Experimental and numerical study of convective flow structure above heated horizontal plates of different sizes

I O Sboev, K A Rybkin and M M Goncharov
Perm State University,15 Bukireva str., Perm 614990, Russia

E-mail: ivan-sboev@yandex.ru

Abstract. Free thermal convection above heated plates formed in circles of different diameters is studied within our paper. Experimental and numerical studies are carried out in a three-dimensional tank filled with water under specified boundary conditions. The initial stage of thermal convection development is mainly considered. The study of liquid flow structure is particularly observed, as well as evolution of temperature field in a plane of the central section of the working sections. Within the study different heating conditions are considered: the diameter of the round source is in the range from 0.010 to 0.024 m, and the Rayleigh number varies in the range from $1.0 \cdot 10^2$ to $1.30 \cdot 10^5$. The standard system of Navier-Stokes equations is numerically solved by finite element method to study systematically convection from heat sources of various sizes by the software package COMSOL Multiphysics. Values of the control parameter are estimated in the course of the work. At the control parameter the transition from one flow regime to another one is observed. In addition, an experimental study is carried out to verify the results of numerical simulation. As a result, a qualitative agreement has been obtained between the calculated and observed flow structures during the experiments.

1. Introduction

By the present the structure and features of development of a united thermal plume created in a liquid layer due to localized heating in space remain one of the most studied manifestations of free thermal convection. Behavior of a flow with large value of the control parameter, for example, Rayleigh or Grashof number in environments with a small Prandtl number is still poorly studied, particularly in experimental part. The circumstance of destruction of the boundary layer by a convective flow near the surface of a heated plate of finite size is known in the available literature, and the plate itself and the free fall acceleration vector $g$ are perpendicular to each other in this circumstance [1-3].

One should point out that the evolution of the convective flow caused by the action of heated bodies of finite size is not only a fundamental problem concerning the transition regimes between flows of different sizes, but it also turns out to be of importance in the question of crystal growth, cooling and ventilation systems, etc. [4-5]. Thus, the description of flow scenarios and the knowledge how heat is transferred from the heaters surface with different characteristics will allow to represent better the ways of convective flows in localized heat sources systems and to control heat transfer processes.

In addition, the results of such studies will be able to find application, for example, in the design of modern cooling systems of microelectronic devices and components, as well as to provide evidence-based selection of optimal parameters of the geometric dimensions of heat exchange surfaces and non-isothermal processes [6-7].
It is also important to note yet the problem of heat and mass transfer from compact bodies is quite widely represented in the specialized literature, by present the available experimental and theoretical materials are not enough to create a generalized model that would allow to compare different heating conditions and properties of carrier fluids with operating various heat-releasing devices.

The present work can lead to prepare this model, in which the development of a convective flow over the surface of a heated round plate of different diameters is studied by numerical simulation in conjunction with a full-scale study. The classification of flow structures arising in the liquid under different heating conditions is also performed.

2. Formulation of the mathematical problem

The cavity bounded by a cylindrical surface is considered within numerical simulations. The computational grid is chosen irregularly, and its nodes are located at the vertices of triangles, and the distance between them is reduced while approximating to the axis of the heater, and nearby the source the linear density of elements of the computational grid is by many times higher than the density near the lateral boundary. The problem is solved in a three-dimensional formulation in a rectangular coordinate system $xy$, its beginning combines with the center of the heated area (figure 1). The direction of the gravitational acceleration vector $\mathbf{g}$ is chosen opposite to the $y$ axis, and the vector itself is orthogonal to the cavity bottom.

![Figure 1. Schematic representation of a cylindrical closed cavity filled with ideal liquid.](image)

The characteristics of the incompressible fluid flow in a homogeneous gravitational field are calculated using the laminar flow approximation in this statement of the problem. Dissipation in the medium under internal friction is not taken into account due to small temperature gradients and low flow rate energy. By this admission thermal convection is described by a system of Navier-Stokes equations in the Boussinesq approximation [8], written in the following form:

$$\partial_t \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\rho^{-1} \nabla p + \nu \nabla^2 \mathbf{u} + (1 - \beta T) \mathbf{g} e$$  \hspace{1cm} (1)

$$\partial_t T + \mathbf{u} \cdot \nabla T = \left( \frac{\kappa}{\rho C_p} \right) \nabla^2 T$$  \hspace{1cm} (2)

$$\nabla \cdot \mathbf{u} = 0$$  \hspace{1cm} (3)

In the above given equations, the following designations are used: $\mathbf{u}$ is the velocity of the liquid, $\mathbf{g}$ is the gravity vector, directed opposite to the unit vector $e$ of the $y$-axis, and $p$, $T$ are the pressure and temperature of the liquid. The properties of the simulated liquid, such as the kinematic viscosity $\nu$, the thermal expansion coefficient $\beta$, the density $\rho$ and the heat capacity $C_p$, correspond to the parameters of water under normal conditions and temperature of $T_0 = 300$ K [9]. When considering the development of the flow over time, the liquid at the initial moment is assumed to be stationary.

The no-slip condition $\mathbf{u} = 0$ is satisfied for velocity at solid boundaries, at which the liquid vector velocity vanishes. The surface bounding the liquid layer from above has a constant temperature $T_0$, and the round heating area centered at the origin bounded by a circle of diameter $d$ and corresponding to the
physical location of the copper heat exchanger has a temperature \( T_h > T_0 \). The temperature difference \( \Delta T = T_h - T_0 \) between the temperature of the heat source and the upper boundary of the cavity is responsible for the intensity of heating at the given parameters of the liquid and the geometry of the problem.

The thermophysical properties of the liquid are determined by the Prandtl number \( \sigma \):

\[
\sigma = \frac{\nu \rho C_p}{\kappa}, \quad (4)
\]

Rayleigh number is a common characteristic of heating:

\[
Ra = \frac{g \beta \Delta T}{\sigma} \left( \frac{\rho C_p}{\kappa} \right)^2 L^3. \quad (5)
\]

Due to the fact that the liquid mainly moves away from the lateral and upper boundaries of the cavity at the initial stage of convective flow development, it is assumed in our paper that the only significant parameter of length in the considered problem formulation is the value associated with the size of the heater. After the work [6] it is more convenient to use as a parameter the universal parameter \( L = \frac{s}{m} \) related to the surface area of the localized heat source \( s \) and its perimeter \( m \). This lets take into account not only the size of the heat source in such problems but also its shape and therefore one could summarize the results for different situations.

The body of equations (1)-(3) is solved numerically using the finite element method by software package COMSOL Multiphysics. The initial temperature of the undisturbed liquid as well as the upper isothermal solid boundary is taken to be \( T_0 = 300 \) K. The temperature difference in the heater area \( \Delta T \) takes values from 0.5 to 85 K, which correspond to the range of Rayleigh numbers \( Ra \) from \( 1.0 \cdot 10^2 \) to \( 1.3 \cdot 10^5 \). A grid with \( 7.67 \cdot 10^5 \) computational nodes has been selected according to the performed parametric study [10].

3. Experimental set-up

To verify and to visualize the convective processes occurring in the considered problem formulation, an experimental unit is schematically depicted in figure 2. The unit includes a cubic cavity filled with distilled water (\( T_0 = 300 \) K, \( \sigma = 6.9, \rho = 0.97 \) g/cm\(^3\)) with heat-insulated side walls and an open upper boundary. The height of the working layer of fluid exceeds at many times the diameter of the copper heat exchanger that is equal to \( d = 0.020 \) m. Heating is provided by a copper plate, which is located in the centre of the cavity bottom. The bottom side of the copper heat exchanger meets the resistor. To be able to control the source temperature inside the heater there was a differential thermocouple junction. Measurement and regulation of the resistor temperature is carried out by microvoltmeter with polling frequency of 4 Hz, a power regulator, and stabilized DC source. The signal received from the measuring devices is received and processed by computer. The cavity has been placed in space so that the vector \( g \) remains perpendicular to the copper heater plane.

The heat source is illuminated from above through the open boundary by a laser knife with a wavelength of 532 nm, that allows to observe the trajectory of motion of light-scattering polyamide particles suspended in the liquid (\( \rho_1 = 1.05 \) g/cm\(^3\)), as well as to visualize the flow by adding a three-percent aqueous glycerin solution containing a fluorescent dye (\( \rho_2 = 1.05 - 1.10 \) g/cm\(^3\)).

Shooting in the laser knife plane \((z = 0)\) is taken with a digital camera with a frequency of 3 frames/s. The results of observations are images of the particle trajectories and instantaneous concentration fields above the heater at different points in time.

4. Results and discussion

The evolution of the temperature field in the plane of the axial section of the cylindrical cavity \((z = 0)\) has been examined for different dimensions \( d \) of a heated plate within systematic numerical calculations of the thermal perturbations development in a fluid with Prandtl number \( \sigma = 6.9 \) (water). Heat sources with diameters of 0.020 and 0.024 m are considered in the simulation.
Figure 2. The general scheme of the experimental set-up: the cubic tank, on its lower bottom there is a localized heater in the form of a copper plate with a diameter \( d = 0.020 \) m; microvoltmeter (µV); microcontroller used to maintain required heating temperature (PID); DC power supply; computer; laser knife (DPSS laser, 532 nm); digital CMOS camera.

As a result of calculations it is shown that a single axisymmetric heat torch for small Rayleigh numbers \((100 < Ra < 500)\) develops above a circular heater with a diameter \( d = 0.010 \) m \((L = 0.0025 \) m\). The axisymmetric plume has been well studied at present. It is known that the initial stage of the formation of such a flow is accompanied by the destruction of the boundary layer formed near surface of the heat source and by development of a flow formed as a torus above the heater. The axis of a generated vortex coincides with the vertical axis \( y \), and the diametrical temperature distribution near the source surface has a single global maximum localized at the origin. The evolution of boundary layer and subsequent development of such a thermal plume are described in details in various papers \([11, 12]\).

It is only important to note that the flow rate under the considered convection regime is relatively small, for this reason when increasing the distance to surface of the heated plate the liquid temperature decreases inside the heated liquid column as noted in \([13]\) inversely proportional to the vertical coordinate \( y \).

Figure 3. Observed in the plane of laser knife \((z = 0)\) flow structure above the round heater with \( L = 0.005 \) m \((a)\). The picture has been obtained by combining several photos and shows the structure of developing from 3 to 9 seconds after start of thermal convection \((L = 0.005 \) m, \( Ra = 5.37 \cdot 10^4 \) \((\Delta T = 35 \) K), \( \sigma = 6.9)\); evolution of the concentration field in the plane \( z = 0 \) under the same heating conditions \((b)\).
The boundary layer deformation at the stage of thermal plume development was studied in many papers. It was also observed in the present work (figure 3.a). Therefore, it is expected that a flow change also affects the temperature distribution in the laboratory resulting in two global peaks at temperature profile [10]. The fluid flow has a vortex structure as a result of boundary layer separation, and the main rotation places at periphery of a heated area. The vortex begins to emerge under the action of buoyancy force dragging the heated liquid. The resulting horizontal pressure gradient leads to a decrease in the vortex ring diameter and a single thermal plume is formed above the heater over time (figure 3.b). It should be said that before the moment of combining two streams into a single central plume the fluid near the origin is also rotated resulting in a vortex ring above the heat source, but it is smaller in comparison with the vortex ring on the periphery. In addition, the rotation of the small vortex ring has the opposite direction.

According to the calculations there are changes in the distribution of a vertical component of velocity $u_y$ of fluid motion by forming this thermal plume. Besides there is change in temperature profile above the source surface. Obviously, the highest flow rate is achieved at the periphery of the heater, where at first mechanical equilibrium violation occurs. It is observed following from the above if the boundary layer is deformed, the flow nature created by the compact heat source also changes, that indicates the presence of a new convection regime.

Similarity in the distribution of velocity $u_y$ and temperature $T$ above the heater surface is noted, in this regard the plane of the axial section of the cavity ($z = 0$) is more convenient to study various convection regimes by observing structure of the temperature field $T(x,y)$. In this representation one three-dimensional vortex ring is splitting into two flat convective streams (figure 4.a). In turn, it is more convenient to register the velocity or concentration field in the plane $z = 0$ during the experiment, so the criterion for classification of convection regimes is the profile shape of the vertical component $u_y$ of velocity along the line $y = 0.002$ m. It is to mention that the correspondence between structures of temperature (theory) and concentration (experiment) fields is especially evident only at the initial stage of plume development (figure 4.b).

![Figure 4](image_url)

Figure 4. A three-dimensional image of the isothermal surface in a fluid and the corresponding temperature field $T(x,y)$ in the vertical plane (a); instantaneous temperature field in the plane $z = 0$ ($L = 0.005$ m, $Ra = 5.0 \cdot 10^4 (\Delta T = 32.5$ K)) (b). By visualizing the heating conditions correspond to the selected control parameter $Ra$, but it is not possible to observe the third stream in the center of the layer in the experiment. Discrepancy of results of calculations and experiment could be caused by idealization of numerical models and factor of impurities in glycerin during visualization with the fluorescent dye.

Convective regimes research is carried out under the following heating conditions in this paper: $d = 0.020$ m ($L = 0.005$ m, $Ra$: $1.0 \cdot 10^4 (\Delta T = 6.5$ K), $5.0 \cdot 10^4 (\Delta T = 32.5$ K), $1.30 \cdot 10^5 (\Delta T = 84.6$ K)) and $d = 0.024$ m ($L = 0.006$ m, $Ra = 1.3 \cdot 10^5 (\Delta T = 48.9$ K)).
Thus, the boundary layer is also destroyed by the flow with a diameter $d = 0.020 \text{ m}$ arising near the source boundaries with growth of the control parameter ($Ra = 1.0 \cdot 10^4 \left(\Delta T = 6.5 \text{ K}\right)$) and two streams begin to develop in the liquid (figure 4.b1). Sometime after having begun these streams collapse due to a horizontal component of pressure forces, forming a united central convection stream.

At $Ra = 5.0 \cdot 10^4 \left(\Delta T = 32.5 \text{ K}\right)$ two streams on the periphery appear above the round heater of the same radius as described above, but in this case, the boundary layer in the center of liquid layer has time to be destroyed by an additional flow formed in a united smaller stream (figure 4.b2). Development of such a structure also represents the union of these streams into a united thermal plume in the center of liquid layer. For the same heater at $Ra = 1.30 \cdot 10^5 \left(\Delta T = 84.6 \text{ K}\right)$ the central heater part is also deformed after the boundary layer has been split, but unlike the previous case another pair of streams is formed in fluid. However, smaller plumes for the selected size of the source have had time to reach each other until all the streams are combined into a single plume (figure 4.b3).

One should note that the structure of the temperature field after uniting in several thermal plumes has qualitative coincidence and it does not depend on the boundary layer changes those have happened before in all considered cases at a fixed size of the heat source. In other words, described sequence of stages of boundary layer development ultimately leads to formation of qualitatively similar convective structures.

**Figure 5.** Temperature field evolution in the plane $z = 0$ above a circular heater with $L = 0.006$ m and $Ra = 1.30 \cdot 10^5 \left(\Delta T = 48.9 \text{ K}\right)$. The resulting convection regime is characterized by the presence of periodic stability loss even after formation of a central united thermal plume.

Increasing size of a heated area to $d = 0.024$ m leads to the following convective regime (figure 5). The boundary layer evolution repeats the above scenario during the first five seconds, but the small-
scale streams at the same control parameter are combined primarily with large-scale plumes, and only then both streams approximate to each other, forming a united thermal plume.

A pair of smaller scale (in comparing with the main stream) plumes is formed again after formation of a united large-scale stream in the same place above the heater in which there was a repeated boundary layer deformation. An important feature of the described regime is the emergence and association of small-scale plumes called in the literature with the word "bulge". It should be emphasized that such a process is periodically repeated with time.

5. Conclusions
The formation of a convective stream at localized heating depending on heating conditions was considered within this paper.

One of the main results is the case when the size of heat source increases, in this connection the variety of convection regimes observed in fluid increases, too. Thus, a united convective stream in liquid develops at small values of the control parameter by a localized heater, the boundary layer is deformed in the central part of a heated area at the initial stage of stream formation. With an increase at the Rayleigh number, there was a change in the nature of boundary layer deformation where the most intense flow occurs not in the center, but on periphery of the heat source. The calculation of flow evolution at large values of Rayleigh numbers ($Ra \sim 10^5$) shows that over time the concentric vortex ring with the heater collapses into a united thermal plume. However, the instability of the boundary layer may periodically occur under such regimes.

The described results of the study are in agreement with many similar studies, as well as confirmed by the experimental observations. Thus, the phenomenon of qualitative changes in the flow character at the thermal plume formation stage is described but insufficiently studied in the literature. This can be calculated in the framework of laminar flow approximation with accurate agreement with the full-scale experiment using direct numerical simulation.

Acknowledgments
The paper has been performed by support of RFBR grant (project No. 18-31-00165).

References
[1] Plourde F, Pham M V, Kim S D and Balachandar S 2008 J. Fluid Mech. 604 99–123
[2] Hattori T, Bartos N, Norris S E, Kirkpatrick M P, Armfield S W 2013 Exp. Therm Fluid Sci. 46 139–150
[3] Hattori T, Norris S E, Kirkpatrick M P and Armfield S W 2013 J. Fluid Mech. 732 105–127
[4] Maruyama S, Ohno K, Komiya A and Sakai S 2002 J. Cryst. Growth 245 (3–4) 278–288
[5] Pretot S, Zeghmati B and Le Palec G 2000 Appl. Therm. Eng. 20 (10) 873–891
[6] Bergman T L, Incropera F P and Lavine A S 2011 Fundamentals of heat and mass transfer (John Wiley & Sons) p 1050
[7] Kreith F, Manglik R and Bohn M 2010 Principles of Heat Transfer (Cengage Learning) p 696
[8] Shevtsova V, Ryzhkov I I, Melnikov D E, Gaponenko Y A and Mialdun A 2010 J. Fluid Mech. 648 53–82
[9] Sboev I O, Kondrashov A N, Rybkin K A, Burkova L N and Goncharov M M 2018 Micrograv. Sci. Tech. 30 (4) 369–375
[10] Kondrashov A, Sboev I and Dunaev P 2017 Int. J. Therm. Sci. 122 85–91
[11] Cagney N, Newsome W H, Lithgow-Bertelloni C, Cotel A, Hart S R and Whitehead J A 2015 Geochim. Geophys. Geosyst. 16 (3) 579–599
[12] Moses E, Zocchi G and Libchaberer A 1993 J. Fluid Mech. 251 581–601
[13] Davaille A, Limare A, Toutou F, Kumagai I and Vatetville J 2011 Exp. Fluids. 50 (2) 285–300