International Communications in Heat and Mass Transfer 95 (2018) 9–18

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Oscillating convective airflow in a vented cavity with a heated immersed body. Influence of the heating intensity

B. Zamora

Dpto. Ingeniería Térmica y de Fluidos, Universidad Politécnica de Cartagena, Doctor Fleming s/n, Cartagena 30202, Spain

ARTICLE INFO

Keywords:
Natural Convection
Numerical Simulation
Oscillatory flows
Heating intensity
Open cavities

ABSTRACT

The influence of the air variable properties on the buoyancy-driven airflows established in vented square cavities with an inner heated body is numerically investigated. Two-dimensional, unsteady, and turbulent simulations are obtained, considering uniform wall temperature heating conditions. The low-Reynolds $k-\omega$ turbulence model is employed. The average Nusselt number and the dimensionless mass-flow rate are obtained for a range of the Rayleigh number varying from 10^4 to 10^{12}. The results obtained for different heating intensities are analyzed and compared. The conditions under which the flow becomes clearly transient, giving rise to an oscillatory solution, are determined. The dimensionless oscillating period of the transient Nusselt number exhibits a logarithmic decay as a function of the Rayleigh number. The structure of the flow into the cavity as a function of time, are shown.

1. Introduction

1.1. Background

The natural convection airflows in cavities and enclosures has received a considerable attention from researchers (Ostrach [1], Bejan [2], Henkes and Hoogendorn [3], Turan et al. [4], among others). Several geometries have been studied, including different heating conditions. Examples of numerical studies focused on square cavities with different morphologies, are the works conducted by Bilgen and Balkaya [5], and Muftuoglu and Bilgen [6], for instance.

The case commonly known as *cavity heated from the side* consists of a cavity (rectangular or square in most of cases) in which the horizontal walls are insulated, whereas the vertical walls are at hot (T_h) and cold (T_c) temperatures, respectively; the numerical benchmark solution of De Vahl Davies [7] has constituted a reference work for comparison and validation purposes (Markatos and Pericleous [8], Ampofo and Karayiannis [9], Ridouane et al. [10], for instance). Another sample configuration is that known as *cavity heated from bellow*. The fundamental difference between the enclosures or cavities heated from the side and those heated from below has clearly exposed by Bejan [2]. In the first configuration, the convective flow is present for very small temperature difference ($T_h - T_c$) between the two opposite side walls. On the contrary, in the second configuration, the temperature difference must exceed a given *critical* value, above which the flow induced by buoyancy forces is detected.

Let us consider two parallel plates in the horizontal direction, being heated the lower one and cooled the upper one. When the plates are long enough, the convective flow appears above a *critical* value of the Rayleigh number (based on the inter-plate spacing H) equal to 1708. The flow pattern is commonly named as *Rayleigh-Bénard convection*, and it can be identified by counterrotating two-dimensional rolls, being the cross section almost square. For larger values of Ra_H, the cells break down and the motion is turbulent (mainly for $\mathrm{Ra}_H > 10^5$). Similar structures to the *Bénard cells* can appear under given circumstances in other configurations, such as cavities or enclosures, in which the bottom plate can be fully or partially heated. As expected, several works can be found in this matter, as experimental (Corvaro and Paroncini [11], for instance), as numerical (Sourtiji et al. [12], among others). The last authors validated their numerical results with those obtained by Khanafer et al. [13] and Markatos and Pericleus [8], for enclosures heated from the side. Basically the same configuration was considered by Calcagni et al. [14] in their numerical and experimental study. Sharma et al. [15] validated their turbulent numerical results with those obtained for Calcagni et al. [14] and Aydin and Yang [16]. The last works will be considered in this work for validation purposes (configuration of Fig. 1a).

1.2. Cavities including an immersed body

Because of the considered bodies or obstructions within the cavity may increase (or decrease) the Nusselt numbers at walls, a considerable research on this matter can be found in the literature. In addition, for the fully understanding of the flow characteristics, it is necessary the

E-mail address: blas.zamora@upct.es.

https://doi.org/10.1016/j.icheatmasstransfer.2018.03.026

Nomenclature

b	width of the vents, m (Fig. 1c)
c_p	specific heat at constant pressure, J kg^{-1} K^{-1}
Fo	Fourier number, Fo = $\alpha_\infty t_0/l^2$
g	gravitational acceleration, m s^{-2}
Gr_l	Grashof number $g\beta(T_w - T_\infty)l^3/\nu_\infty^2$
H	height of the cavity (Fig. 1), m
H_c	height (and length) of the heated inner body (Fig. 1b and c), m
h_x	local heat transfer coefficient, $-\kappa(\partial T/\partial n)_w/(T_w - T_\infty)$, W m^{-2} K^{-1}
L	length of the cavity (Fig. 1), m
L_h	length of the heated wall (Fig. 1a), m
l	typical length, m
M	dimensionless mass-flow rate, $m/\rho_\infty\alpha_\infty$
m	mass-flow rate, kg s^{-1} m^{-1} (two-dimensional)
Nu_l	average Nusselt number based on l, isothermal cases, Eq. (3)
Nu_x	local Nusselt number, $h_x l/\kappa$
P	average reduced pressure, N m^{-2}
p	pressure, N m^{-2}
Pr	Prandtl number, $\mu c_p/\kappa$
R	constant of air, R = 287 J kg^{-1} K^{-1}
Ra_l	Rayleigh number based on l, $(\mathrm{Gr}_H)(\mathrm{Pr})$
$\widetilde{\mathrm{Ra}}_l$	Rayleigh number from which the flow becomes oscillating
T, T'	average and turbulent temperatures, respectively, K
$\widetilde{T}$	dimensionless oscillating period
t, t_0	time, typical time, s
$-\overline{T' u_j}$	average turbulent heat flux, K m s^{-1}
U_j, u_j	average and turbulent components of velocity, respectively, m s^{-1}
$-\overline{u_i u_j}$	turbulent stress, m^2 s^{-2}
u_τ	friction velocity, $u_\tau = (\tau_w/\rho)^{1/2}$, m s^{-1}
V	absolute value of velocity, m s^{-1}
x, y	cartesian coordinates (Fig. 1), m
y_1	distance between the wall and the first grid point, m
y^+	$\rho y_1 u_\tau/\mu$

Greek symbols

α	thermal diffusivity, $\kappa/\rho c_p$, m^2 s^{-1}
β	coefficient of thermal expansion, $1/T_\infty$, K^{-1}
κ	thermal conductivity, W m^{-1} K^{-1}
Λ	heating parameter, Eq. (2)
μ	viscosity, kg m^{-1} s^{-1}
ν	kinematic viscosity, μ/ρ, m^2 s^{-1}
θ	dimensionless temperature difference, $\theta = (T - T_\infty)/(\Lambda T_\infty)$
ρ	density, kg m^{-3}
τ	dimensionless time, $\tau = \alpha_\infty t/l^2$
τ_w	wall shear stress, N m^{-2}
ω	specific dissipation rate of k, s^{-1}

Subscripts

c	cooled
h	heated
w	wall
∞	ambient or reference conditions

Superscripts

$-$	averaged value

study of the structures of resulting buoyant plumes, as well as their stability or the presence of *bifurcation points* as the Rayleigh number increases, for different geometric configurations (Gebhart et al. [17], Desrayaund and Lauriat [18], Bouafia and Daube [19], among others).

Sun et al. [20] conducted a numerical study of combined natural convection and surface radiation in a square cavity with a centered (heated) immersed body, on the basis of their previous experimental experiences. The considered configuration is outlined in Fig. 1b. The flow motion is stable up to a $\mathrm{Ra}_H \approx 2 \times 10^5$ for pure convection. Below this value, the flow is steady and symmetric, but for larger values of Rayleigh, the flow becomes unsteady and asymmetric. For stable scenarios, they detected two symmetric, counter-rotating, Rayleigh-Bénard type cells at the upper part of the cavity, in turn delimited by two large cells rotating in opposite directions along the vertical walls. Given the similarities, note that this cited work can be considered as a reference for our computations.

Studies focusing on other aspects of the problem, such as the effects of the variable thermophysical properties of air are more scarce, although it can be found some relevant works, such as those of Bouafia and Daube [19], using the low Mach number approximation, as well as Sourtiji et al. [12], in this case for a square enclosure heated from below, without any baffle in the interior.

1.3. Variable thermophysical properties

Numerically, the force driving the flow can be simulated by means of the Boussinesq approach, which only retains the density variations due to thermal gradients in the buoyancy term of the momentum equation (the rest of the properties are considered constant). However, intense heating conditions can change drastically the properties of the flow, and therefore the predictions of the heat transfer coefficients at walls (Emery and Lee [21], Guo and Wu [22], Hernández and Zamora [23]). In the field of interest, the influence of the variable properties of fluid should not be neglected in some cases; in fact, this work analyses the behavior of the flow under intense heating conditions. In the concerned literature, under given circumstances a clear decrease of the heat transfer coefficients are detected for intense heating conditions, due to the *thermal drag* (related to the density decreasing) and the *viscous drag* (related to the viscosity increasing) phenomena, described for instance by Guo and Wu [22] and Hernández and Zamora [23].

1.4. Objectives

A limited attention has deserved the explained convective airflow when the cavity is considered as partially open, i.e., with some vents through which the fluid can entry or exit, besides a centered inner body. The regarded configuration is given by Fig. 1c. Here, the flow mainly enters through the lower vents of width b, and goes up mainly through the upper vents. The immersed body is heated at uniform temperature T_h, whereas the rest of the walls remain adiabatic. The structure of the flow motion will be studied, following the ideas exposed in the literature previously cited. It can be expected that the flow was turbulent and unstable or oscillating for high values of the Rayleigh number. Hence, the numerical simulation is carried out as transient. The Rayleigh number from which the flow becomes oscillating (and therefore unsteady), will be determined. In addition, the transience of the flow will be explained, highlighting the oscillating flow patterns encountered for given ranges of parameters.