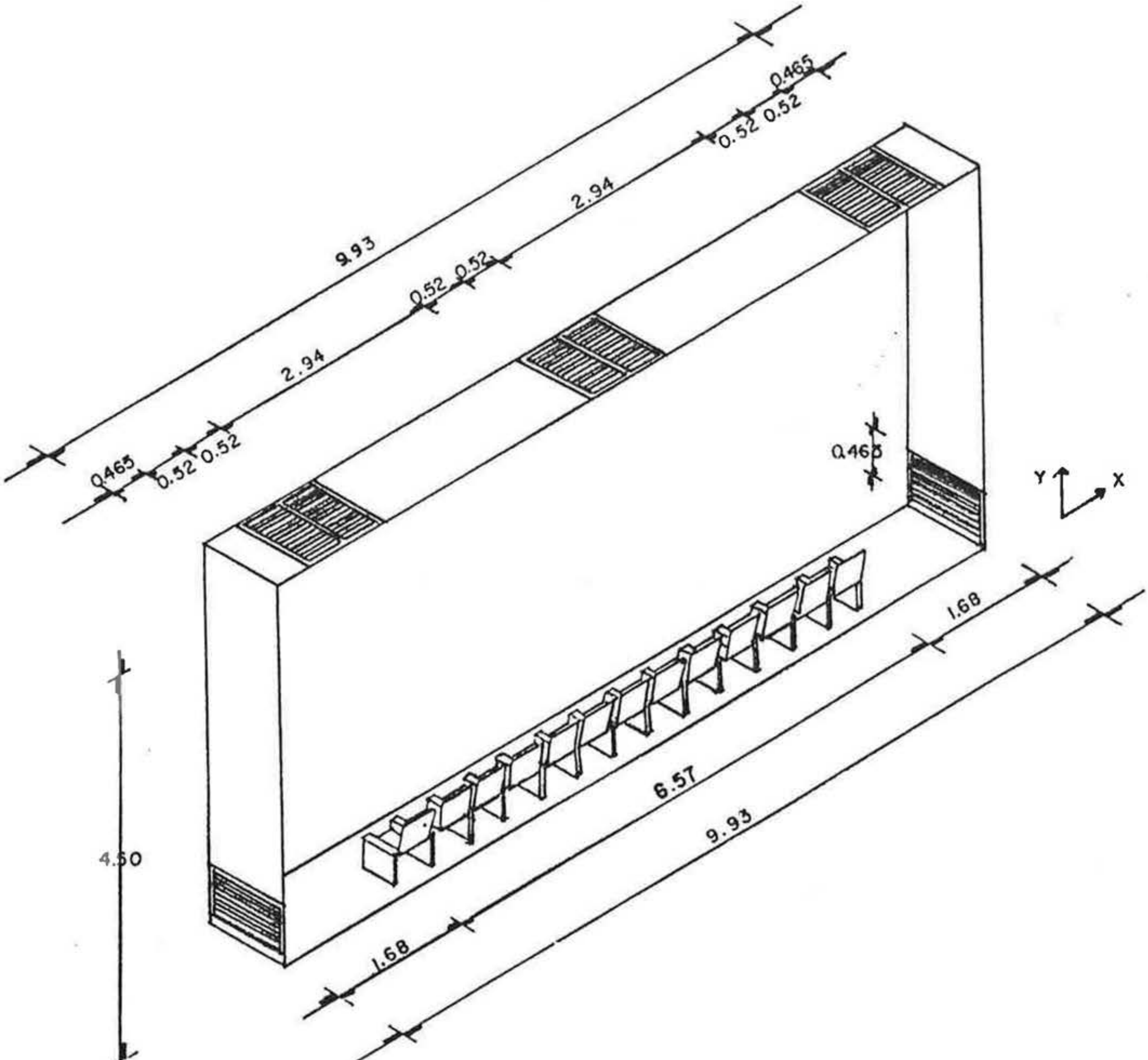


Figure 1 Auditorium cross-section (dimensions in m)

$$\frac{\partial}{\partial x_j}(\rho U_j \phi) = \frac{\partial}{\partial x_j}(\Gamma_\phi \frac{\partial}{\partial x_j}) + S_\phi$$

where the dependent variable ϕ may be one of the velocity components (U or V in the x and y directions respectively, or the enthalpy H. Two additional equations are solved for the

Table 1 Source terms in transport equations

Equation	ϕ	Γ_ϕ	S_ϕ
Continuity	1	0	0
Momentum in direction i	U_i	μ_{eff}	$\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j}[\mu_{eff}(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i})]$
Thermal energy	H	μ_{eff}/σ_{eff}	0
Turbulence energy	κ	μ_{eff}/σ_κ	$G - \rho\varepsilon$
Turbulence dissipation	ε	$\mu_{eff}/\sigma_\varepsilon$	$(\varepsilon/\kappa)(C_1 G - C_2\rho\varepsilon)$

turbulence energy κ; and the rate of dissipation of κ, ε. The coefficients Γ_ϕ and S_ϕ and corresponding to each of these dependent variables as well as the empirical coefficients appearing in the turbulence model are given in Table 1.

In Table 1

$$G \equiv \frac{\partial U_i}{\partial x_j}[(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i})] \qquad \mu_{eff} = \mu + \mu t = \mu + C_\mu \rho \kappa^{2/\varepsilon}$$

and for the turbulence model the constants are: $C_\mu = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$, $\sigma_t = 0.9$, $\sigma_\kappa = 1$ and $\sigma_\varepsilon = 1.22$.

The temperature of the boundaries (walls, ceiling and floor) and the incoming air temperature were given the following values, which were obtained from measurements[19] and therefore correspond to temperatures resulting from actual weather conditions: the east and west walls and floor 26°C, the ceiling 27°C, and the air entering through the inlets 22°C. For the natural convection flows the pressure at the air inlets and outlets was set to be equal to the atmospheric pressure, while in forced convection the Reynolds number (Re) was altered by varying the average flow velocity at the air outlets; for some of these cases the velocity used to extract the air inside the auditorium was varied in accordance with the number of air changes per hour necessary to maintain the specified quality of air inside the auditorium[21], in order to provide different inlet Reynolds numbers. The convergence criterion used in the simulation was to compare the results obtained after each iteration with those from the previous one and when the two sets of results differed by less than 10^{-6}, the calculation was terminated. Mass and energy balances were carried out to establish that both mass and energy were conserved. The initial velocity values were all zero and the starting value for the temperature field was 22°C. All initial field conditions were identical for all cases. Grid independence tests were carried out for both the natural and forced convection cases with computational meshes of sizes 10×10, 18×18, 20×20, 25×30, 46×36, 40×40 and 60×60. An irregular mesh of 46×36 volumes (in the x and y directions respectively) was found to provide a good compromise between adequate accuracy and acceptable computer time costs. This mesh was used for the simulation of all the cases presented here. For the natural convection cases the cpu time used was around 30 minutes per simulation, while for each forced convection run around 1.3 hours were used on a VAX 8800 computer.

3 Results and discussion

3.1 Natural convection

Flows with Rayleigh numbers Ra of 10^3, 10^4, 10^5, 10^6, 10^7 and 1.85×10^{10} were predicted, but the results for Ra = 10^3, 10^6 and 10^7 are not shown here for economy of presentation. The flow patterns inside the auditorium for Ra = 10^4 are shown in Figure 2: Two symmetric recirculations are formed with upward flow in the centre. This flow is similar to that observed both experimentally and numerically for natural convection in a cavity with an aspect ratio of 0.5 heated from below[22]; the auditorium aspect ratio is 0.473. The flow directions result from two main factors. First, the seats represent a zone of high temperature and a plume-like flow rising in the

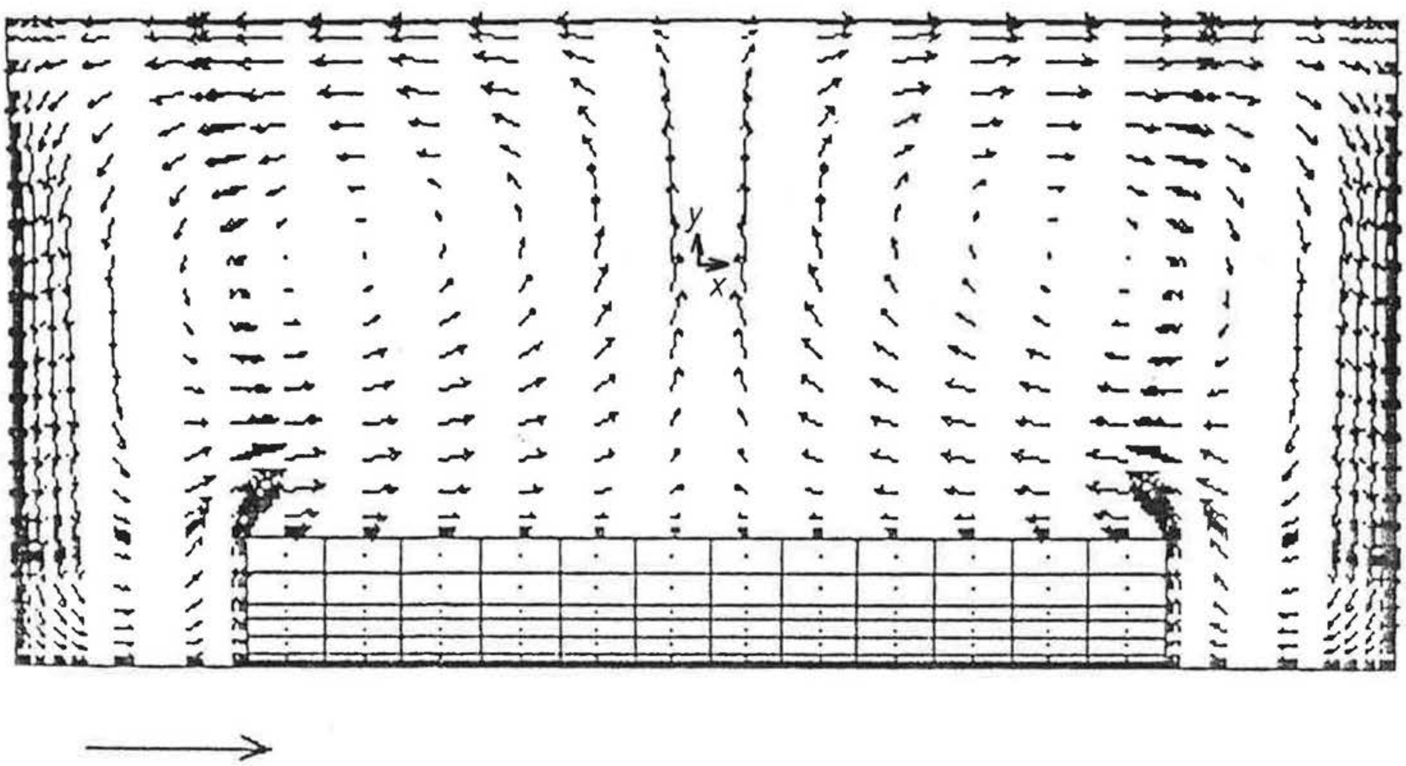


Figure 2 Velocity vector distribution, Ra = 10^4: Scale vector = 0.1 m s^{-1}; min: 0.0 m s^{-1}; max: 0.36 m s^{-1}

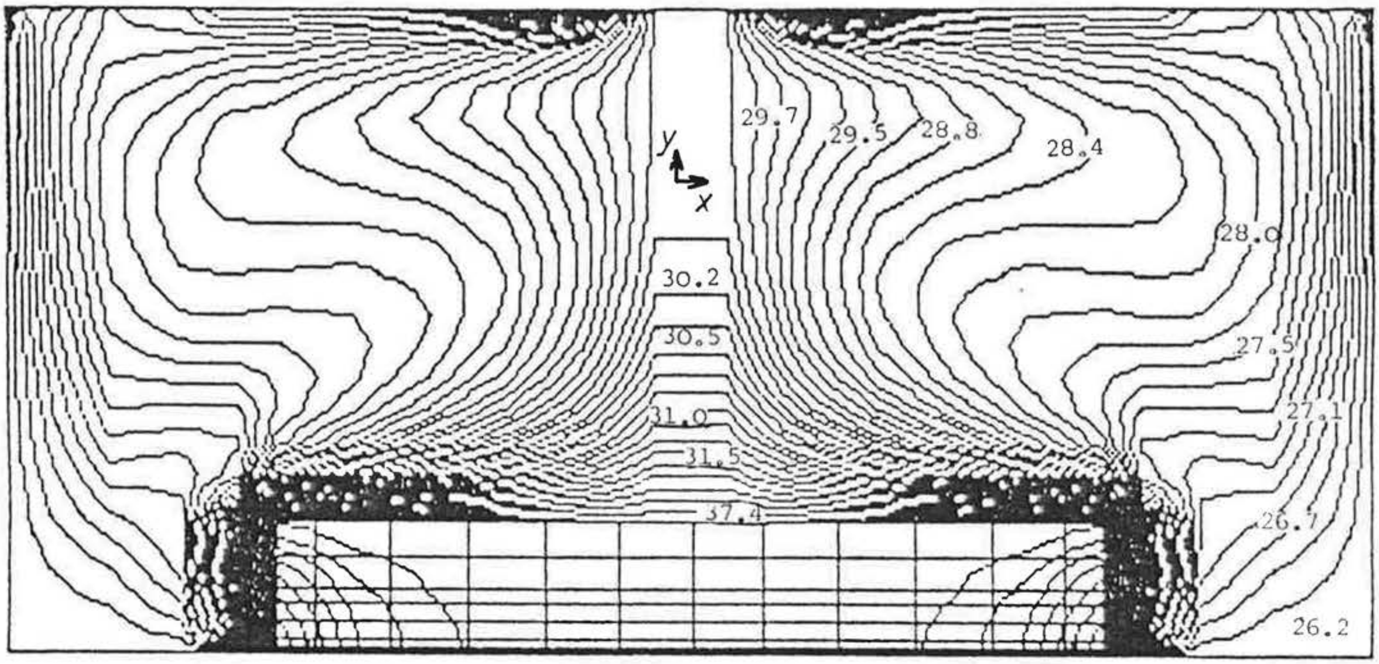


Figure 3 Isotherms, Ra = 10^4: Contours values in °C; T_{min} = 26.0°C; T_{max} = 32.99°C

centre is formed; second, the air in the auditorium is at a higher temperature than the walls and as it is cooled a downward flow is formed near the walls. This movement continues until the air reaches the floor, where it changes direction and mixes with the air entering through the inlets. The temperature contours for Ra = 10^4 are shown in Figure 3; the highest temperatures are found above the seats. Large gradients are present near the edges of the seats. The temperature field near the ceiling indicates the position of the three air outlets, where the isotherms are perpendicular to the ceiling, in contrast to the inlets which do not seem to have a large effect on the velocity and temperature fields. The results for Ra = 10^5 are shown in Figure 4. The two recirculation zones are more elongated in comparison with the previous case, as are the boundary layer regions near the walls. These are reflected in corresponding changes in the isotherm patterns, as can be observed in Figure 5. For Rayleigh numbers greater than 10^5, small asymmetries were observed in the distributions. These asymmetries could be due the precision of the numerical solution. The inclusion of the κ–ε model in the natural convection calculations may eliminate such asymmetries: this has been implemented by Markatos and Pericleous[11] for Rayleigh numbers greater than 10^6 in an square differentially heated cavity and by Ozoe *et al.*[23] for a water-filled rectangu-

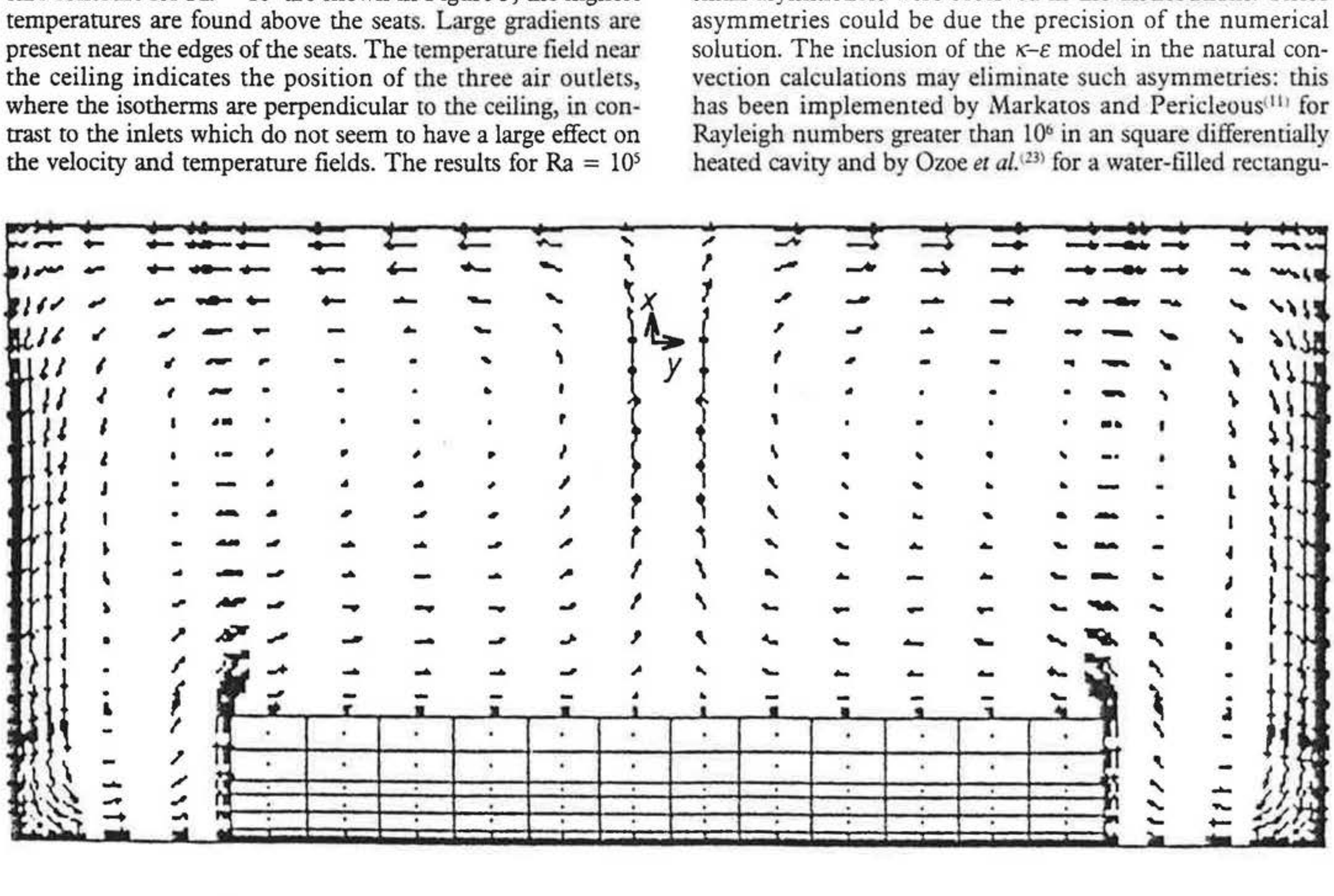


Figure 4 Velocity vector distribution, Ra = 10^5: Scale vector = 0.1 m s^{-1}; minimum: 0.0 mm s^{-1}; maximum 1.12 mm s^{-1}

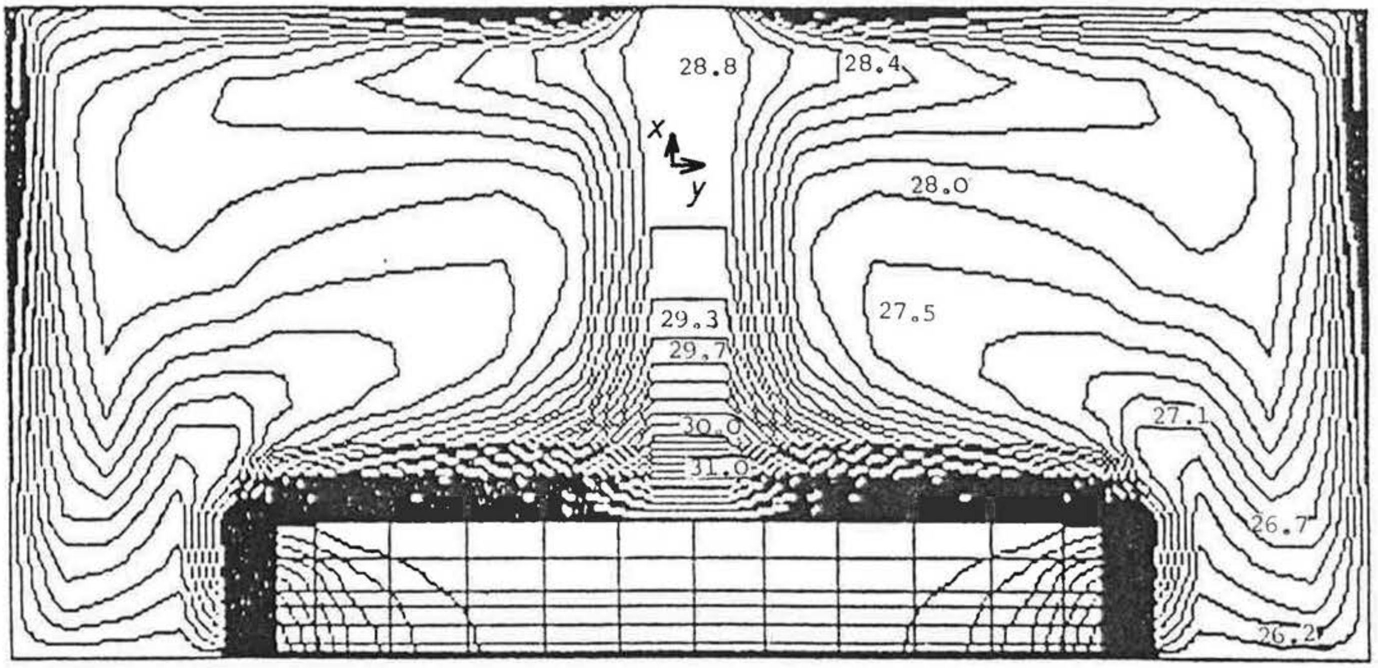


Figure 5 Isotherms, Ra = 10^5: Contour values in °C; T_{min} = 26.0°C; T_{max} = 32.99°C

lar cavity for Rayleigh numbers above 10^9. The flow for Ra = 1.85×10^{10} corresponds to the actual normal conditions in the auditorium. The velocity field is shown in Figure 6 and considerable changes can be observed in comparison with the previous cases: the flow entering through the inlets can be clearly distinguished, the near-wall airflow is directed upwards and the velocities over and around the seats are higher. Figure 7 shows the corresponding temperature con-

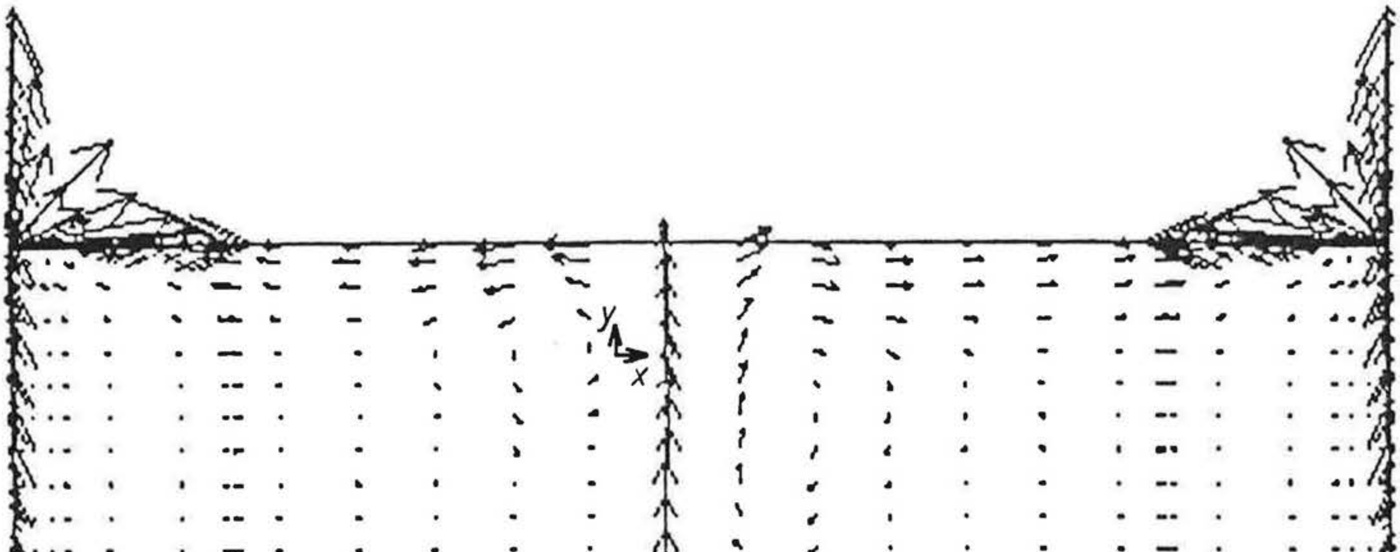


Figure 6 Velocity vector distribution, Ra = 1.85×10^{10}: Scale vector = 0.1 m s^{-1}

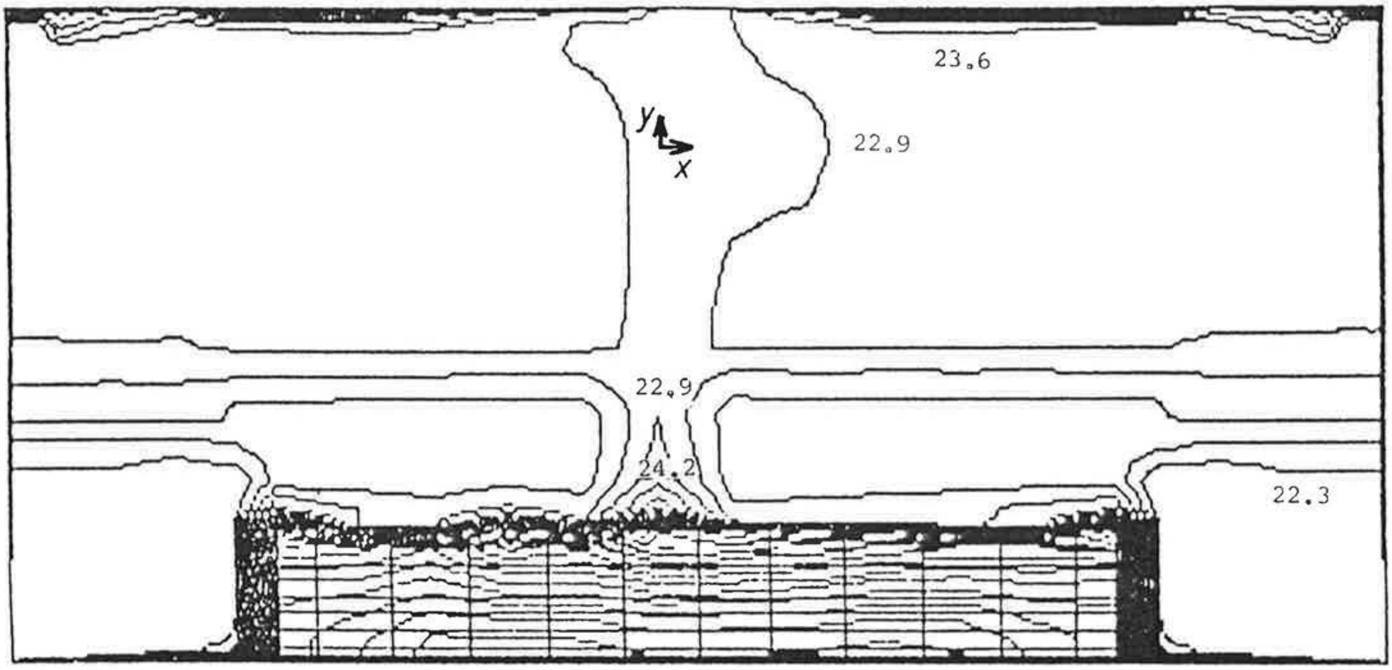


Figure 7 Isotherms, Ra = 1.85×10^{10}: Contour values in °C; T_{min} = 22.0°C; T_{max} = 31.77°C

tours: the temperature gradients in the auditorium are smaller and the near-wall temperatures are lower.

3.2 Forced convection

In the predictions presented below, the movement of air inside the auditorium is produced by the interaction of natural and forced convection effects. Predictions were made for Reynolds numbers of 10^3, 5×10^3, 10^4, 5×10^4 and 10^5, corresponding to 1.40, 6.98, 13.96, 69.8 and 139.60 air changes per hour. The Rayleigh number was kept constant at 1.85×10^{10} in all cases. The number of air changes per hour recommended by ASHRAE[21] is always less than 70, but the results for the two larger Re's were obtained in order to test the numerical model used. Those results not shown here are presented in Reference 24. It can be observed in Figure 8 (Re = 10^3), that the air flow moves upwards toward the outlets. The boundary layers over the walls are directly affected by the combination of forced and natural convection effects, which in this case act in the same direction, producing the upward movement. Part of the flow entering the auditorium is deflected by the seats. The flow entering through the inlets is more clearly formed than with natural convection only. Part of the flow moves upwards along the wall without making any contribution to the cooling of the auditorium. The maximum velocities are

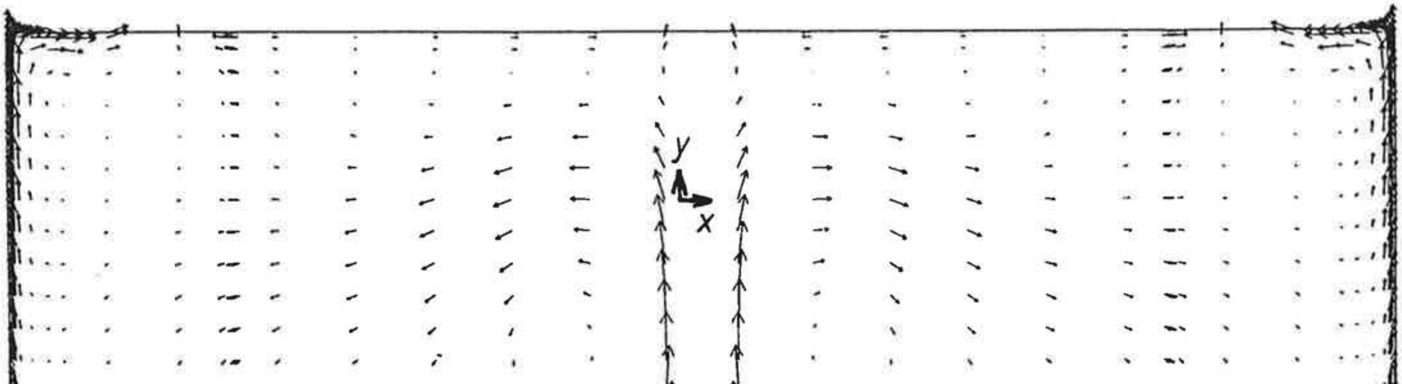


Figure 8 Velocity vector distribution, Re = 10^3

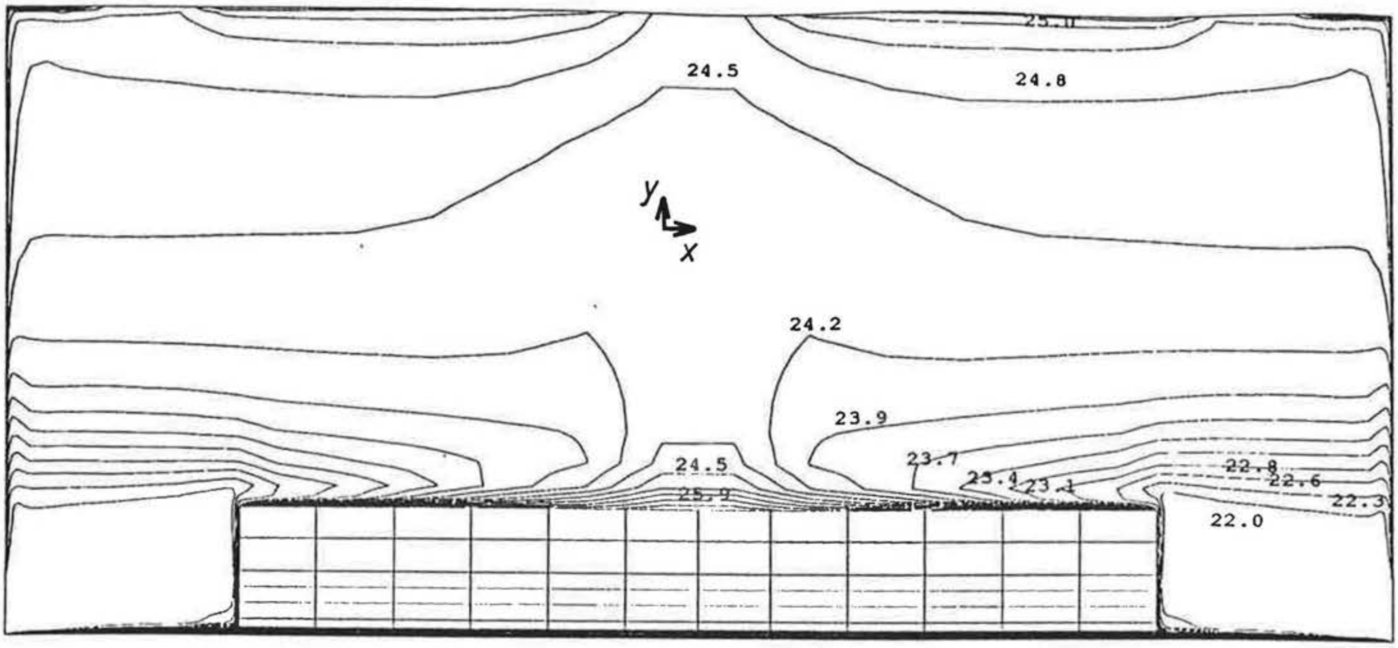
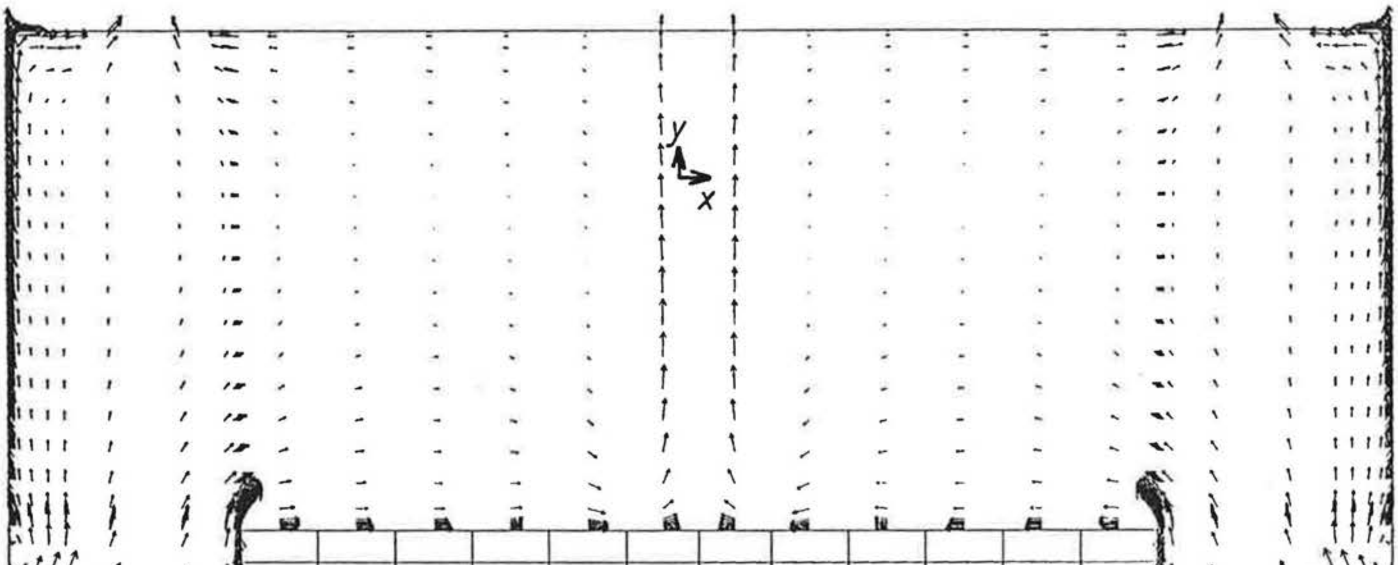


Figure 9 Isotherms, Re = 10^3: Contour values in °C; T_{min} = 22.0°C; T_{max} = 32.76°C

present near the walls and seats. The temperature contours are shown in Figure 9; the gradients inside the auditorium are relatively small. The flow with Re = 10^4 is shown in Figure 10. The effect of the air entering the auditorium is more pronounced than for Re = 10^3. The flow direction changes when the airstream meets the edge of the seats, producing a flow vertically upwards near the seat/aisle boundary toward the outlets. The velocities in the region above the seats are generally low, except for a small region near the centre of the room. The temperature field shows two large zones of uniform temperature of 22°C, Figure 11. There is clearly little mixing of the air above the seats and this was reflected in the low (near zero turbulence kinetic energy values calculated across most of the auditorium which are shown in Figure 12. In Figure 13, Re = 10^5, the flow entering the auditorium is again deflected by the seats but the velocities of the upward flow are larger. A recirculation region is formed over each aisle near the top end of the inlets, with the air near the lateral walls flowing downwards. Apart from the air flow deflected by the seats, practically all the air flow entering the auditorium is directed toward the aeolic turbines. Figure 14 shows that temperature gradients are present in the region above the seats and that a local accumulation of heat is formed over the centre.

3.3 Concluding remarks

Predictions of the natural and forced convection processes within an auditorium have been carried out with a finite-vol-



⟶ : 1.16 m/s. Min : 1.3038E-03 Max : 6.0252E-01

Figure 10 Velocity vector distribution, Re = 10^4

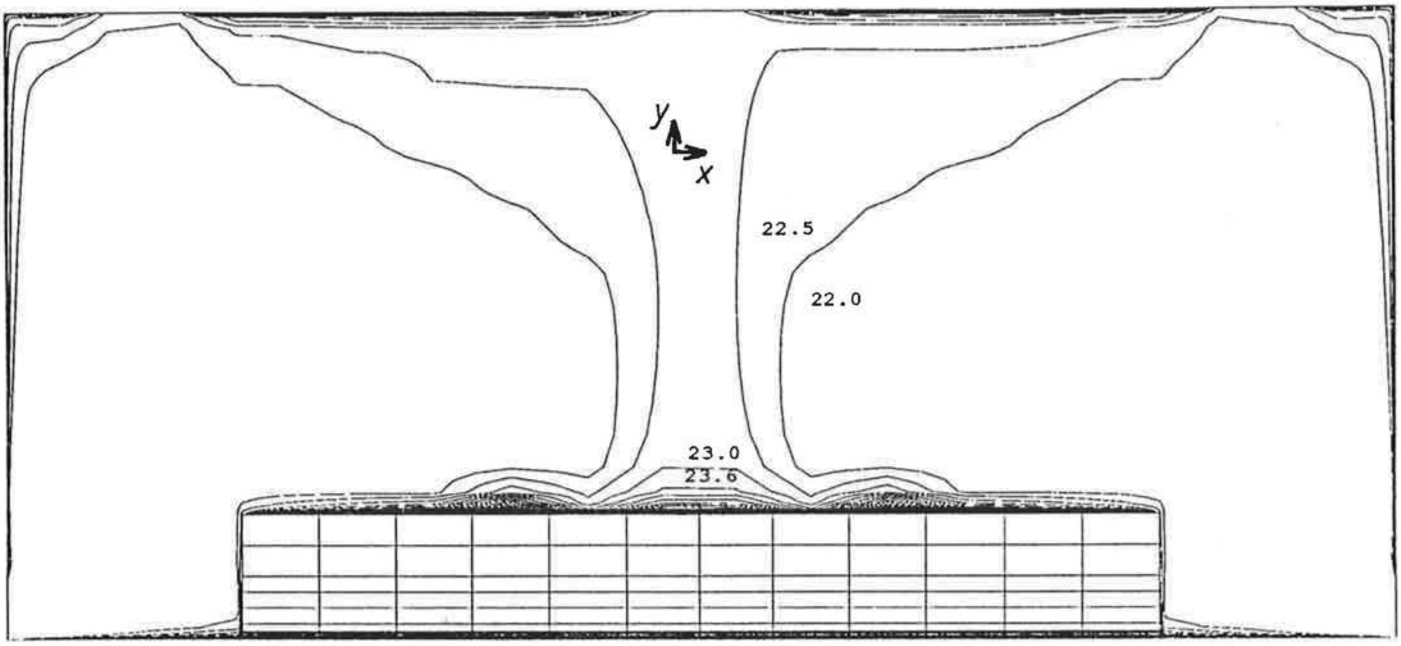


Figure 11 Isotherms, Re = 10^4: Contours values in °C; T_{min} = 22.0°C; T_{max} = 32.43 °C

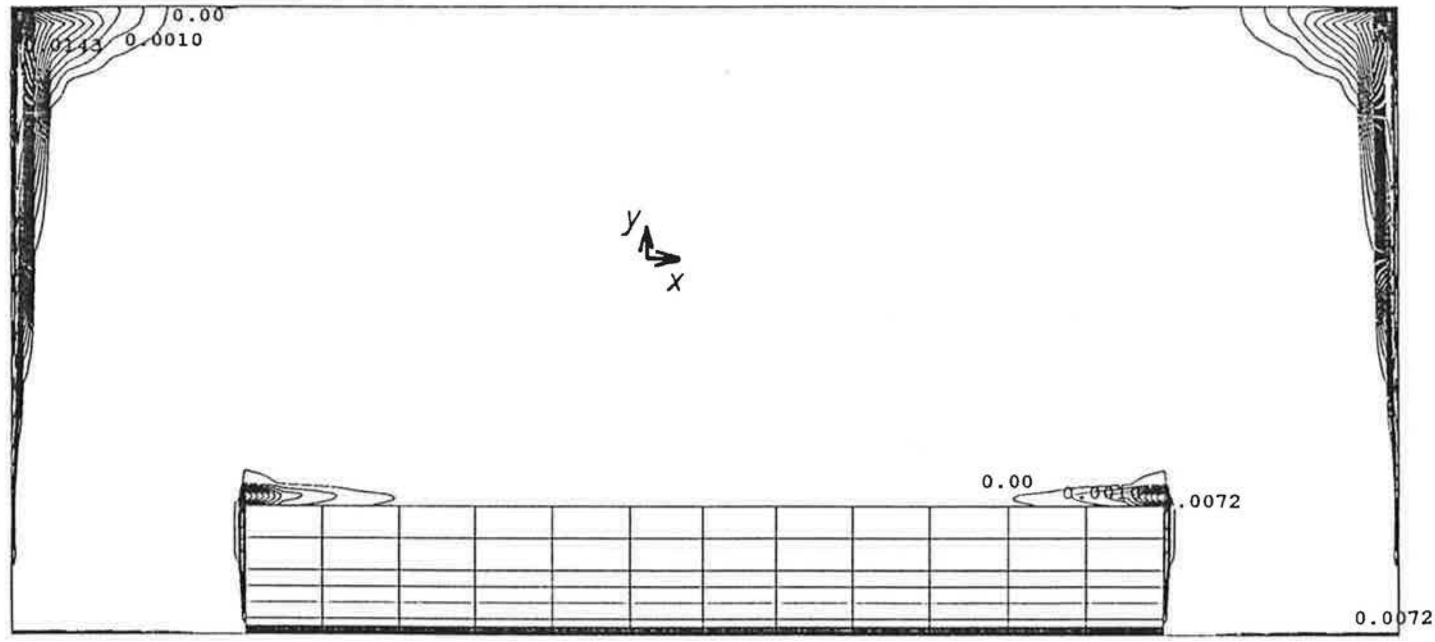


Figure 12 Turbulence kinetic energy countours, Re = 10^4; Contours values in $m^2 s^{-2}$; K_{max} = 0.014 $m^2 s^{-2}$

ume computational fluid dynamics code. The effects of increasing Rayleigh and Reynolds numbers were investigated. The predictions indicated a local accumulation of heat near the centre of the auditorium for most of the cases investigated. This is related to the strong deflection of the airflow by the seats, which results in relatively low convection velocities over the seat area with the low-wall inlet location which exists in the auditorium. The results presented above allow the evaluation of several aspects of the flow within the auditorium and the assessment of their implications for the thermal comfort of the occupants. Work is in progress to obtain experimental data in the auditorium for the evaluation of the predictions.

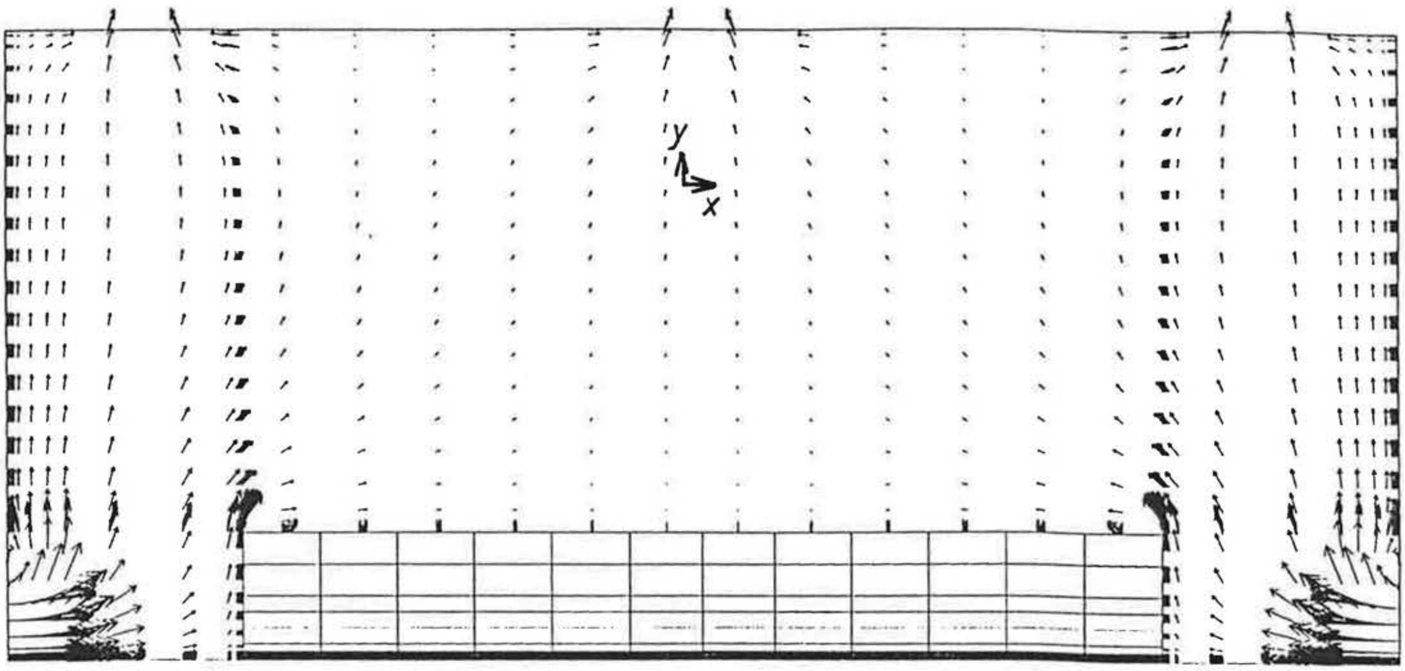
References

1 Chen Q *Indoor air flow, air quality and energy consumption of a building* PhD Dissertation, Delft University (1988)

2 Alamdari F, Hammond G P and Melo C Appropriate calculation methods for convective heat transfer from building surfaces *I. Chem. E. Symp. Series* **86** 1201–1211 (1984)

3 Vasquez B, Samano A and Yianneskis M The effect of air inlet location on the ventilation of an auditorium *Proc. Seminar on Computational fluid dynamics for the building services engineer, Institution of Mechanical Engineers, London, 26 November 1991* pp 56–66 (1991)

4 Hjertager B M and Magnussen B F Numerical prediction of three-dimensional turbulent buoyant flow in a ventilated room in *Heat Transfer and Turbulent Buoyant Convection II* pp 429–440 (Washington



⟶ : 8.79 m/s.   Min : 3.6824E-02   Max : 6.3113E+00

Figure 13 Velocity vector distribution, Re = 10^5

DC: Hemisphere) (1977)

5 Timmons P, Albright D Furry B and Torrance E Experimental and numerical study of air movement in slot-ventilated enclosures *ASHRAE Trans.* **86**(11) 221–240 (1980)

6 Etheridge D W and Nolan J A Ventilation measurements at model scale in turbulent flow *Building and Environment* **14**(1) 53–64 (1979)

7 Chen Q, Vander Kooi J and Meyers A Measurements and computations of ventilation efficiency and temperature efficiency in a ventilated room *Energy and Buildings* **12** 85–99 (1988)

8 Bauman F, Gadgil A, Kammerud R, Altmayer E and Nansteel M *Convective heat transfer in buildings: Recent research results* Lawrence Berkeley Laboratory Report **LBL-13883** (Berkeley, CA: Lawrence Berkeley Laboratory) (1983)

9 Gadgil A J *On convective heat transfer in building energy analysis* PhD thesis, University of California and Lawrence Berkeley Laboratory Report **LBL-10900** (Berkeley, CA: Lawrence Berkeley Laboratory) (1980)

10 Jones P and O'Sullivan P Modelling of air flow patterns in large single volume spaces *Proc. SERC Workshop on Developments in Building Simulation Programs, Loughborough University* (1985)

11 Markatos N C and Pericleous K A Laminar and turbulent natural convection in an enclosed cavity *Int. J. Heat and Mass Transfer* **27**(4) 755–772 (1984)

12 Markatos N C, Malin M R and Cox G Mathematical modelling of buoyancy-induced smoke flow in enclosures *Int. J. Heat Mass Transfer* **25**(1) 63–75 (1982)

13 Markatos N C and Cox G Hydrodynamics and heat transfer in enclosures containing a fire source *Physicochemical Hydrodynamics* **5**(1) 53–66 (1984)

14 Hoffman N A and Galea E R An extension of the fire-field modelling technique to include fire-sprinkler interaction - I: The mathematical basis *Int. J. Heat Mass Transfer* **36**(6) 1435–1444 (1993)

15 Hoffman N A and Galea E R An extension of the fire-field modelling

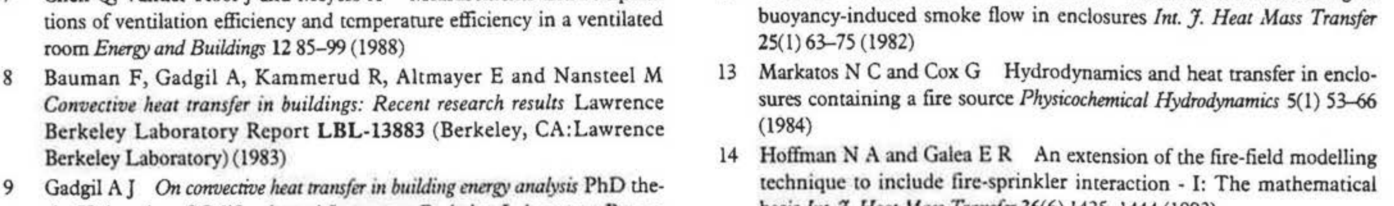


Figure 14 Isotherms, Re = 10^5: Contour values in °C; T_{min} = 22.0°C; T_{max} = 31.46°C

technique to include fire-sprinkler interaction - II: The simulations *Int. J. Heat Mass Transfer* **36**(6) 1445–1457 (1993)

16 Chen Q, Suter P and Moser A A database for assessing indoor air flow, air quality and draft risk *ASHRAE Trans.* **9**(2) 150–163 (1991)

17 McGuirk J J and Whittle G E Calculation of buoyant air movement in buildings-proposals for a numerical benchmark test case *Proc. Seminar Computational fluid dynamics for the building services engineer, Institution of Mechanical Engineers, London, 26 November 1991* pp 13–32 (1991)

18 Gray D D and Giorgini A The validity of the Boussinesq approximation for liquids and gases *Int. J. Heat Mass Transfer* **19**(3) 545–551 (1980)

19 Vazquez B BSc Thesis, Chemical Faculty, University of Mexico, Mexico (1986)

20 Rosten H I and Spalding D B *The PHOENICS equations* Report **TR/99** (London: CHAM Ltd) (1987)

21 *ASHRAE Handbook: Fundamentals* (Atlanta, GA: American Society of Heating, Refrigeration and Air-Conditioning Engineers) (1981)

22 Gomiciaga R MSc Thesis, University of Mexico, Mexico (1990)

23 Ozoe H, Mouri A, Ohmuro M, Churchil S W and Lior N Numerical calculations of laminar and turbulent natural convection in water in rectangular channels heated and cooled isothermally on the opposing vertical walls *Int. J. Heat Mass Transfer* **28**(1) 125–138 (1985)

24 Vazquez B MSc Thesis, University of Mexico, Mexico (1990)