International Journal of Heat and Mass Transfer 81 (2015) 829–845

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Numerical analysis of mixed convection in partially open cavities heated from below

Éliton Fontana [a,b,*], Claudia A. Capeletto [a], Adriano da Silva [c], Viviana C. Mariani [d,e]

[a] Universidade Federal de Santa Catarina, Departamento de Engenharia Química e de Alimentos, 88040-970 Florianópolis, SC, Brazil
[b] Universidade Federal de Santa Catarina, Campus Blumenau, 89065-300 Blumenau, SC, Brazil
[c] Universidade Federal do Rio Grande, Escola de Química e Alimentos, 95500-000 Santo Antônio da Patrulha, RS, Brazil
[d] Pontifícia Universidade Católica do Paraná, Programa de Pós-Graduação em Engenharia Mecânica, 80215-901 Curitiba, PR, Brazil
[e] Universidade Federal do Paraná, Departamento de Engenharia Elétrica, 80060-000 Curitiba, PR, Brazil

ARTICLE INFO

Article history:
Received 22 July 2014
Received in revised form 1 November 2014
Accepted 3 November 2014
Available online 20 November 2014

Keywords:
Mixed convection
Heat transfer
CFD analysis
Periodic flow

ABSTRACT

Mixed and natural convection in cavities are extensively investigated subjects in the field of heat transfer, due to the wide range of situations where this configuration appears in technological applications and natural phenomena. Although the flow behavior is rather complex, most of the numerical studies found in the literature apply simplified two-dimensional stationary models to investigate the influence of the governing parameters. In the study reported herein, the main characteristics of mixed convection in partially open cavities with internal heat sources are investigated using a transient three-dimensional model. An inlet condition occupying half of the entire right wall and an opposite opening in the left wall were included to allow the fluid inlet and outlet. The model also comprises an external region adjacent to the opening, since recirculations can appear at the opening and generate backflow. The heat sources are placed at the bottom wall of the cavity, which is also maintained at a high temperature, while the upper part of the cavity is cooled as a result of the presence of cold walls. These conditions are similar to those found in common Rayleigh–Bénard systems, therefore, the emergence of thermal convective cells occurs when a sufficiently high temperature difference is reached. Different values for the Rayleigh numbers associated with the internal heat source intensity and with the temperature difference between the hot and cold walls were evaluated, allowing the determination of the modifications in the flow field as the buoyancy forces become dominant. The results show that the buoyancy forces are dominant for low Re and high R or Ra_e values, and in this case the flow field correspond to two recirculations rotating in opposite directions. As the relative intensity of the shear flow increases, the recirculations become distorted and the size of the recirculation close to the inlet increases. For higher Re values, when the inertial forces induced by the inlet velocity are stronger than the buoyancy forces, recirculation zones in the upper part of the cavity can also appear due to the influence of the horizontal flow. For intermediate values, the competition between the buoyancy and inertial forces generates a complex dynamic behavior, with periodic and quasiperiodic regimes found for some set of the governing parameters. Furthermore, the importance of a fully three-dimensional model is demonstrated by the analysis of the flow field and energy distribution along the three directions.

1. Introduction

In the last decades, mixed convection heat transfer has been a great interest for many engineering and science researchers in the electronic industry, due to the great requirement of effective cooling strategies for electronic elements. The main objective of their studies has been to understand the fundamentals of various cooling strategies and to achieve high performance cooling methods which meets the heat removal needs of electronic devices with certain geometries [1]. In the present scenario, electronic gadgets and modules are extensively used in strategically important areas like aerospace, defense and biomedical engineering, where malfunction cannot be afforded at any cost [2]. The common practice for cooling of heat generating elements in situations where large heat fluxes need to be dissipated, is to allow a low forced convective cooling and simultaneously utilizing buoyancy effects. The

* Corresponding author at: Universidade Federal de Santa Catarina, Campus Blumenau, 89065-300 Blumenau, SC, Brazil. Tel./fax: +55 48 3721 3375.
E-mail address: eliton.fontana@ufsc.br (É. Fontana).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.11.011
0017-9310/

subject of electronic cooling has therefore generated increased interest in the analysis of fluid flow and heat transfer in discrete heating situations. Besides the cooling of electronic devices, mixed convection in cavities can be observed in several other technological applications, as for example in the ventilation of aircraft cabins [3], heat exchangers [4] and indoor heating with radiators [5].

Several analytical, numerical and experimental studies has been carried out to investigate the heat transfer and fluid flow in mixed convection systems [1,2,6–16]. Most of these studies focus on the heat transfer in vertical channels. For example, natural, forced and mixed convection heat transfer from protruding and flush-mounted discrete heat sources inside vertical channels have been investigated as cited in Dogan et al. [6]. A numerical study on mixed convection in an open cavity with a heated wall bounded by horizontally insulated plates was presented by Manca et al. [8]. They considered three basic heating modes for the open cavity and compared their thermal performances. Brown and Lai [9] studied numerically a horizontal channel with an open cavity and obtained correlations for combined heat and mass transfer which covered the entire convection regime from natural, mixed, and forced convection. Leong et al. [10] investigated the mixed convection for the same geometry used by Brown and Lai [9], showing that the Reynolds and Grashof numbers control the flow pattern and the transition to the mixed convection regime. Hacohen et al. [11] presented an experimental and theoretical study for forced and free convection with flush-mounted and protruding heat sources. Effects of channel geometry, component array height, airflow rate and heat flux were studied. Basak et al. [12] evaluated mixed convection flows within a square cavity with linearly heated side walls and observed that the average Nusselt number on the bottom wall increases by increasing the Prandtl and Grashof numbers. Mixed convection flow in a rectangular ventilated cavity with a heat conducting solid circular cylinder was studied by Rahman et al. [13]. The authors observed that the average Nusselt number at the heated surface was higher for the lowest values of the aspect ratios.

It is well known that problems involving natural and mixed convection in cavities have a rich dynamical behavior, showing in several situations, for example, multiplicity of stationary states [17–19], periodic [20–27], quasiperiodic [27–31] or even chaotic [30–36] regimes. The onset of unsteady periodic regimes in most cases occurs through a Hopf bifurcation [20,25,34,36], giving rise to stable limit cycles, where self-sustained oscillations over the time are observed for the field variables. For the case of natural convection in fluids with low Prandtl number evaluated by Mercader et al. [34], the further increase of the Rayleigh number above the point where the Hopf bifurcation appears originates a complex scenario of bifurcations and eventually chaotic states emerge. Piazza and Ciofalo [31] showed that for the natural convection in a rectangular cavity, during the transition from periodic to chaotic regimes, quasiperiodic states can appear. In this case, the system shows an irregular behavior, regulated by more than one incommensurate frequencies characterized by an open orbit in a torus. The transition to chaos through torus breakdown in natural and mixed convection systems was also observed by several other authors [32,34].

From the above review of the literature it is apparent that the analysis of fluid flow and heat transfer in natural and mixed convection problems is complex, particularly due to the large number of governing parameter and the non-trivial dynamic behavior. Furthermore, when open systems are analyzed, the hypothesis of constant mass and energy inside the cavity cannot be considered and the influence of the mass and energy flow through the open boundaries needs to be taken in account. The problem of natural convection in partially open cavities was previous investigated using two-dimensional models [37–39] and later expanded to three-dimensional models [40]. These studies evaluated the natural convection induced by a temperature difference between vertical walls and the influence of one or more internal heat sources on the flow field and energy distribution inside the cavity and on the mass flow through the opening. In Fontana et al. [40] it was showed that for a cubic cavity the field variables can vary significantly along the three dimensions, particularly for high Rayleigh numbers.

The aim of this paper is to investigate mixed convection in a partially open cavity with internal heat sources, using a three-dimensional numerical mesh comprising a region external to the opening. Although computationally expensive, this model allows a detailed analysis of the problem. The boundary conditions are set so that the natural convection can be induced by the internal heat sources, positioned at the bottom wall, or by the temperature difference between the bottom wall (hot) and the upper and vertical walls (cold). This configuration resembles the Rayleigh–Bénard system, where the buoyancy forces can induce the formation of convective cells. Furthermore, the flow induced by the inlet condition can leads to the formation of eddies inside the cavity, creating a complex system where different forces govern the flow pattern.

The paper is structured as follows: in the next section, a detailed description of the problem is presented, including the physical domain, governing equations and boundary conditions used. In Section 3 the main results are discussed, in particular the different processes of formation of recirculations and the competition between them. In Section 4, concluding remarks end the paper.

2. Governing equations and numerical method

To analyze the influence of the external flow and the intensity of heat dissipation on the flow field inside the cavity, three-dimensional numerical models comprising different numbers of internal heat sources are used. A scheme of the domain representing Case 1 (one source) is shown in Fig. 1. In this figure, W represents the cavity length in the streamwise direction and H the total height of the cavity. An inlet occupying half of the total height is placed in the lower part of the right vertical wall, while a corresponding opening condition is placed on the left vertical wall. An outside region is also considered in the model, since recirculations developed at the opening can cause backflow from the outside to the inside region.

The temperature of the upper and the vertical walls is set to a constant value T_C, while the bottom wall, including the region in direct contact with the internal heat sources, is assumed to be at

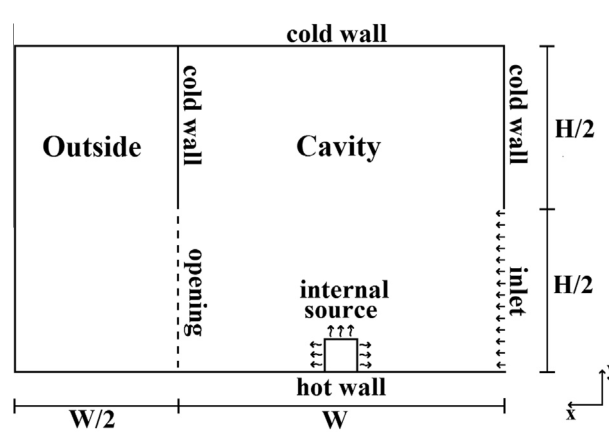

Fig. 1. Frontal view of the three-dimensional domain for Case 1.