International Journal of Thermal Sciences 107 (2016) 220–229

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Heat and fluid flow in an uneven heated chimney

C. Hemmer [a, b], C.V. Popa [a, *], A. Sergent [c, d], G. Polidori [a]

[a] GRESPI/Thermomécanique, Université de Reims Champagne-Ardenne, Moulin de la Housse, BP 1039, 51687 Reims Cedex 2, France
[b] CAMPA, route de Soissons, 51170 Fismes, France
[c] LIMSI, CNRS, Université Paris-Saclay, Rue John Von Neumann, 91400 Orsay, France
[d] Sorbonne Universités, UPMC Univ Paris 06, UFR d'Ingénièrie, 4 place Jussieu, 75005 Paris, France

ARTICLE INFO

Article history:
Received 9 April 2015
Received in revised form
13 April 2016
Accepted 14 April 2016
Available online 22 April 2016

Keywords:
Natural convection
Numerical simulation
Vertical channel
Heat source location
Asymmetric heating
Chimney

ABSTRACT

A 2D numerical study of the natural convection flow in an asymmetrically heated vertical plane channel is carried out in the boundary layer regime for three modified Rayleigh numbers (Ra^*) between $2.25 \cdot 10^6$ and $9 \cdot 10^6$. This configuration is seen to be a simplified model of an electrical heater. This study focuses on the influence of different spatial arrangements of the heat sources on both heat transfer and flow dynamics, and especially on the back flow. In order to simplify the analysis, the working fluid is water, making thermal radiation negligible. It is shown that, at constant heat power and exchange surface, two distant heat sources are more efficient to put fluid into motion and to transfer heat.

1. Introduction

The design of new electric heaters is subordinated to the respect for European standards which the designers have to take into account [1]. One of the main constraints, which determines the internal choice and arrangement of the various components of radiators, is the control of the temperature on accessible surfaces of the heater for obvious safety reasons [2]. One of the lines of thought envisaged by the manufacturers is to consider various heat sources positioned vertically inside the box of the heater, under the same incident electric power [3]. It is in this context that we should consider the present study, where the radiator box is then likened to a vertical open-ended channel within whom the knowledge of the thermo-convective mechanisms becoming established is necessary.

This important step in the design of electric heaters includes several difficulties: the interaction between different modes of heat transfer (conduction, convection and radiation), the shape and the position of the electrical resistances. In order to simplify the design of electric heaters, this paper deals with the numerical investigation of the natural convection flow, neglecting radiation, in a vertical plane channel heated asymmetrically which represents a simplified electric heater.

Beyond the present study initially motivated by an industrial application in the electric heating, the heated vertical channel is also representative of several problems such double-skin facade or Trombe wall, the chimney effects and solar panel to name a few. For example, several studies [4] [5], have shown that the implementation of the concept of double-skin facade in a building with low inertia seems essential to improve the thermal comfort. Despite a great number of experimental and numerical studies concerning convective heat transfer in vertical channel heated or cooled, there is still a lack of fundamental knowledge in the main dynamic and thermal behaviors of such double skin facades in order to maximize their efficiency and minimize resulting energy losses. Numerical simulations of natural convection in agreement with experimental results can allow understanding complex phenomena involved in these cases. Although many experimental studies on the vertical channel have been conducted [6–8], the majority of studies have been limited to thermal measurements and studies concerning the flow dynamics are not so numerous in the literature [9–11]. Common to all these studies is the observation of an upward flow along the heated wall and a downward one along the opposed adiabatic wall which is Ra^* dependent. For example, Elenbaas [6] has determined, in a vertical channel, several

* Corresponding author.
E-mail address: catalin.popa@univ-reims.fr (C.V. Popa).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.04.015
1290-0729/

Nomenclature

A	maximum length of the heated zone, m
b	channel wall spacing, m
C_p	specific heat capacity, J/kg K
g	acceleration of gravity, m/s^2
H	total length of the heating parts, m
H^+	dimensionless vertical coordinate
k	thermal conductivity, W/m K
L	recirculation length, m
$\overline{Nu}$	averaged Nusselt number over the heated length
P	total dissipated heat power, W
P*	static pressure, Pa
Pr	Prandtl number ($Pr = \nu/\alpha$)
Ra	Rayleigh number $\left(Ra = \frac{g\beta\phi b^4}{k\nu^2}Pr\right)$
Ra*	modified Rayleigh number ($Ra^* = Ra/R_f$)
R_f	aspect ratio of the heated zone ($R_f = A/b$)
T	temperature, K
T^+	dimensionless temperature
ΔT	increasing temperature ($\Delta T = T - T_0$), K
u,v	horizontal and vertical velocity components, m/s
x,y	horizontal and vertical and coordinates, m

Greek symbols

α	thermal diffusivity, m^2/s
β	volume expansion coefficient, 1/K
ϕ	heat flux density, W/m^2
Φ	heat flux, W
ρ	density, kg/m^3
ν	kinematic viscosity, m^2/s
μ	dynamic viscosity, Pa.s

kinds of dynamical flow behaviors depending on the modified Rayleigh number (Ra*), which is based on the channel width (b) and the ratio (A/b) where A is the heated length at wall. These authors highlighted that for a low modified Rayleigh number (Ra*<100), the flow regime is fully developed over the entire width of the channel whereas for a high-modified Rayleigh number (Ra*>1000), the flow regime is of boundary layer-type along the heated wall with the presence of a reverse flow observed near the unheated wall. This flow structure was also observed by Manca et al. [12].

For such free convection coupled problem the knowledge of quantitative dynamic quantities is essential for the implement of numerical simulation codes. It is the reason why, in literature, numerical studies [13–15] have tempted to correlate the geometrical and heating channel conditions with the dynamics and structure of the flow. Christian et al. [16] performed a numerical study of the natural convection flow in a ventilated façade, which was modeled by a vertical channel heated with a constant imposed temperature. The author showed for $Ra^* > 100$ that convection is the principal mode of heat transfer. Corresponding correlations for the average Nusselt number were highlighted. Dehghan et al. [17] have studied natural convection in a vertical slot with two heat source elements. The influence of both the heat generation rate and the separation distance between the sources has been investigated. It has been shown that wall conduction was important, and for substrates with a finite thermal conductivity, it is essential that the conjugate analysis be employed. The authors showed that the thermal interference between the heat sources was reduced increasing Ra*. In this study it was also observed that increasing the spacing between the sources had the effect to increase velocities within the channel. This led to an enhancement in the convective cooling of the upper located electronic component. A higher cold mass flow rate entering the cavity was observed when the spacing between the sources increased, which resulted in an increase in the convection heat transfer from both components. Fossa et al. [18] and Menezo et al. [19] studied the case of a non-uniformly heated channel at large numbers of modified Rayleigh channel. They studied a channel with an alternating periodic heating either on both sides or on a single. They measured the wall temperatures to determine the heat transfer in each zone. The authors underlined a new geometrical parameter in the non-uniform heating case. This geometrical parameter has been defined as the distance between the inlet channel and the beginning of the heating zone. They showed that an alternation between heated and unheated zones could enhance the heat transfer up to 20%, compared to a configuration of uniform heating at one wall. From a numerical point of view, two main approaches can be proposed to solve the problem of natural convection flow in the channel: either a complete simulation of the channel and its external environment, or a truncated simulation considering the single channel limited to its geometric limitations. The second approach is problematic in that boundary conditions at bottom and top channel interfaces are *a priori* unknown since the driving flow is located within the computation domain (Garnier [21]). Desrayaud et al. [20] have carried out a numerical study to investigate the sensitivity and the influence of several boundary conditions on the heat transfer in an asymmetrically heated channel. The authors have shown that when the fluid domain is limited only to the channel geometry, the pressure boundary conditions at the inlet and the outlet of the channel are difficult to model. This non-exhaustive state of the art shows how this problem still questions.

The present work aims at evidencing the influence of the spatial arrangement of several heat sources placed on one of the channel side walls by performing 2D simulations. The influence of three different values of the total heat power ($P/2$;P;$2P$) given to the channel flow and five heat source arrangements on the flow, the wall temperature distribution and the heat transfer inside a vertical channel are examined. To compare the influence of the geometrical configurations on the flow and temperature fields, the results analysis is done at constant heat power. The studied range of Ra* corresponds to the boundary layer regime in order to identify the overall response of the backflow at the top channel to the heat source location.

The computational domain includes a large tank surrounding the channel to model a channel with its environment, in order to avoid defining boundary conditions at the top and bottom ends of the channel. The strong coupling between the channel and its environment, which exists in all industrial applications as for the electric heaters, is then implicitly taken into account.

The outline of the paper is as follows. In Sections 2 And 3, we describe the geometrical and physical problem. Sections 4 and 5 details the numerical methods and the validation of the numerical model. Section 6 discusses the influence of the stratification establishing inside the tank on the flow channel. Section 7 analyses the effect of the spatial distribution of several heat sources. Concluding remarks are given in Section 7.