International Journal of Heat and Mass Transfer 101 (2016) 682–691

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Steady and unsteady mixed convection flow in a cubical open cavity with the bottom wall heated

Gorg Abdelmassih, Anton Vernet, Jordi Pallares *

Department d'Enginyeria Mecànica, Universitat Rovira i Virgili, Av. Països Catalans 26, 43007 Tarragona, Spain

ARTICLE INFO

Article history:
Received 28 January 2016
Received in revised form 4 May 2016
Accepted 17 May 2016

Keywords:
Mixed convection
Open cavity
PIV
DNS
Turbulent

ABSTRACT

In this study we analyze experiments and numerical simulations of steady and unsteady mixed convection flow in a cubical cavity located at the bottom of a square channel. The Reynolds numbers based on the mean flow velocity and the channel width are in the range $100 \leqslant Re \leqslant 1500$ and the Richardson numbers vary within $0.1 \leqslant Ri \leqslant 10$. Particle Image Velocimetry has been used for the measurements in a water channel. Three-dimensional direct numerical simulations have been carried out with a second order finite volume code considering the Boussinesq approximation since, for the experimental conditions considered, the variation of the physical properties with temperature has no significant influence on the overall flow topology. For $100 \leqslant Re \leqslant 1500$ and $Ri \leqslant 0.1$ the flow is steady and it consists in a single roll that exhibits larger velocities as the Richardson number is increased. An unsteady periodic flow is found at $Re = 100$ and $Ri = 10$. Alternate flow ejections from the cavity to the channel occur near the lateral walls while the flow enters the cavity from the channel through the central part of the cavity. A conditional sampling technique has been used to elucidate the evolution of the mean unsteady turbulent flow at $Ri = 10$. It has been found that the alternate flow ejections persist for all the Reynolds analyzed. The computed Nusselt numbers are in general agreement with a previously reported correlation, valid for two dimensional cavities of different aspects ratios.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

During several decades efforts have been directed to analyze fluid flow and/or heat transfer processes in flows over heated open cavities due to its importance in several engineering applications. This type of geometry can be found, for example, in the landing gear wells [1–6]. Cooling of electronic components [5–7] is one of the most used applications at small scales for the flow over an open cavity. It is simple in design and has a cheap maintenance cost. The electronic component considered as the source of heat and the natural or mixed convection effecting on the flow structure is the crucial point. It can be classified under three main categories regarding to the location of the heated wall, e.g. the electronic board, which can be located on the leading vertical wall (natural convection assisting flow), on the trailing vertical wall (natural convection opposing flow) or on floor of the cavity (heated from below), see [7–11].

In renewable energy research it has been reported that the installation of wind barriers along the perimeter of solar cells improves their absorption efficiency. The geometry of the flat solar collectors with the wind barriers has the same geometry as an open cavity at medium scales.

Early studies of flow past cavities were achieved in the 1960s [1–3]. More studies followed to reveal the flow structure and the heat transfer process occurring for both natural and mixed convection for geometries with different aspect ratios [4–6,9]. Manca et al. [7], investigated experimentally the opposing mixed convection flow in an open cavity with a heated wall bounded by a horizontal unheated plate for ($100 \leqslant Re \leqslant 2000$) and ($4.3 \leqslant Ri \leqslant 6400$). The cavity aspect ratio was ($0.5 \leqslant AR \leqslant 2.0$). They reported that for low Reynolds numbers, the forced motion penetrates inside the cavity, and the vortex structure is adjacent to the unheated vertical plate. At higher Reynolds numbers, the vortex structure has a larger extension while AR is held constant. The effects of the position of a heated wall on mixed convection in a channel with an open cavity have been studied numerically in [8]. A two-dimensional numerical approach was considered with different aspect ratios. The authors found that the maximum temperature values decrease as the Reynolds number and Richardson number increase for all the studied configurations.

Three dimensional numerical studies of the flow and heat transfer characteristics for assisting and opposing incompressible laminar flow past an open cavity can be found in Stiriba [10] and Stiriba

* Corresponding author. Tel.: +34 977 559 682; fax: +34 977 559 691.
E-mail address: jordi.pallares@urv.cat (J. Pallares).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.05.074
0017-9310/© 2016 Elsevier Ltd. All rights reserved.

Nomenclature

AR	aspect ratio
f	frequency
g	gravity acceleration
Gr	Grashof number
L	cavity dimensions
Nu	Nusselt number
p	pressure
Pr	Prandtl number
Re	Reynolds number
Ri	Richardson number
t	time
T	temperature
u, v, w	velocity components
U_0	inflow velocity
$\vec{V}$	velocity vector
x, y, z	cartesian coordinates

Greek letters

Δ	increment
α	thermal diffusivity
β	thermal expansion coefficient
λ_2	second largest eigenvalue of the velocity gradient tensor
ν	kinematic viscosity

Superscripts and subscripts

$*$	non-dimensional quantity
∞	reference value
H	hot
l	local
S	surface averaged
w	wall

et al. [11]. They reported that the flow exhibits a three-dimensional structure and it is steady for $Re = 100$ with $(0.001 \leqslant Ri \leqslant 10)$ and $Re = 1000$ with $(0.001 \leqslant Ri \leqslant 1)$. The forced flow dominates the transport mechanism and a large recirculating zone occurs inside the cavity which results in heat transfer mainly by conduction. Abdelmassih et al. [12], performed numerical simulation of incompressible laminar flow in a three-dimensional channel with a cubical open cavity with a bottom wall heated. Air-flow has been used in the inflow. They noted that the buoyancy is weak and not affect the steadiness of the flow for small Richardson numbers $Ri \leqslant 0.1$ on a range of $Re \leqslant 1000$.

O'Hern et al. [13], carried out an experimental study using Particle Image Velocimetry (PIV) techniques for an isothermal configuration of an open cavity. Water inflow in isothermal configuration was used with steady state in the range of Reynolds numbers from 100 to 900. These authors reported the difficulties found using PIV because the maximum velocity inside the cavity is many times smaller than the flow velocity over the cavity. Stiriba et al. [14], compared the velocities of the air-flow structure within a rectangular cavity heated from below. They reported that increasing the Richardson number generates remarkable velocities differences while the inflow Reynolds is held constant. Zamzari et al. [15], performed a two dimensional study of the entropy generation and mixed convection in a horizontal channel with an open cavity. Laminar air-flow in the ranges $200 \leqslant Re \leqslant 500$ and $0.25 \leqslant Ri \leqslant 1$ is considered. Their results show that the cavity flow, heat transfer rates and entropy generation are strongly affected by variations of the Reynolds number, Richardson number and the aspect ratio.

Most of the works found in the literature are numerical studies for both two and three dimensional configurations with a varying Re, Ri and AR. However a few experimental studies have been reported specially using PIV techniques because its measurement challenges [13]. In addition, the flow structure and its time evolution when the flow becomes unsteady have not been analyzed in detail [12][14][16]. The main objective of this work is to study the mixed convective flow and three dimensional instabilities, focusing on the effect of the buoyancy forces on the flow topology for an incompressible laminar water inflow in an open cavity with a bottom wall heated. The ranges of Reynolds and Richardson numbers considered $(100 \leqslant Re \leqslant 1500$ and $0.1 \leqslant Ri \leqslant 10)$ correspond to the usual operating conditions of some processes associated with the cooling of electronic components. The effect of Ri on the flow stability and the analysis of the flow topology and its time evolution when the flow is unsteady are the main parts of this work. The experimental measurements allowed the confirmation of the numerical results.

2. Numerical approach

2.1. Physical problem and computational domain

The geometry of the channel including the cubical open cavity of size L and the computational domain used in this study are shown in Fig. 1.

The inflow located at $x = 0$ has uniform velocity U_0 and temperature T_∞. Convective Euler boundary conditions are imposed at the outflow located at $x = 4L$ and the non-slip boundary condition is applied for the rest of the boundaries. The cavity is heated from below at a constant temperature T_H, and the remaining walls of the cavity and of the channel are adiabatic. A cooling incompressible water-flow has been used. The Prandtl number of water is $Pr = 7$.

2.2. Governing equations and numerical method

The 3DINAMICS finite volume parallel code has been used in this work [10–12,14].The code solves numerically the three-dimensional incompressible Navier–Stokes Eqs. (1–3) for mixed convection [17], on non-uniform staggered Cartesian meshes. The SMAC-method is used to join continuity and momentum equation, in which, the Poisson equation for the pressure is computed with the biconjugate gradient method (BiCGtab). The convective and

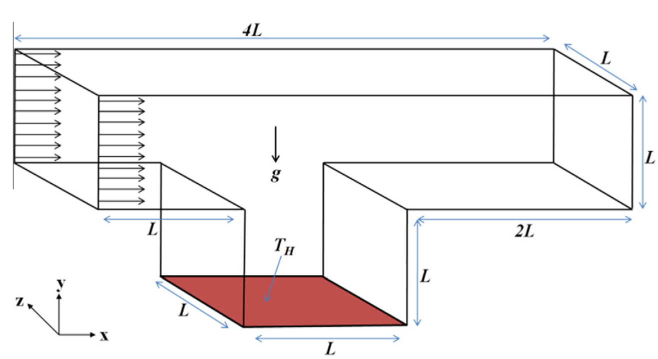

Fig. 1. Sketch of the computational domain.

Download English Version:

https://daneshyari.com/en/article/7055360

Download Persian Version:

https://daneshyari.com/article/7055360

Daneshyari.com