International Journal of Thermal Sciences 122 (2017) 85–91

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Heater shape effects on thermal plume formation

A. Kondrashov*, I. Sboev, P. Dunaev

Perm State University, Bukirev st. 15, Perm, Russia

A R T I C L E I N F O

Article history:
Received 27 September 2016
Received in revised form
25 July 2017
Accepted 13 August 2017

Keywords:
Natural convection
Thermal plume
Thermal boundary layer
Local heat source

A B S T R A C T

This paper presents the results of numerical modeling of the evolution of thermal plumes in a fluid layer heated locally from below. Special attention is paid to the development of a boundary layer that precedes thermal plume generation. Equations for natural convection are solved using the ANSYS Fluent software package. The calculated temperature fields are compared with the results of a full-scale experiment. The variation of parameters of the numerical model gives two mechanisms of heat transfer from a local heat source – conductive and convective. In the first case, uniform heating of fluid above the heater surface leads to the formation of a single plume. In the second case, the boundary layer near the heater edges experiences deformation that causes a convective jet to bifurcate. The purpose of the present work is to study the effect of heat source shapes (triangular, square, pentagonal and circular) on the boundary layer structure. For each heater shape, the critical value of the Rayleigh number determined. It is shown that bifurcation of a thermal plume is initiated when the Rayleigh number exceeds a critical value.

1. Introduction

Steady-state flows in enclosures [1–8] and the dynamics of convective jets or plumes in relation to heating conditions [9–13] are mainly considered in the problems of thermal convection from a localized heat source. However, the approaches dealing with these problems do not take into account the initial stage of convective flow evolution, which may play an important role in the cases when the processes associated with heat transfer proceeds more quickly than the generation of steady-state flows.

It is known that under specific heating conditions (at the initial stage of flow generation) the bifurcation of a thermal plume occurs [14,15], which in turn is able to affect heat transfer conditions in the system. For instance, in works [16–18], bifurcation has been observed at the stage of formation of transient or turbulent plumes. At sufficiently strong heating of the fluid by a localized heat source, a convective plume loses its stability because of the establishment of periodic vortex structures – puffs. The puffing is associated with the growth of Rayleigh-Taylor instability arising in the fluid flux adjacent to the heater surface. As a result, thermal plumes develop like pulses [16]. Besides, there is experimental evidence [19] that the vibration frequency of convective plumes is related to the generation of convective rolls on the heater boundary.

Apparently, the temperature and shape of a heater and the geometry of a working cavity should have a strong impact on the development of flow instability. Pioneer works devoted to investigating the effect of heat source shapes on heat transfer processes date back to the mid 20th century [20–22]. In particular, the analysis of the experimental results obtained for angulated objects [23] has revealed that their angles have only a negligibly weak effect on the integral heat transfer characteristics.

Recent publications [24–26] offer a theoretical description of steady-state flows adjacent to the surface of horizontal plates of different shape and size. For example, in Ref. [25], the authors conducted a study of heaters of triangle, square, hexagon and circle geometries and proposed two theoretical models. The results predicted by one of these models agree well with the experimental outcomes.

However, in the works mentioned above the mechanism responsible for the generation of thermal plumes has not been completely elucidated. Down to date the dependence of this process on heat source shape remains to be investigated. Problem solving by experiment is complicated by several factors. First of all, local heating is often produced by electrical current passing through a conductor [11,13], which, because of its inertia against temperature fluctuations, has a negative effect on processes that proceed faster than the heating-up time of a heat source. Furthermore, temperature field visualization techniques do not allow one to monitor temperature distribution inside the layer with required accuracy, and shadow and interference methods yield only an

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

http://dx.doi.org/10.1016/j.ijthermalsci.2017.08.012
1290-0729/

average flow pattern [15].

One way to simplify the study of hydrodynamic systems is to create a numerical model enabling the variation of problem parameters at low cost. This makes it possible to exclude any dependence of the physical properties of fluid on different external factors and to investigate the influence of boundary conditions on the basic properties of heat transfer. Because the examined system is highly idealized, the results of numerical modeling should be verified by comparing with the full-scale experiment data.

With the advent of high-performance computers, different physical and engineering processes can be successfully modeled using computer-aided engineering (CAE) systems. The spectrum of current tasks includes spatial modeling in continuum mechanics, calculation of strength and dynamics of solids, investigation of electromagnetic phenomena, etc. A comparison of the experimental data with the numerical calculation results or the results obtained by other models supports the reliability of the CAE systems. In our investigation, we use the ANSYS Fluent software package for computational fluid dynamics that ensures high-efficiency simulations.

Hence, it can be concluded that there is a gap in the understanding of flow generation above the localized heat sources of different shape. By varying heating intensity, heater dimension and shape, we have determined the regions in which the evolution of a convective flow follows two different scenarios: a) the fluid layer above the heater surface is heated uniformly, which results in the formation of a single thermal plume; b) the boundary layer loses its stability at heater edges, and therefore a plume starts to bifurcate.

2. Problem statement

We consider a flow in the liquid layer bounded by rigid walls. The flow occurs due to local heating from below. A computational domain (Fig. 1) has the form of a rectangular parallelepiped of height h with a square base of side l. The cavity is oriented so that its lower side lies in the XY-plane of a Cartesian coordinate system, and the gravitational acceleration vector g directed downward is parallel to the vertical Z-axis. The flow is generated by a heater of radius r, the center of which coincides with the origin of coordinates.

The evolution of a thermal plume is modeled by the finite volume method using the ANSYS Fluent software package. Fluid motion is assumed to be laminar, and, because of low flow velocities ($\sim 10^{-3}$ m/s), energy dissipation caused by internal friction is ignored. In this approximation, the flow is described by the system of dimensional Navier-Stokes equations containing:

- equation of momentum balance

$$\partial_t u_i + \partial_j(u_i u_j) = -\frac{1}{\rho}\partial_i P + \nu\partial_j\left[(\partial_j u_i + \partial_i u_j)\right] + g\beta(T - T_0)e_z \tag{1}$$

- heat conduction equation

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

- equation of continuity

$$\partial_i u_i = 0. \tag{3}$$

We use here the following notation: u – velocity vector, P – pressure, T – temperature, ρ – density, ν – kinematic viscosity, β – thermal expansion coefficient, χ – thermal diffusivity of fluid, g – gravitational acceleration vector, $T_0 = 300K$ – initial temperature of the fluid.

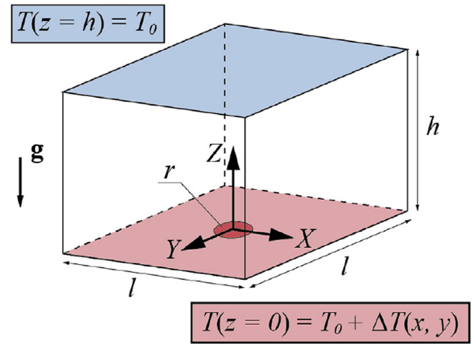

Fig. 1. Schematics of a computational domain with boundary conditions. The origin of coordinates coincides with the middle of the lower boundary.

A no-slip condition is imposed on solid boundaries. The upper boundary is taken to be isothermal and it serves as a cooler $T(z = h) = T_0$. The temperature distribution is prescribed at the lower boundary as

$$T(z = 0) = T_0 + \Delta T \cdot \left(\left[1 + e^{-\gamma\left(\sqrt{x^2+y^2}+r\right)}\right]^{-1} - \left[1 + e^{-\gamma\left(\sqrt{x^2+y^2}-r\right)}\right]^{-1}\right). \tag{4}$$

The function is taken so that the overheating of a heater ΔT is localized inside the area of radius r with the center at the coordinate origin. The parameter γ in equation (4) is responsible for the width of the region of transition from the local temperature difference of the source ΔT to the boundary temperature T_0.

Simulations are carried out in a computational domain of height $h = 24$ mm and side length $l = 28$ mm. The system of governing equations is solved at the nodes of a rectangular mesh with a maximal space step of $0.5mm$, with bias to the lower boundary for better resolution of gradients in the area of heating. Total number of mesh nodes in the domain is $3.04 \cdot 10^5$. The number of elements on the surface of a heat source of radius $r = 5.0$ mm reaches the value of $4.0 \cdot 10^2$.

The parametric analysis (Table 1) reveals that a two-fold increase in the number of nodes changes the time it takes the plume to reach the upper boundary by 0.6% and mean kinetic energy in the system at the steady state by 1.4% at most, and therefore a further mesh refinement becomes unreasonable. The choice of time step is based on the value of Courant number, not exceeding 0.8 during the simulations. We use SIMPLE algorithm for pressure-velocity coupling and the second-order discretization upwind scheme for the convection terms of each governing equation.

As a governing parameter characterizing the intensity of heating, a Rayleigh number is employed. The only parameter with the dimension of length that affects the development of a boundary layer at the initial stage of plume formation is the heater dimension r. Thus, the expression for Rayleigh number takes the form:

Table 1
Mesh validation.

Total number of nodes	Average kinetic energy, Pa
103200	$2.6 \cdot 10^{-6}$
200276	$5.67 \cdot 10^{-5}$
304290	$5.176 \cdot 10^{-5}$
650056	$5.102 \cdot 10^{-5}$