5710

The effect of air inlet location on the ventilation of an auditorium

B VAZQUEZ, BSc, MSc, D SAMANO, M YIANNESKIS, BSc, MSc, PhD,
Mechanical Engineering Department, Kings College London, UK.

ABSTRACT

A numerical study of the ventilation patterns inside an auditorium of the University of Mexico was performed. A general computer program was used to solve the conservation equations of mass, momentum and energy. Simulations of the two-dimensional heat transfer and fluid flow processes within a cross-section of the auditorium were carried out. Data were obtained in the form of air velocity and temperature distributions. Several cases were studied to investigate the effect of the variation of geometrical characteristics of the auditorium; results are presented for different locations of the air inlets on the auditorium walls. The location of the inlets was found to affect significantly the flow pattern and temperature field and thus to influence directly the thermal comfort conditions inside the auditorium.

The heat generated by the occupants, the effect of weather conditions on the temperature of the walls and on the incoming air stream and the operation of the ventilation fans were all modelled and/or accounted for and the location of the air inlets on the auditorium walls which provides the optimum thermal comfort conditions for the auditorium occupants was determined.

INTRODUCTION

The main interest in the study of convection phenomena in building ventilation is to understand the flow patterns, the heat transfer processes and the temperature fields involved, in order to optimize the design of the equipment employed, the location of heating elements or ventilation openings, etc. A primary objective is the provision of appropriate thermal comfort conditions within a building. Another important consideration is the minimisation of the energy expended to heat or cool a building and it is necessary to establish strategies for saving energy. However, due to the complexity of the governing equations, analytical solutions are often not possible and the understanding of convection flows has therefore been restricted. However, the equations can be solved using numerical methods.

The thermal comfort of the persons inside a building is affected directly by the air speed and temperature gradients, and it is very important to predict the flow patterns and temperature distribution accurately. 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 of the University of Mexico, and to estimate the effect of changes in the size, number, type and position of the air inlets and outlets on the velocity and temperature distributions and the comfort of the occupants.

A number of studies of the convection heat transfer processes inside buildings have been reported. However the authors are not aware of any work with similar conditions to those found in the University auditorium described below. Buildings of various shapes and sizes have been investigated but none of sufficient similarity to allow a comparison with the building under study. The numerical studies reported have been concerned with both two- and three-dimensional laminar and turbulent flows. The k-ε model has been used most extensively in the prediction of 3-dimensional turbulent flows. Experimental studies for laminar and turbulent flows in buildings with mechanical air conditioning systems have also been reported. Some of the research reported has been concerned with the determination of the heat transfer coefficient on vertical walls inside buildings. A major part of the research effort has aimed to establish the flow patterns inside rooms with the purpose of identifying the conditions that provide a uniform distribution of the air temperature.

Hertager and Magnussen [1], Timmons et al [2] and Etheridge and Nolan [3] carried out numerical studies and found that the near-wall zone, which includes the boundary layer, was not adequately modelled; this was attributed partly to the size of the mesh used in that zone and partly to limitations of the models. In experimental studies (see, for example, Qinyang et al [4], Bauman et al [5], Gadgil [6]) full-scale chambers have been used under well-controlled conditions. Measurements obtained in these studies have shown good agreement with numerical results. Qinyang et al [4] and Timmons et al [2] 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 [5] 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.

DESCRIPTION OF PHENOMENA.

In the present study the air distribution, heat transfer convection and temperature gradients inside a University of Mexico auditorium located at Temixco, Morelos, were calculated across a two-dimensional cross-section, which corresponds to the section of the auditorium containing the second row of seats. The thermophysical characteristics of air and the boundary conditions were kept identical to those measured inside the auditorium.

The section simulated is located under three ventilation fans: these are aeolic (wind-driven) turbines. The simulation was carried out for two-dimensional, steady-state forced convection. It was considered that the Boussinesq approximation is valid, so that all physical properties of the fluid remain constant, except for the density in the body force term. The Boussinesq approximation has been shown to be valid for temperature gradients inside a room of less than 28.6°C (Gray and Giorgini, [7]).

Figure 1 shows the scheme domain and the rectangular cross-section of the auditorium. The vertical boundaries represent the east and west walls and the horizontal boundaries represent the floor and ceiling. In the figure one inlet is located in the lower part of each of the lateral walls, and three outlets are located on the ceiling. The boundary at the bottom is divided into three zones, a central one of width "c" and height "m" that represents the 13 seats occupied by persons, and two zones next to the walls representing the aisles. The heat generated by the occupants in the central zone was modelled by employing a temperature value of 33°C as the skin temperature and a body surface area of 1 m^2 per person.

The temperature of the boundaries (walls, ceiling and floor) and the ambient air temperature were given the following values: the east and west walls 26°C, the ceiling 27°C, the floor 26°C and the air entering through the inlets 22°C. These values were obtained from measurements (Vazquez [8]) and therefore correspond to temperatures resulting from actual weather conditions.

The equations governing the phenomena of forced convection inside the auditorium are:

Continuity

$$\rho_o \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = 0$$

Momentum in the x direction

$$\left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = - \frac{1}{\rho_o} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)$$

Momentum in the y direction

$$\left(u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = - \frac{1}{\rho_o} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) - \frac{(\rho - \rho_o)\, g_y}{\rho_o}$$

Energy

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$

where u, v, T and P are the velocities in the x- and y-directions, the temperature and the pressure respectively. The physical properties are the density ρ, the kinematic viscosity ν, the thermal diffusivity α, and the gravitational acceleration g. The subscript "0" indicates the reference value.

The boundary conditions set were as listed below. The parameters e, f, n, L, H, c, l and m are defined in Figure 1.

1.- Vertical walls east and west:

$x = 0,\ n \le y \le L;$

a) velocity u,v = 0
b) temperature T = 26°C

$x = H,\ n \le y \le L;$

a) velocity u,v = 0
b) temperature T = 26°C

2.- Aisles:

$0 \le x \le d,\ y = 0;$

a) velocity u,v = 0
b) temperature T = 26°C

$d + c \le x \le H,\ y = 0;$

a) velocity u,v = 0
b) temperature T = 26°C

3.- Heat generation zone (seats)

$d \leq x \leq H - d$, $y = m$;
a) velocity $u,v = 0$
b) temperature $T = 33°C$

$x = d$, $0 < y < m$;
a) velocity $u,v = 0$
b) temperature $T = 33°C$

$x = H - d$, $0 < y < m$;
a) velocity $u,v = 0$
b) temperature $T = 33°C$

4.- Air inlets on laterals walls:

$x = 0$, $0 < y < n$; gauge pressure = 0
$x = H$, $0 < y < n$; gauge pressure = 0

5.- Ceiling:

$0 < x \leq e$, $y = L$;
a) velocity $u,v = 0$
b) temperature $T = 27°C$

$e + f \leq x \leq e + f + l$, $y = L$;
a) velocity $u,v = 0$
b) temperature $T = 27°C$

$e + 2f + l \leq x \leq H - (e + f)$, $y = L$;
a) velocity $u,v = 0$
b) temperature $T = 27°C$

$e + 3f + 2l \leq x \leq H$, $y = L$;
a) velocity $u,v = 0$
b) temperature $T = 27°C$

6.- Ceiling air outlets:

a) $e < x < H - (e + 2f + 2l)$, $y = L$;
$u = 0$, $v = V_m$

b) $e + f + l < x < H - (e + f + l)$, $y = L$;
$u = 0$, $v = V_m$

c) $e + 2f + 2l < x < H - e$, $y = L$;
$u = 0$, $v = V_m$

where $V_m = m/\rho * A$ is the average flow velocity at the air outlet section.

COMPUTATIONAL METHOD.

The PHOENICS computer code (version 1.14) was employed for the predictions presented in this paper. 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 value used for the false transient for both components of velocity and the temperature varied between cases. The initial velocity values were all zero and the starting value for the temperature field was 22°C. All initial conditions were identical for all cases. A irregular mesh of 46 x 36 volumes (in the x- and y-directions respectively) was used for the simulation of all the cases studied.

RESULTS AND DISCUSSION.

The air is extracted through the three outlets located at the ceiling, each of which is connected to an aeolic (wind-driven) turbine. The movement of air inside the auditorium is produced by the interaction of the forced convection produced by the turbines and the buoyancy force generated by the simulated body heat of the occupants and by the temperature gradients caused by the inlet and boundary conditions. The velocity used to extract the air inside the auditorium varies in accordance with the number of air changes per hour necessary to maintain the specified quality of air inside auditoria [9]. The air velocity in the outlets is 0.5 m/s: the corresponding flow rate and Reynolds number were 1.52 m^3/s and 86,400 respectively.

It might be expected that the number, location and size of air inlets affect the flow patterns inside the auditorium and, as a result, the air velocity and temperature around the occupants is different in each case. In this part of the research programme, the number and size of air outlets were kept constant and only the height of air inlets was modified. The heights at which the air inlets were located for the seven cases studied are shown in Table 1 below.

Table 1: Location of air inlets

CASE	NODE NUMBER	DISTANCE OF AIR INLET BASE FROM FLOOR (m)
1	1-9	0.00
2	10-11	0.46
3	12-19	0.93
4	20-21	1.39
5	22-23	1.86
6	24-25	2.32
7	26-27	2.79

Figures 2 through 8 below show the behaviour of the air flow within the auditorium for the 7 cases studied. Figures 2a-8a show the velocity distribution of the air motion produced in the auditorium. The corresponding room air temperature distributions are shown in Figures 2b through 8b.

In Figure 2a the air entering the auditorium through the air inlets is deflected by the seats, resulting in the generation of a strong flow vertically upwards near the seat/aisle boundary. A recirculation region is formed over the aisle, with the air near the walls flowing downwards. In all, four recirculation regions are formed over the cross-section. Two of these are located over the seats and occupy most of the auditorium. The smallest air velocities are found in the central region, where the flow is directed downwards. Figure 2b shows that temperature gradients are present in the region above the seats and that a local accumulation of heat is formed over the seats in the centre of the auditorium.

When the height of the vents is increased (case 2, Figure 3a), the incoming air is not deflected as in the first case. As a result, two large recirculation zones are formed near the walls. The air velocity over the seats is slightly higher than in case 1. Six recirculation regions are formed across the auditorium: the two mentioned above, two more above the seats and a further two zones near the floor of the aisles. The isotherms of Figure 3b show small gradients near the inlets and also very near the seat/aisle boundary, while the temperature is uniform in the rest of the domain.

The results for the third case (Figure 4a) show the air entering the auditorium at a level exactly above the seats, aiding thus the discharge of the body heat generated by seated persons. The air flows over the seats toward the centre. In addition, the suction produced by the aeolic turbines induces an air flow above the seats over most of the auditorium towards the outlets. The incoming air from each vent forms two recirculation regions: an upper one above the seats and a lower one in the aisles rotating in the opposite direction. The air stream from the inlet is divided into two parts in the area above the seats: the first one rising to the outlets located in the ceiling and the other forming the aforementioned recirculation zone, through which air is entrained into the incoming jet. Likewise, part of the air flow from the lower recirculation zone in the aisles (between the seats and the walls) is entrained into the main stream entering through the air inlets. The flow pattern described above was also indicated by the numerical results of Bauman et al [5] and the experimental results of Givoni [10] which were obtained in a wind tunnel. The temperature field is very uniform over the whole of the cross-section, as shown in Figure 4b. The temperature around the seats is smaller than those in cases 1 and 2.

For case number 4 (Figure 5a) the air entering through the inlets is directed over the seats and the air velocity distribution in that region is not dissimilar to that of case number 3. The overall flow pattern may be similar to that of case 3, but the velocities are in general lower over most of the auditorium. The temperature field (Figure 5b) also shows a pattern similar to that found for case 3, except near the aisle floor, where the temperature is even more uniform in this case.

In Figure 6a two strong recirculation regions are formed over the seats because of the airstream entering the auditorium. The direction of the flow above the seats is different to those in the last three cases. The incoming air flow from each inlet is divided in two main currents. One flows toward the centre of the auditorium, resulting in the formation of a recirculation above the seats. The second current moves toward the lateral air outlet and part of this flow is recirculated toward the interior, forming a wall-jet type of flow over the walls. Only a small part of this airflow leaves the auditorium through the central outlet on the ceiling. The recirculations formed in the aisle rotate in the opposite direction to that in the other cases analyzed; this is mainly due to the reversal of the direction of the flow over the seats. The temperature field has also changed significantly in comparison to the previous cases. This is to a large extent caused by the fact that most of the incoming air is directly evacuated by the outlets and only a small part of the flow is recirculated. As a result, strong temperature gradients are formed near the seats and the aisles.

The results for case number 6 (Figures 7a and 7b) show a similar flow behaviour to that of case number 5. The main difference between them is found in the size of the central recirculation zones, their influence extending further toward the walls in this case. The temperature field for this case also shows many similarities with that for case 5. However, the isotherms do not extend as far into the centre of the room in the present case.

The case 7 results (Figure 8a) show that the magnitudes of the vectors forming the central recirculations have decreased, and a large amount of the air flow entering trough the inlets leaves the auditorium through the lateral outlets, without any significant beneficial effect on the air inside the auditorium. An accumulation of temperature is found in the aisle/seat boundary region (Figure 8b) and the isotherm distribution is very similar to that for case 6.

In Figures 2a through 8a the formation of recirculations, directly related with the location of the air inlets, outlets and seats, was shown. In the first two cases the velocity direction over the seats is from the centre of the domain towards the walls, while for cases 3 and 4 it is reversed (from the walls to the centre), and finally for cases 5, 6 and 7 the flow direction is again from the centre to the walls. This behaviour means that in cases 3 and 4 the air that the occupants receive is coming directly from outside the room, and in the remaining cases the air is mixed inside the room before it reaches the occupants. The maximum value of the air velocity was found for case 3 (3.26 m/s); in this case the air inlet is located at a height similar to that of the seats.

The temperature contours inside the auditorium shown in Figures 2b through 8b, indicate that in cases 3 and 4 the temperature field is quite uniform, while in the other cases non-uniform temperature fields were predicted. This behaviour is directly related to the flow patterns discussed above. It is possible to conclude that cases 3 and 4 are the most favourable in order to obtain thermal comfort conditions inside the auditorium.

The results obtained provide useful information for the understanding of the heat transfer and fluid flow processes in building ventilation. The findings clearly indicate that CFD methods can yield essential information which can significantly aid the design of building ventilation and air conditioning.

CONCLUDING REMARKS

1.- Predictions of the flow pattern and temperature distributions inside an auditorium were carried out for seven different air inlet heights.

2.- The results revealed that the flow pattern and temperature distribution are strongly influenced by the location of the air inlets.

3.- The numerical method provided important information on how the air speed and temperature gradients generated within the building affect the thermal comfort of occupants. An air inlet height of 0.93-1.35 m (cases 3 and 4) was found to be the most favourable for obtaining thermal comfort conditions.

REFERENCES

[1] Hertager B.M. and Magnussen B.F. (1977), Numerical prediction of three-dimensional turbulent buoyant flow in a ventilated room, Heat Transfer and Turbulent Buoyant Convection, II, 429-440.

[2] Timmons P., Albright D., Furry B. and Torrance E. (1980), Experimental and numerical study of air movement in slot-ventilated enclosures, ASHRAE Transactions Semi-annual Meeting, L.A. California, 86, Part 11, 221-240.

[3] Etheridge D.W. and Nolan J.A. (1979), Ventilation measurements at model scale in turbulent flow, Building and Environment, 14, 53-64.

[4] Qinyang C., Vander Kooi J., Meyers A., (1988), Measurements and computations of ventilation efficiency and temperature efficiency in a ventilated room, Energy and Buildings, 126, 85-99.

[5] Bauman F., Gadgil A., Kammerud R., Altmayer E. and Nansteel M. (1983), Convective heat transfer in buildings: Recent research results, Lawrence Berkeley Laboratory Report, LBL-13883,15.

[6] Gadgil A.J. (1980), On convective heat transfer in building energy analysis, Ph.D. Thesis and Report LBL- 10900, University of California, U.S.A.

[7] Gray D.D. and Giorgini A., (1980) The validity of the Boussinesq approximation for liquids and gases, Int, J. Heat Mass Transfer, 19, 545-551.

[8] Vazquez B. (1986) Modelo para establecer el comportamiento termico de un auditorio, BSc Thesis, Facultad de Quimica UNAM, Mexico.

[9] ASHRAE (1981) Fundamentals Handbook, Atlanta U.S.A.

[10] Givoni, (1969) Cited by [5]

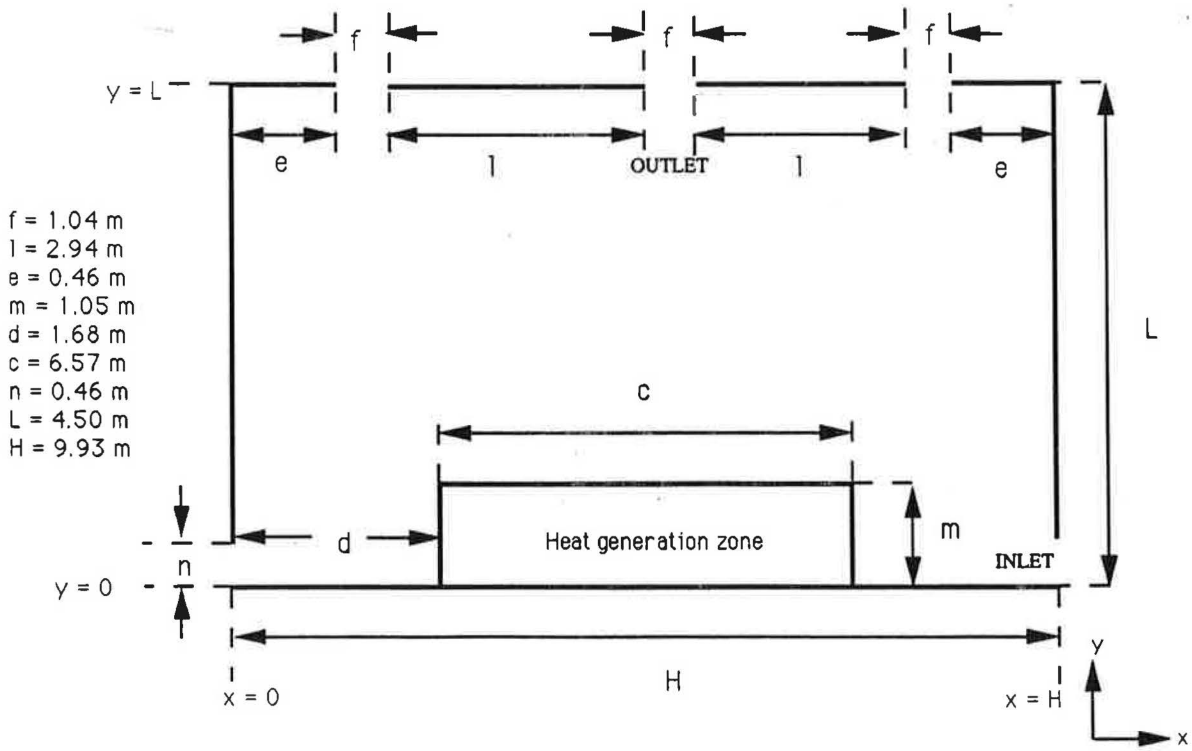


Fig 1 Numerical scheme domain

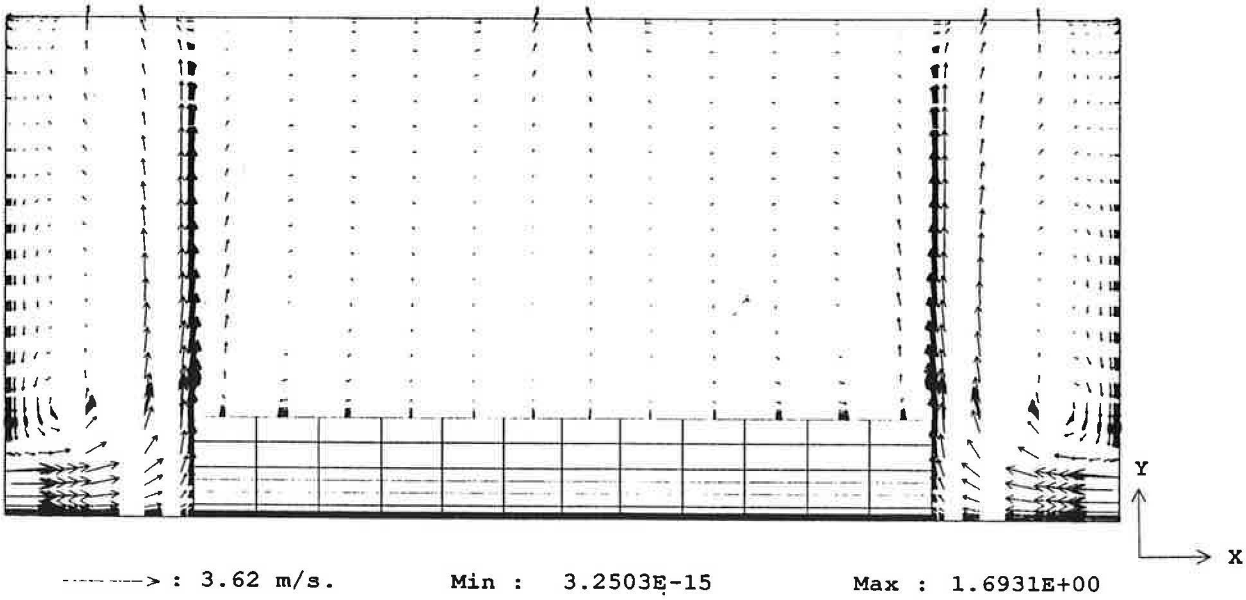


Fig 2 (a) Velocity vector distribution for 0.0m air inlet location;
Re = 8.62×10^4, Ra = 1.85×10^{10}, case 1

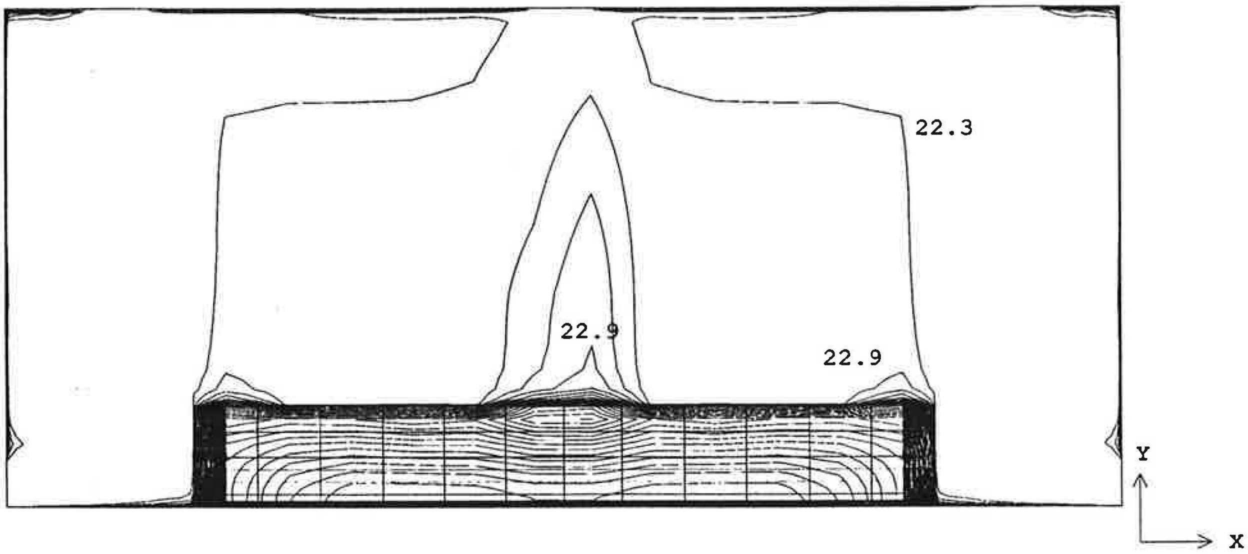


Fig 2 (b) Isotherms for 0.0m air inlet location; Re = 8.62×10^4,
Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 32.07 °C, case 1

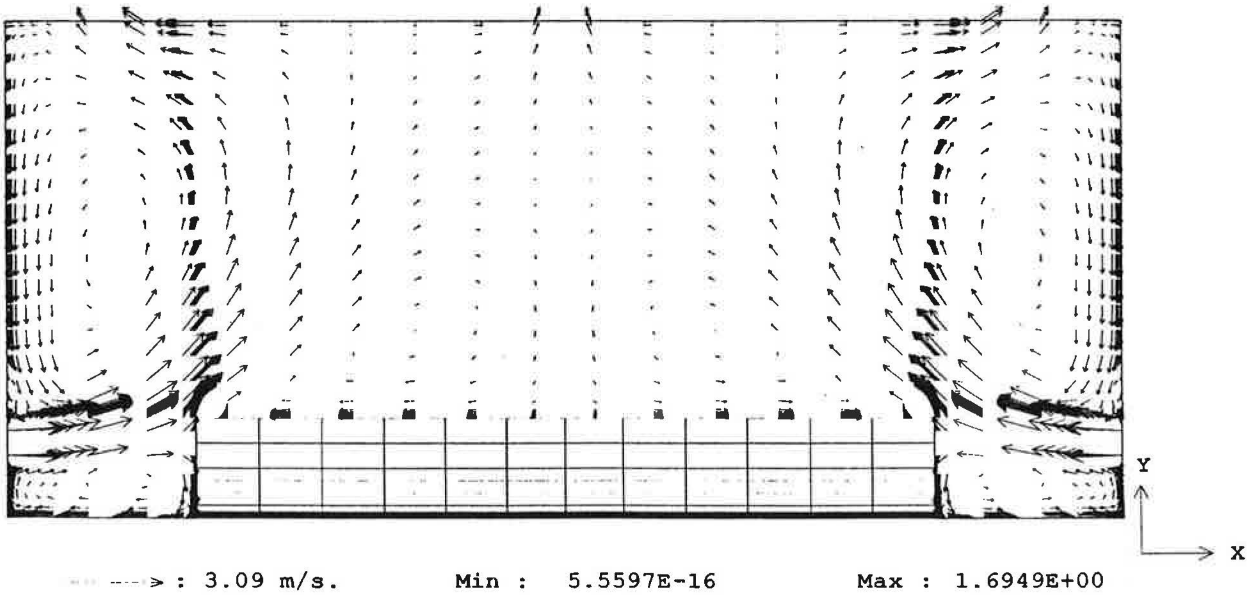


Fig 3 (a) Velocity vector distribution for 0.46 m air inlet location;
Re = 8.62×10^4, Ra = 1.85×10^{10}, case 2

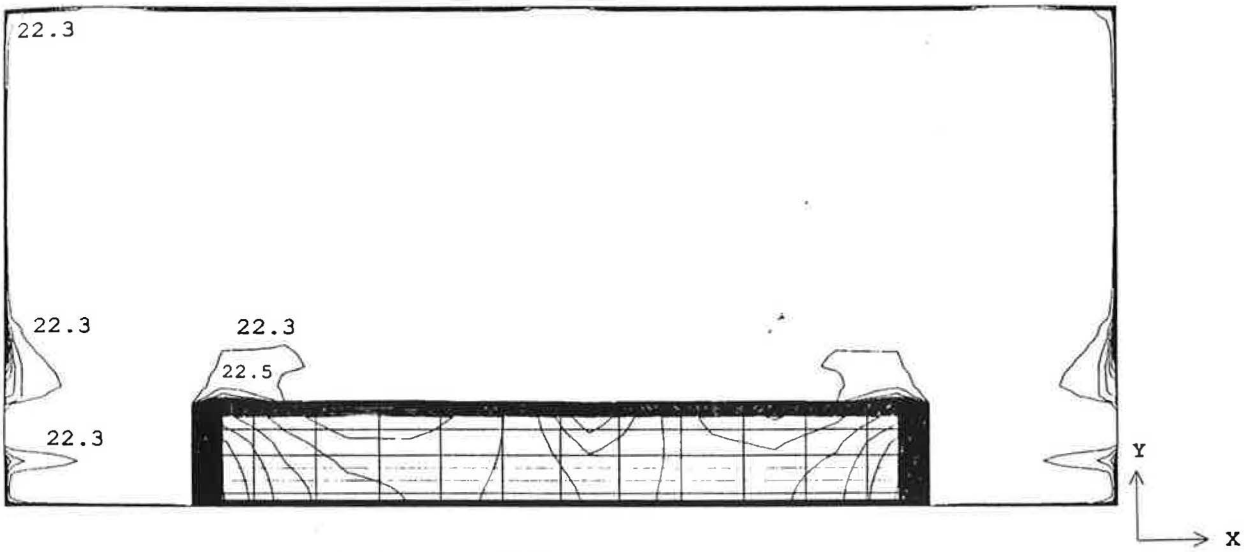


Fig 3 (b) Isotherms for 0.46 m air inlet location; Re - 8.62×10^4, Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 27.85 °C, case 2

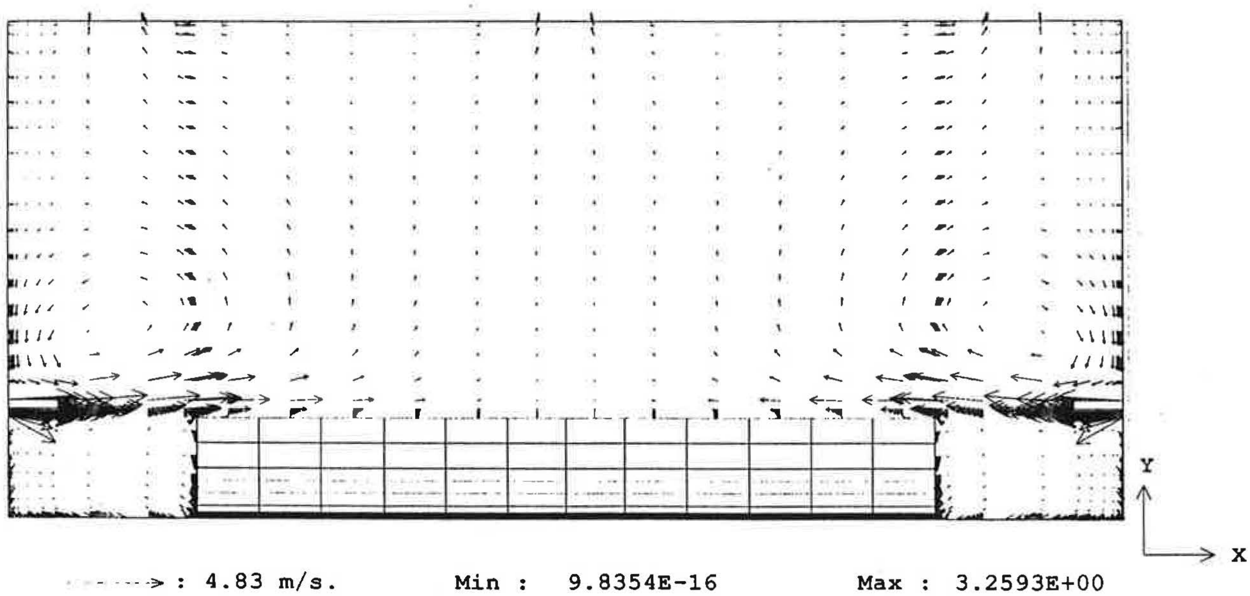


Fig 4 (a) Velocity vector distribution for 0.93 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, case 3

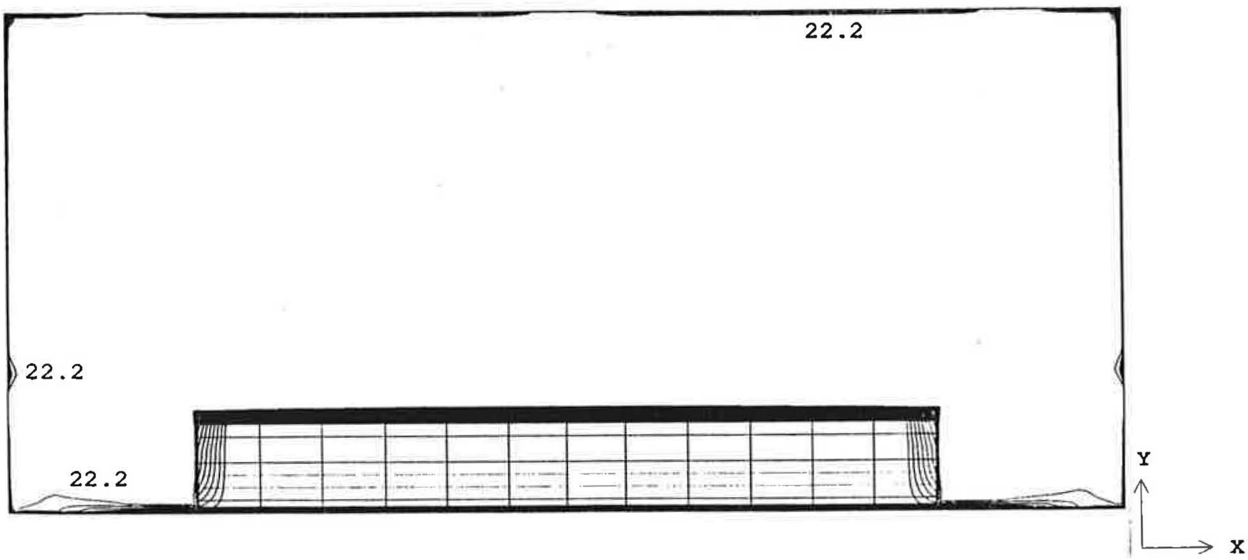
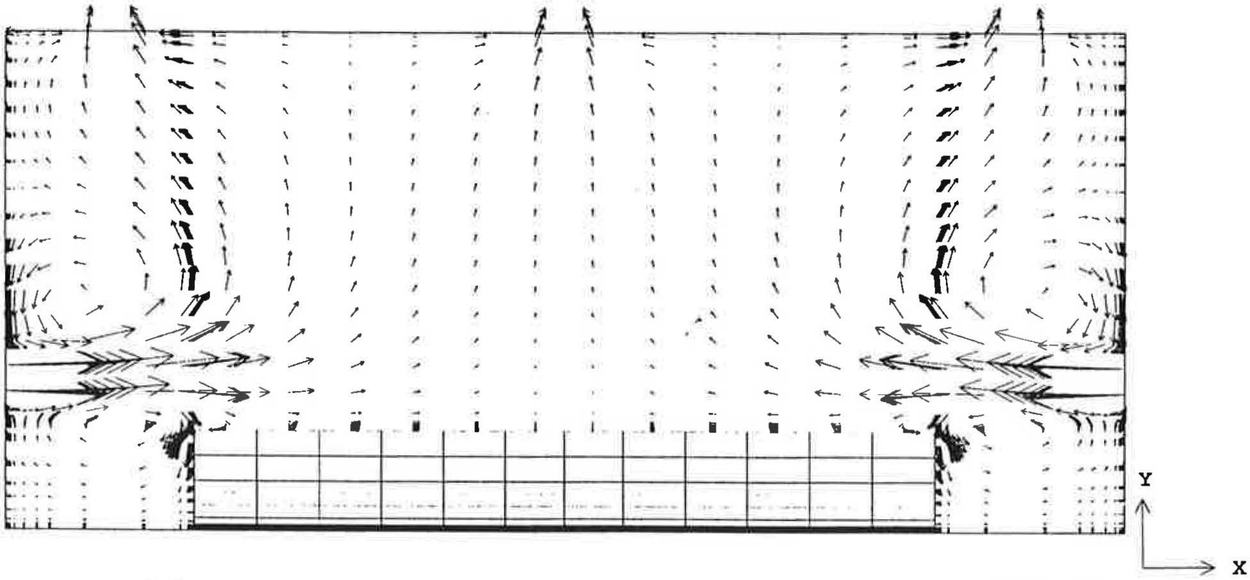


Fig 4 (b) Isotherms for 0.93 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 27.85 °C, case 3



---> : 1.70 m/s.   Min : 9.4939E-16   Max : 1.6605E+00

Fig 5 (a) Velocity vector distribution for 1.39 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, case 4

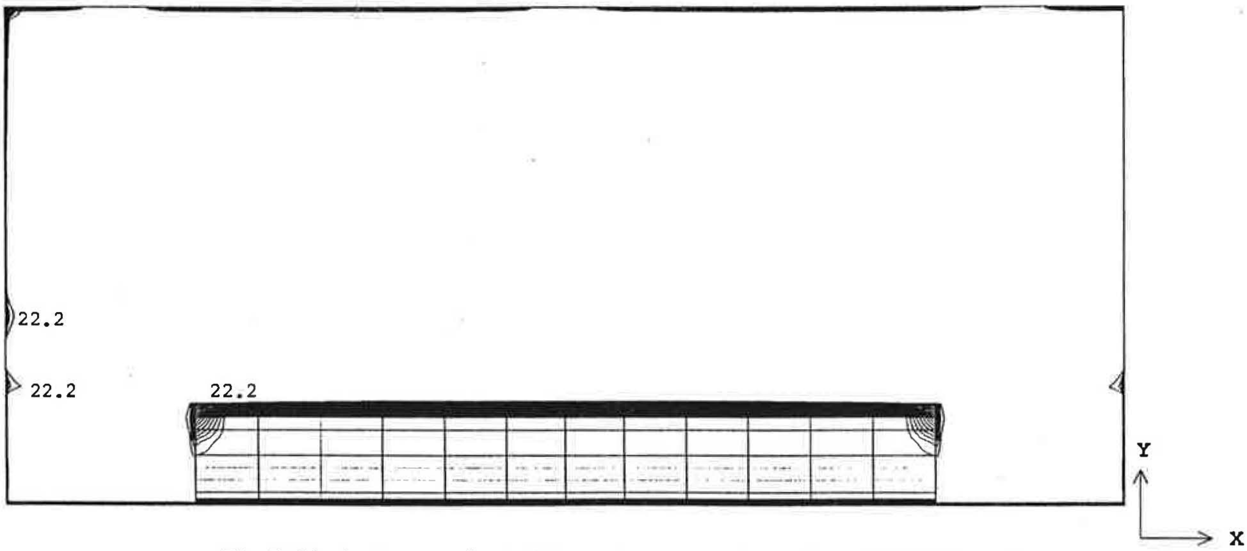
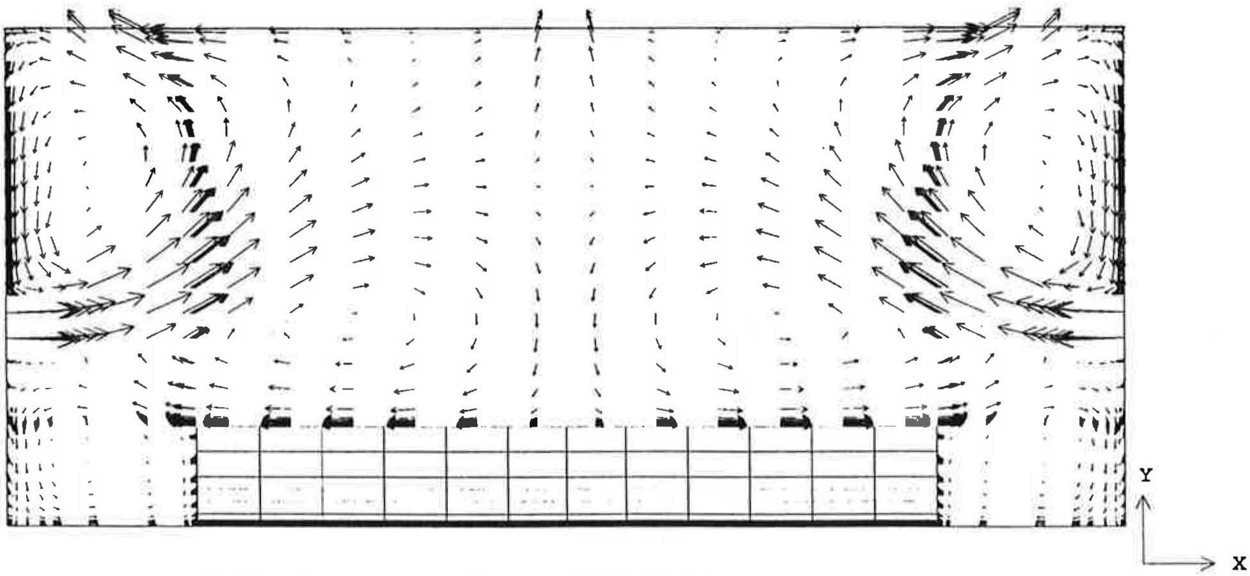


Fig 5 (b) Isotherms for 1.39 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 28.28 °C, case 4



---> : 2.37 m/s.   Min : 8.7050E-16   Max : 1.6740E+00

Fig 6 (a) Velocity vector distribution for 1.86 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, case 5

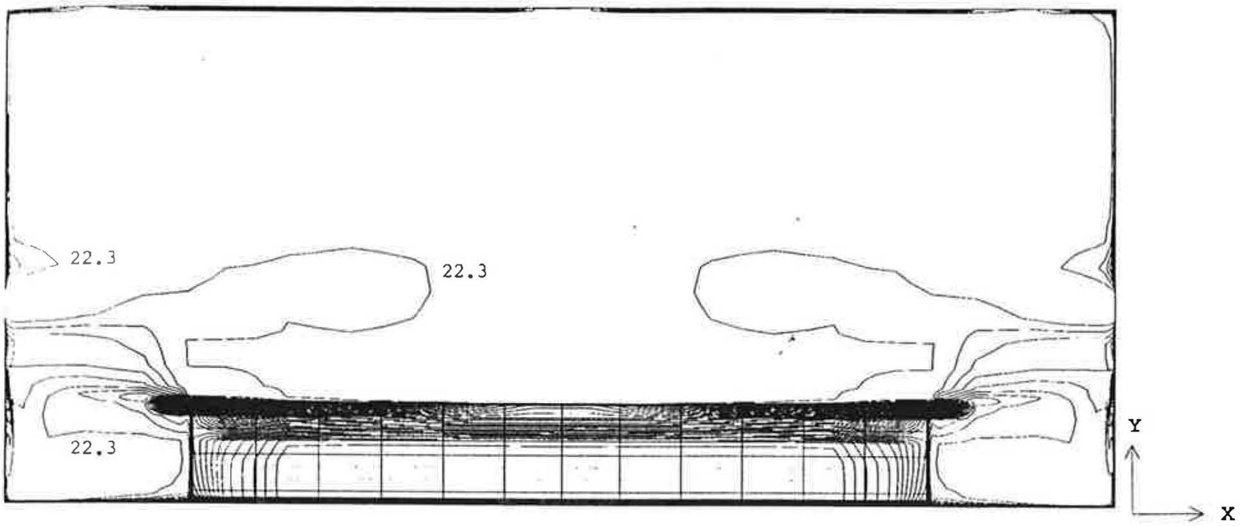


Fig 6 (b) Isotherms for 1.86 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 31.14 °C, case 5

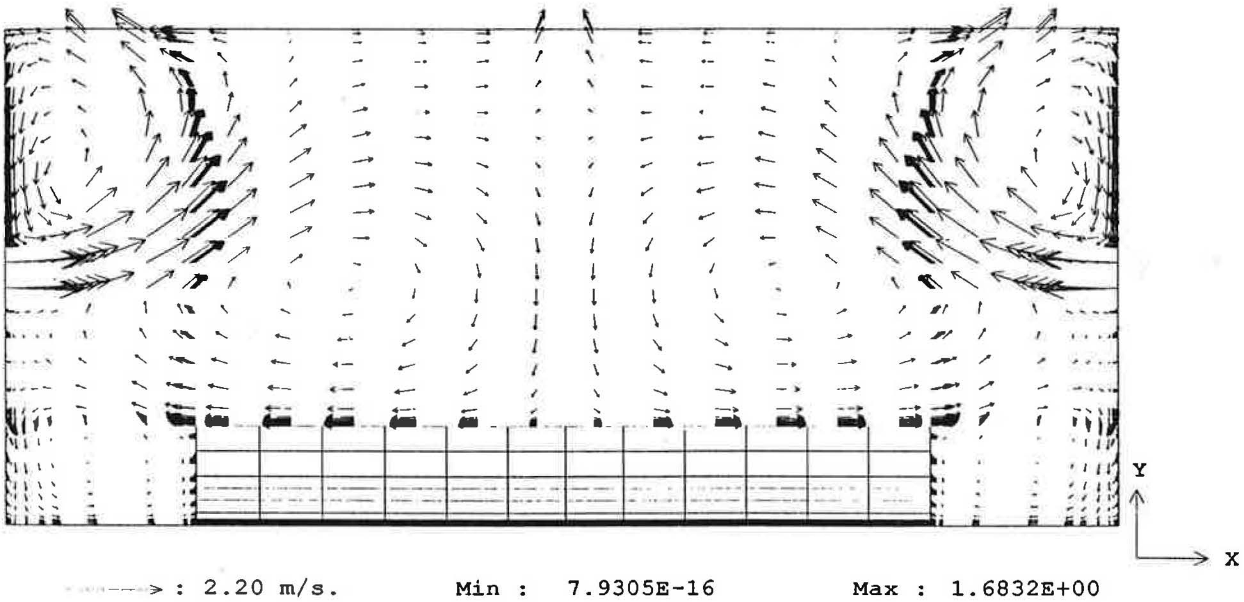


Fig 7 (a) Velocity vector distribution for 2.32 m inlet location; Re = 8.62x104, Ra = 1.85x1010, case 6

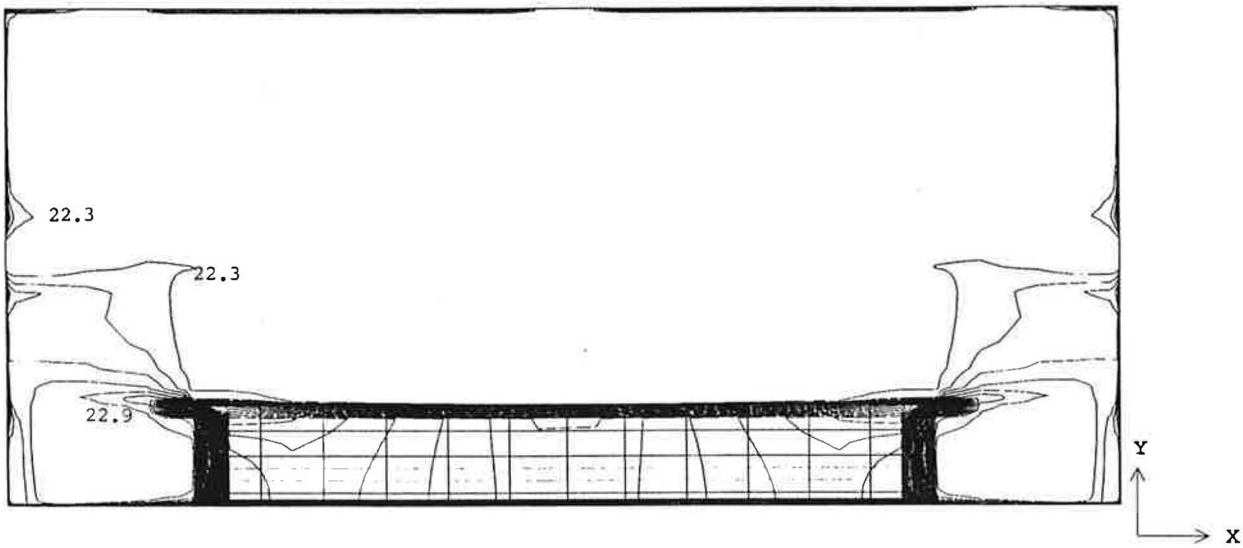


Fig 7 (b) isotherms for 2.32 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 30.99 °C, case 6

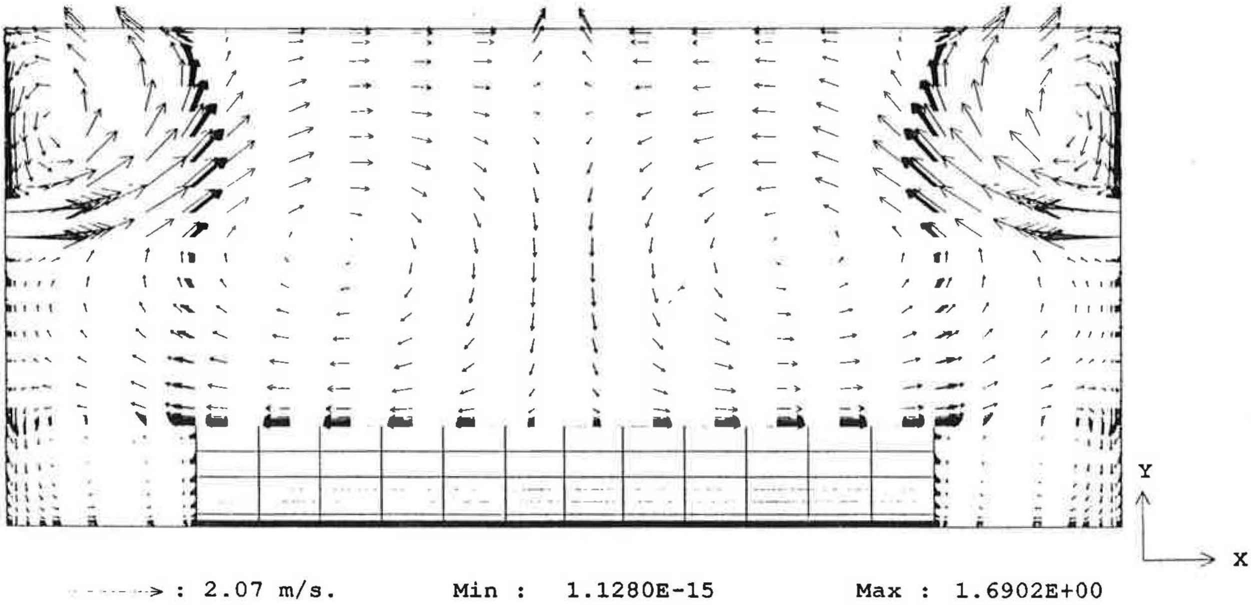


Fig 8 (a) Velocity vector distribution for 2.79 m air inlet location; Re = 8.62x104, Ra = 1.85x1010, case 7

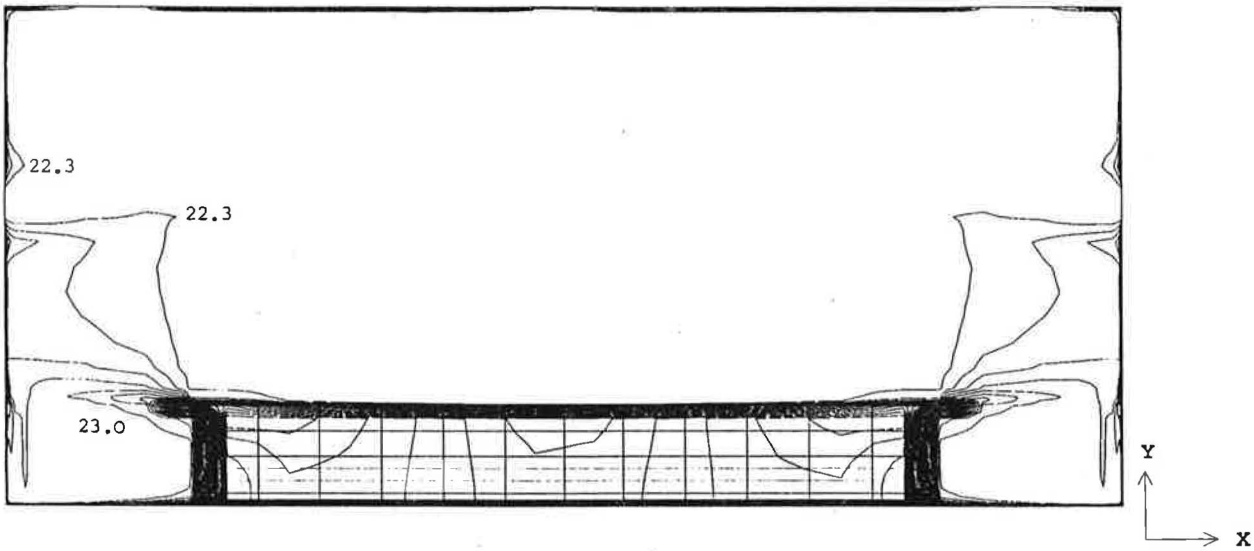


Fig 8 (b) Isotherms for 2.39 m air inlet location; Re = 8.62×10^4, Ra = 1.85×10^{10}, T_{min}=22.0°C, T_{max} = 30.99 °C, case