Pergamon

Int. Comm. Heat Mass Transfer, Vol. 29, No. 3, pp. 345-354, 2002

0735–1933/02/$–see front matter

PII: S0735-1933(02)00324-X

NATURAL CONVECTION IN A TWO-DIMENSIONAL ENCLOSURE HEATED SYMMETRICALLY FROM BOTH SIDES

S. P. Das, S. Chakraborty and P. Dutta
Department of Mechanical Engineering
Indian Institute of Science
Bangalore 560 012, INDIA

(Communicated by A.R. Balakrishnan)

ABSTRACT

Experimental and numerical studies are performed for transient and steady natural convection phenomena in a two-dimensional cavity heated symmetrically from both sides with a uniform heat flux. In the experimental work, temperatures are measured with thermocouples located along the side walls and also along the centreline of the tank. Flow visualisation is performed by video recording the flow using a fluorescent die illuminated by ultraviolet light. It is observed that this situation leads to a well-mixed layer at the top marked by multiple loops, below which the fluid is thermally stratified. The flow pattern in the cavity is found to be significantly different from the well-studied phenomena of differentially heated enclosures. Numerical modelling is also performed using a finite volume method. The measured temperatures and flow visualisation results show a good agreement with the numerically generated results.

Introduction

Study of natural convection in enclosures has many engineering applications such as in building technology, cooling of electronic equipment, solar collectors, materials processing, manufacturing, and so on. The importance of this subject in several diverse fields of science and engineering is amply reflected by the vast amount of research effort dedicated to this topic during the past few decades [1, 2]. Depending on the geometry and orientation of the enclosure, the natural convection phenomena can be classified into two broad classes [3]:

(a) bottom heated enclosures, which lead to the well-studied classical subject of Benard flow [4], and (b) side heated enclosures, which have received considerable attention in the past couple of decades.

Most of the studies related to side heated cavities involve differential heating i.e., one side is heated while the opposite side is cooled. The heated side is either maintained at a certain temperature or subjected to a given heat flux. Several experimental and computational studies have been reported in the literature concerning transient and/or steady state natural convection in differentially side heated cavities, such as those reported in Simpkins and Dudderar [5], Fusegiet et al. [6], Yewell et al. [7], Schladow et al. [8], and Schladow [9]. On the other hand, studies on symmetrically heated cavities (i.e. heated equally from both sides) have largely been ignored, to the best of our knowledge. However, this class of problems has several important engineering applications such as in cryogenic storage tanks, electrical and electronic equipment, materials processing, etc. Although some specialised studies with similar boundary conditions with the objective of investigating thermal stratification in cryogenic storage tanks have been reported (Lin and Hassan [10]; Tanyun et al. [11]), a systematic study of the natural convection phenomenon in such a situation is yet to be found in the literature.

The present work is a preliminary attempt to study transient natural convection phenomena in a two-dimensional cavity heated symmetrically from both sides. With the top of the cavity left open to atmosphere and bottom wall insulated, this situation leads to a well-mixed layer at the top, below which the fluid gets thermally stratified. The flow physics and the resulting convection heat transfer can be significantly different from the well-studied phenomena of differentially heated enclosures. This has motivated us to perform an experimental and numerical study of the this side-heated cavity, so as to understand the heat transfer and fluid flow phenomenon leading to thermal stratification inside the enclosure.

Mathematical Formulation

The problem considered for the study is shown schematically in Fig. 1. The enclosure is filled with water at room temperature at time $t=0$. At $t>0$, the two side walls are subjected to a uniform heat flux. The bottom wall is adiabatic while the top is left open to the atmosphere. The main advantage of an open top is that it permits us flow visualisation by shining a sheet of light from the top.

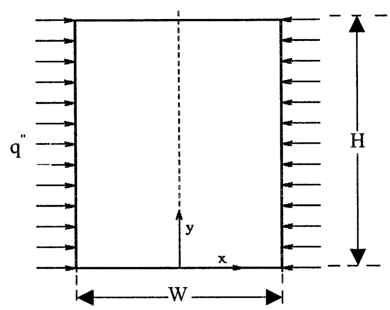

FIG. 1
Schematic diagram of the physical model

The fluid is assumed to be incompressible, although buoyancy effects are incorporated by invoking the Boussinesq approximation. Also, viscous dissipation is neglected in the energy balance equation. With these assumptions, the governing equations in a Cartesian co-ordinate system are as follows:

Continuity:
$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

x-momentum:
$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \tag{2}$$

y-momentum:
$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right]+\rho g\beta\left(T-T_{ref}\right) \tag{3}$$

Energy:
$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right] \tag{4}$$

where ρ is the density, ν is the kinematic viscosity, α is the thermal diffusivity, and β is the volumetric coefficient of thermal expansion of the fluid, T is the temperature, and u, v are the velocities in x, y directions respectively.

Boundary conditions: Side walls: $u=v=0,\quad -k\frac{\partial T}{\partial x}=q''$ (W/m^2) (5)

Bottom: $$u = v = 0, \quad \frac{\partial T}{\partial x} = 0 \tag{6}$$

Top: $$v = 0, \quad \frac{\partial u}{\partial y} = 0, \quad -k\frac{\partial T}{\partial y} = h\ (T - T_\infty) \tag{7}$$

The initial conditions are: $u = v = 0$, and $T = T_\infty$ everywhere. Since the top of the cavity is left open to the atmosphere, the effective heat transfer coefficient, h, in eq. 7 includes the effect of convective as well as evaporative heat losses.

Experimental and Numerical Procedure

Experiments are performed so as to simulate and understand the thermal stratification in the side-heated cavity. The objective of the experiments is to study the transient temperature variation at various locations in the cavity, and also perform flow visualisation at various stages of the stratification process. The first objective deals with the quantification of thermal stratification through temperature measurements, while the latter objective is to study the fluid flow phenomenon associated with the stratification. Together, they can give some insight into the physics of the stratification phenomenon.

The experimental arrangement consists of a rectangular tank made of two brass plates and two glass plates forming the four walls (fig. 2). The tank is filled with water, which forms the fluid medium for our study. The top of the tank is open to atmosphere while the bottom is insulated. The brass plates are chrome plated to avoid corrosion by water. A uniform heat flux is introduced through each brass plate with the following arrangement. Nichrome wire is wound on a mica sheet, one side of which is insulated using an asbestos sheet and a plywood sheet while the other side is covered by another mica sheet and attached to the brass plate. The plane consisting of the vertical centrelines of the two brass plates forms our two-dimensional domain of study. For the purpose of flow visualisation, fluorescent dye is introduced in small quantities through 1.5 mm diameter stainless tubes at strategic locations along the centrelines of the brass plates.

Four thermocouples are attached along the centreline of one of the brass walls to record the wall temperatures, while another four are tied on a rod dipped in the middle of the tank to record the centreline temperature distribution. All thermocouples are connected to a data acquisition system to record the transient temperature variation at various locations.

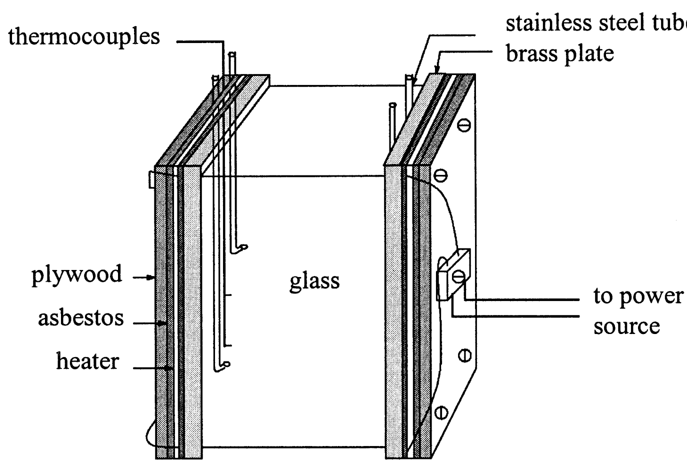

FIG. 2
A schematic diagram of the experimental arrangement

In the two-dimensional plane of study described above, the width, *W,* of the tank is 22 cm. The effective height of the cavity, *H*, can be controlled by filling it with water up to a desired level (maximum height is 33 cm). In this arrangement, the aspect ratio (*H/W*) can be varied by changing the water level in the tank. The maximum heat flux through the brass walls is designed at 1000 W/m^2. The modified Rayleigh number, Ra_H^*, based on heat flux, q'', can be calculated as:

$$Ra_H^* = \frac{g\beta H^4 q''}{\alpha \nu k} \tag{8}$$

Using water properties at an average temperature of 310 K, with a heat flux of 500 W/m^2 and an aspect ratio of 1:1, Ra_H^*=6.1 x 10^{10}. The critical value of modified Rayleigh number, Ra_H^*, for transition to turbulence in a side heated cavity is of the order of 10^{12} (Turner, [12]). Hence, the parameters chosen in the present case study corresponds to a flow in the laminar regime. We have deliberately chosen a laminar regime for our case study so as to understand some of the basic issues regarding stratification in a side-heated cavity.

Flow visualisation is performed by injecting fluorescein sodium and rhodamine dyes along the heated side-walls. The dyes are injected typically after 4.5 to 5 hours of heating, after ensuring that the flow in the cavity becomes fairly developed. The development of the flow is monitored by recording the temperatures measured by the thermocouples situated on the heated walls and along the cavity centreline. Once the temperature curves flatten after some duration of heating (although the system may not have reached a steady state), it can be assumed that the dye mixing pattern will qualitatively depict the nature of flow.

The corresponding numerical simulation of transient natural convection inside the tank is performed by numerically solving the governing differential equations along with the boundary and initial conditions. The cooling effect due to evaporation is calculated using a method described in [13]. The numerical simulation is carried out using a pressure based finite volume method according to the SIMPLER algorithm [14]. In this numerical study we make an attempt to simulate the case corresponding to the case study in the experiments. A 40x40 non-uniform grid system is used for discretisation of the domain.

Results and Discussion

Experiments are performed with an initial water temperature of about 23 $^{\circ}$C (room temperature). The temperature variations recorded by the thermocouples situated on the cavity centreline and on the heated walls are shown in figs 3 and 4, respectively. The corresponding results from the numerical simulation shown in the same figures are in excellent agreement with those from the experiments. As observed in fig. 3, the temperature curves flatten to a reasonable extent after about five hours of heating, although they do not reach steady state by then. It is assumed that if the dye is injected at this stage, the flow pattern will not change significantly during the dye mixing period (typically about 30 minutes), and the dye mixing pattern will qualitatively depict the nature of flow in the cavity at this stage of the heating process.

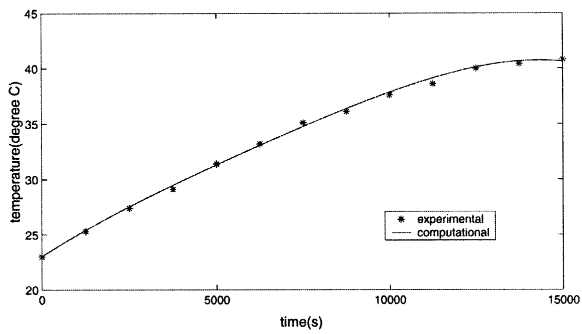

FIG. 3
Variation of centreline temperature with time (at a height y=0.11 m from the bottom)

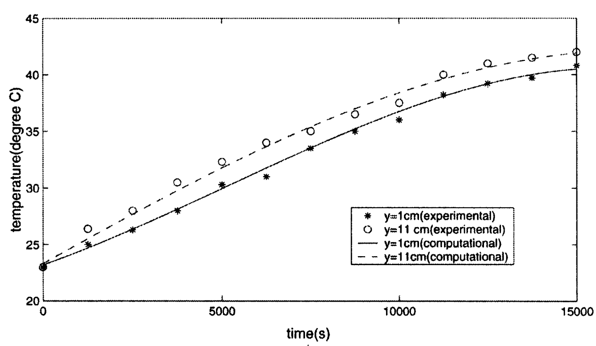

FIG. 4
Variation of wall temperature with time

Fig. 5 shows the experimental dye mixing pattern (on the left half) and the corresponding numerically generated velocity vectors (on the right half). Although the velocity distribution shown is an instantaneous one, there is a good qualitative agreement with the corresponding dye mixing pattern since the dye is injected only after the flow becomes developed, and it is expected that the flow pattern will not change significantly during the dye mixing period. The dye mixing pattern is also predicted numerically by using a streakline as shown in figure 6. As observed in fig. 5, the fluid near the heated wall moves up due to buoyancy. After reaching the top surface it turns towards the centre. Due to symmetry at the centre, it then turns towards the bottom. But as the fluid descends, it starts losing momentum due to viscous and buoyancy effects (acting against the direction of fluid motion), and then turns again towards the heated wall. As the fluid approaches the heated wall, it again bends downward before changing its direction back to the centre of the cavity. This downward movement near the wall suggests that the circulating fluid temperature is higher than that of the layer adjacent to the boundary layer along the wall. This is possible because in the uniform wall heat flux case, the wall temperature increases with its height [3], and hence the circulating fluid is hottest when it reaches the top of the cavity. Although the fluid will lose some heat while executing its first loop, as it approaches the wall again its temperature may still be higher than that of the boundary layer adjacent to the wall.

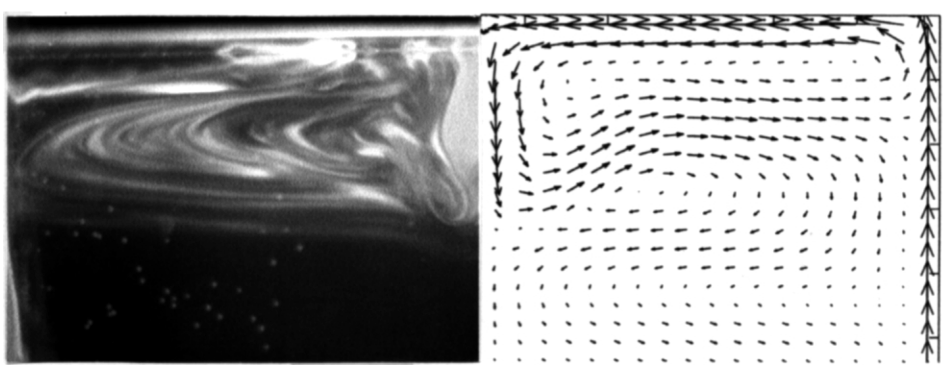

FIG. 5
Composite figure showing experimental flow visualisation (shown on left) and corresponding velocity vectors (shown on right). Only a portion of the cavity is shown.

Fig. 7 shows the temperature contours in the cavity at the end of the heating process. It is clear from the fig. 7 that there is a well-mixed layer at the top, below which the fluid is stratified. In the velocity vector plot in fig. 5, we notice a bifurcation point at the location where the first loop ends and the fluid approaches the heated wall. From the temperature contours of fig. 7, it becomes clear that above this point, the approaching fluid is colder than that of the vertical boundary layer adjacent to the wall, and therefore it gets heated and moves up. Below the bifurcation point, the fluid approaching the vertical boundary layer is cooled and hence it descends to form the secondary loop.

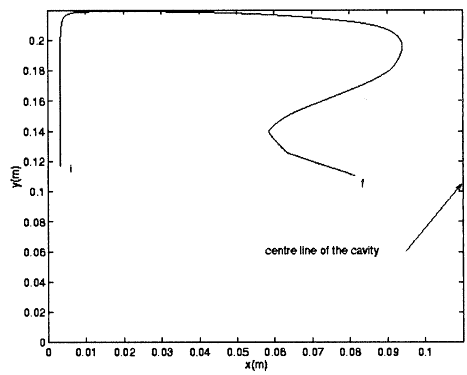

FIG. 6
Plot of trajectory of a fluid particle using the transient velocity field, integrating from a position ‘i’ at time=0s to a position ‘f’ at time=100s.

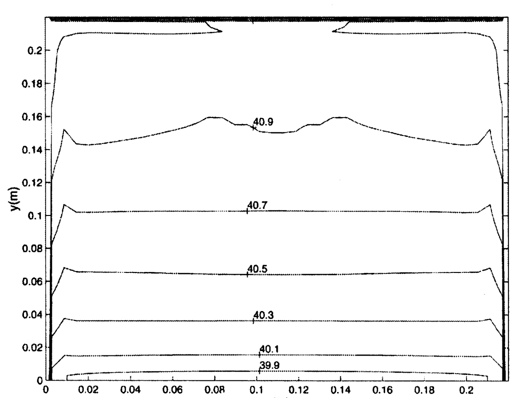

FIG. 7
Isotherms at time=15000s.

Conclusions

An experimental and numerical study is performed for transient natural convection in a two-dimensional rectangular enclosure heated symmetrically from both sides. For the set of flow parameters we have used, the case study leads to a laminar flow. It is observed that the fluid undergoes multiple loops near the top of the cavity, leading to a well-mixed zone near the top of the cavity, below which a thermally stratified zone is observed. Numerical simulation is performed for this case, and the agreement with the flow visualisation and temperature measurement is found to be good.

This study clearly shows that the flow pattern is significantly different form those with differentially heated cavities. The present study is a preliminary investigation of this unique flow phenomenon. Future studies with similar boundary conditions can be performed for a detailed characterisation of the flow physics.

References

[1] Ostrach, S. 1988. *Journal of Heat Transfer*, **105**, 1175-1190

[2] Yang, K. T. 1987. Chapter 13 in Kakak, S., Shah, R. K., and Aung, W. *Handbook of Single Phase Convective Heat Transfer*, Wiley, New York.

[3] Bejan, A. 1995. *Convection Heat Transfer*, Wiley, New York, 219

[4] Pellew, A. and Southwell, R. V. 1940. *Proceedings of the Royal Society*, **A176**, 312-343

[5] Simpkins, P. G., and Dudderar, T. D. 1981, *Journal of Fluid Mechanics*, **110**, 433-456.

[6] Fusegi, T., Hyun, J. M., Kuwahara, K., and Farouk, B. 1991, *International Journal of Heat and Mass Transfer*, **34**, 1543-1557

[7] Yewell, R., Poulikakos, D., and Bejan, A. 1982, *Journal of Heat Transfer*, **104**, 533-538

[8] Schladow, S. G., Patterson, J. C., and Street, R. L. 1989, *Journal of Fluid Mechanics*, **200**, 1121-1148

[9] Schladow, S. G. 1990, *Journal of Fluid Mechanics*, **213**, 589-610

[10] Lin, C. S., and Hassan, M. M. 1990, *AIAA* paper No. 90-2375, 1990

[11] Tanyun, Z., Zhongpin, H., and Li, S. 1996. *Advances in Cryogenic Engineering*, **41**, 155-161

[12] Turner, J. S. 1973. *Buoyancy Effects in Fluids*. Cambridge University Press, 368

[13] Incropera, F. P., and Dewitt, D.P. 1996. *Fundamentals of Heat and Mass Transfer.* John Wiley & Sons, New York.

[14] Patankar, S. V. 1980. *Numerical Heat Transfer and Fluid Flow*. McGraw-Hill.

Received November 1, 2001