International Journal of Heat and Mass Transfer 118 (2018) 58–65

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Stationary convective regimes in a thin vertical layer under the local heating from below

A. Kondrashov *, E. Burkova

Perm State University, Russia

ARTICLE INFO

Article history:
Received 25 July 2017
Received in revised form 18 October 2017
Accepted 23 October 2017
Available online 22 November 2017

Keywords:
Natural convection
Local heat source
Convective patterns

ABSTRACT

Numerical and experimental results obtained for the steady-state flow in a thin vertical liquid layer heated locally from below are discussed. Based on these results, two laminar regimes at Rayleigh numbers from $Ra = 10^2$–10^7 have been distinguished. The first regime is characterized by a symmetric two-roll convective pattern. As soon as the Rayleigh number exceeds some critical value, the mirror symmetry is broken, and convective plumes tilt. A deflection of convective plumes from the vertical affects heat transfer conditions, and therefore this behaviour can be viewed as a second laminar regime. The location of the boundary between these two laminar regimes is considered in relation to the heat conduction at lateral boundaries, the aspect ratio of sides, the heater size, and the thickness of a working liquid layer.

1. Introduction

Since the mid-twentieth century, steady-state convective flows generated by a local heat source have been extensively studied. As early as 1976, researchers reported the results of numerical simulation obtained for the rectangular channel, one side of which was maintained at constant temperature and the other was subjected to heating produced by a compact heater [1]. They also revealed the complex relationship between convective flow patterns and heat transfer intensity.

In recent years, many research teams have examined the various locations of local heat sources inside a closed cavity [2–6]. This issue has been studied by both experiments [7,8] and numeric simulations [9,10]. It has been found that boundary conditions have a strong impact on convective flow patterns [11,12] and, consequently, on heat transfer intensity. Apart from a single heat source, different arrangements of heat sources have been explored [13–16] so that one can control and adjust the flow, e.g. by changing the direction of flow swirling and the number of convective rolls [17]. Such phenomenons were considered for both two-dimensional [6,18] and three-dimensional cases [19,20].

The problem of convection from a local heat source was primarily of fundamental interest, but then, with the rapid advances in the semiconductor industry, it becomes the problem of an applied character. For example, a comprehensive discussion of the convection cooling of multi-core processors can be found in [21]. As soon as several cores are combined into one element, the temperature distribution on the surface of a central processing unit will depend on the interference between heat flows from each heat source. Therefore, in order to optimize heat transfer processes and to avoid local overheating of the CPU housing, it is essential to gain reliable information on convection flow patterns that may occur in the environment.

In addition to removing the heat from electronic components, the convective flow from a localized heat source may assist in performing measurements. For example, there are accelerometers measuring external inertial factors through the deflection of convective plumes from the axis [22,23]. Engineers designing such devices face the problems of providing a steady-state flow regime over a wide range of parameters, and one way to achieve the desired goal is to limit the degrees of freedom [24,25]. The usage of thin layers makes it possible to improve the predictability of the flow and to facilitate an interpretation of factors leading to distortion of temperature fields in accelerometers [26].

Despite the high stability of convective flows in plane layers, it is known that, as in the three-dimensional case [27,28], different currents can arise in one and the same geometry depending on heating intensity [29–32]. The sensitivity of convective sensors is proportional to temperature field distortion in a working cell, and that is the reason why engineers are interested in maximum possible heating intensity, which is necessary to maintain the flow symmetry [33].

These investigations were applied in a number of applications, but most of them are focused on some partial cases of flows in the vessels of specific geometry, while the issue of generalization

* Corresponding author.
E-mail address: akon.psu@yandex.ru (A. Kondrashov).

https://doi.org/10.1016/j.ijheatmasstransfer.2017.10.096
0017-9310/

Nomenclature

x, y, z	coordinates	β	thermal expansion coefficient
H	hight of the cavity	χ	thermal diffusivity coefficient
L	length of the cavity	T_0	temperature on the upper boundary
d_z	liquid layer thickness	ΔT	overheating of the heater
R	radius of the heater	Ra	Rayleigh number
$\vec{g}$	gravity acceleration vector	Pr	Prandtl number
F_z	volumetric force responsible for viscous friction	A	aspect ratio
ν	kinematic viscosity	Nu	Nusselt number
u	flow velocity	q_{conv}	convective heat flux
P	pressure	$q_{thermal}$	thermal heat flux
T	temperature	α	Rayleigh number power coefficient
ρ	density	$C(x)$	correlation function value

of data on the convective patterns generated by local heat sources remains practically ignored in the literature [34]. In particular, to the best of our knowledge, too little attention has been paid to the problem considering the influence of heating intensity, heater and working cell sizes, and thermal boundary conditions on convective flow patterns and heat transfer in vertical plane layers, which is the reason for discussing it in our investigation.

In this paper, we present the results of experimental and numerical studies performed to assess the influence of flow organization conditions on the flow structure and the integral heat transfer characteristics. Analysis of the results revealed at least two convection regimes, which replace each other as the Rayleigh number increases. A two-roll symmetric flow pattern was observed in a shallow rectangular cell subjected to weak heating from below. When the intensity of heating exceeds a certain threshold value, the mirror symmetry is lost, which impedes the growth of the Nusselt number with increasing Rayleigh number.

2. Mathematical formulation of the problem

The steady-state flow in a thin vertical liquid layer bounded by solid walls and arising under the action of local heating from below is investigated. Our study incorporates laboratory experiments and numerical simulations.

The main flow in thin layers proceeds on the plane, and therefore we consider the two-dimensional problem of a rectangle of height H and width L. The rectangle is oriented so that its lower boundary coincides with the x-axis of the Cartesian coordinate system. The gravitational acceleration vector $\vec{g}$ is directed downward, and it is parallel to the vertical y-axis passing through the middle of the examined domain. The flow is generated with the aid of a heater of radius R, the center of which coincides with the origin of coordinates. A schematic diagram for the working cell is given in Fig. 1.

There were two main purposes of using plane geometry. Economy of the computational resources for multiply calculations was first. The second one was comparison test of two-dimensional model results with experimental data for a thin liquid layer. So, for three-dimensional modeling of a thin vertical layer, the equation of fluid motion is supplemented with a volumetric force F_z [35] responsible for viscous friction on the lateral walls of the cell in the direction z perpendicular to the xy plane of the main flow:

$$F_z = -12\frac{\nu u}{d_z^2}, \tag{1}$$

where ν is the kinematic viscosity of a working medium, u is the flow velocity modulus, and d_z is the thickness of a simulated layer.

We assume that the flow of fluid inside the computational domain is laminar and ignore the energy dissipation due to external friction forces. The flow in this approximation is described by the Navier-Stokes system of equations in dimensional form. This system consists of the momentum conservation equation

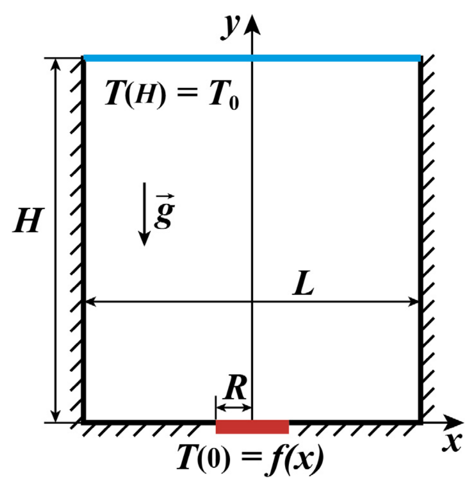

Fig. 1. Schematic diagram of the computational domain.

$$\partial_t u_i + \partial_j(u_i u_j) = -\frac{1}{\rho}\partial_i P + \nu\partial_j[(\partial_j u_i + \partial_i u_j)] + g\beta(T - T_0)e_y + F_z, \tag{2}$$

the heat transfer equation

$$\partial_t T + (u_i\partial_i)T = \chi\partial_i^2 T, \tag{3}$$

and the continuity equation

$$\partial_i u_i = 0. \tag{4}$$

The following notation is used: P-pressure, T-temperature, ρ-density, β-volume expansion coefficient, χ-thermal diffusivity, g-gravitational acceleration, T_0-initial temperature of the fluid.

Boundary conditions are as follows:

- the non-slip condition ($u = 0$) is imposed on solid boundaries;
- the upper boundary is assumed to be isothermal and serves as a cooler $T(y = H) = T_0$;
- either adiabatic conditions ($\nabla T_n = 0$) or the constant temperature equal to the upper boundary temperature T_0, are imposed on the side walls of the cavity ($x = \pm L/2$), depending on the implementation;
- temperature distribution representing localized heating inside the domain with a semi-width R is imposed on the lower boundary: