Numerical simulation of natural convection in wedge-shaped domain with isothermal free surface

N A Yaroslavtseva and N G Ivanov

Department of Fluid Dynamics, Combustion and Heat Transfer, Peter the Great St.Petersburg Polytechnic University, Polytechnicheskaya str. 29, 195251, St.Petersburg, Russia

E-mail: nadya_yar94@mail.ru

Abstract. The contribution deals with 2D laminar unsteady natural convection in a wedge-shaped reservoir model induced by the isothermal surface heating of a water basin being colder than surrounding atmosphere. The problem formulation considered corresponds to large-scale convection development during a cloudy day when the solar radiation impact is negligible. Numerical simulation was performed using an in-house Navier-Stokes code SINF. The focus of the paper is on the accurate resolution of the initial period of the convective circulation pattern development. The dependence of the predicted convective structure on the computational domain size as well as on the boundary condition at the free surface is analysed. The influence of geometry on the buoyancy-induced flow formation is discussed.

1. Introduction

Natural convection induced by heating or cooling of the shallow nearshore zone with non-uniform depth plays an important role in transportation of nutrients, pollutants and chemical species. Started in 1980s, numerous field experiments in lakes or coastal waters showed stable large-scale flows forming in this zone. These flows could have an impact on the pollutants residence time in the nearshore regions. For example, experiments by James and Barko performed in 1988 [1] examined nighttime convective circulation patterns; the effect of the patterns on the transport of phosphorus between littoral and pelagic zones was significant. Later contributions also pointed to the effects of the nearshore convection, e.g., measurements by Niemann et al. [2] allowed to suppose that the convective circulation is the origin of the reef-borne material transport to the deeper strata of the Red Sea.

The origin and development of convective circulation patterns detected in the field experiments have been studied also in many contributions operating with scaling analysis and numerical simulation, see, e.g., the reviews in [3, 4]. Many papers reported the analysis data for test problems with imposed horizontal temperature gradient that drives the flow. Apart from this scenario, other contributions considered different mechanisms of horizontally uniform thermal forcing when the geometry plays the key role in the formation of the convective flows. Thus, Lei and Patterson [5] carried out an analysis of the unsteady natural convection in a triangular domain induced by the absorption of solar radiation. The horizontally uniform internal heating source due to radiation was set in this case, and the principal driving force for the circulation came from the heat exchange on the sloping bottom. For the same triangular domain, Lei and Patterson [6] investigated also the case of nighttime cooling through the water surface, and a heat flux condition quantified the heat loss. Later Bednarz et al. [7] considered natural convection in a wedge-shaped domain for periodic thermal...
forcing corresponding to daytime heating and nighttime cooling at the water surface. Numerical simulation in [7] has modeled the flow development relevant to cloudy atmospheric conditions without any influence of the solar radiation and the flow is driven by the ambient temperature changes.

Bednarz et al. [8, 9] confirmed experimentally development of convection in a wedge-shaped reservoir model. In [8] the experimental model was cooled from above, while in [9] the quasi-steady natural convection flow in the domain subject to periodic thermal forcing at the water surface was considered. Yu et al. [4] performed a scaling analysis for the case of the iso-flux surface heating with the focus on the horizontal conduction effects. On the contrary, Mao et al. [3] considered the isothermal surface heating that corresponds to the overcast day conditions when the air temperature is higher than the water body temperature and the solar radiation impact is negligible. In a rectangular reservoir, an isothermal warm surface results in a stable thermal stratification without any motion, so that the simplified isothermal surface heating model points to pure effects of the wedge shape.

The current study considers the wedge-shaped reservoir model with the same surface boundary condition as was adopted in [3]. Numerical simulation of two-dimensional (2D) transient natural convection induced by isothermal surface heating was performed for various computational domains, with the focus on the accurate resolution of the initial period of the convective circulation pattern development.

2. Problem definition and computational aspects

The 2D wedge-shaped reservoir model considered is shown in figure 1, $x$ is the distance from the shoreline and $y$ is the vertical position, the origin of the coordinate system is set at the shoreline. The flat free surface approximation is used. The water depth varies with the offshore distance until the maximum depth of the reservoir, $h$, is reached. The bottom slope $A = \tan \alpha$ is one of the parameters of the problem. Two cases with the bottom slope values of $A = 0.1$ and $A = 0.2$ are considered in the current paper.

To the right of the region with the sloping bottom the water depth is uniform. To make sure that the length of the region with the horizontal bottom is enough to avoid the influence of boundary BC on the nearshore convection, a special study of the effect of the distance FC on the solution was performed, FC length values of $10h$ and $15h$ were considered. To avoid the singularity problem near the shoreline, an auxiliary vertical boundary AD is introduced, $AD = 0.1h$, and small triangle to the left of AD is excluded from the computational domain.

Development of convective circulation patterns from the initial stationary isothermal state is modeled, and the initial temperature is assumed as the reference temperature, $T_{ref}$. Starting from the time instant $t = 0$, heating is imposed at the free surface, and the boundary is treated as isothermal with the uniform temperature $T_b$. The reference temperature difference is defined as $\Delta T_{ref} = T_b - T_{ref}$.

The buoyancy-induced transient flow is simulated using the 2D Navier-Stokes and energy equations. The incompressible fluid with constant physical properties is considered. The buoyancy effects are represented with the Boussinesq approximation. Assuming that the buoyancy velocity $V_b = (g\beta\Delta T_{ref}h^3)^{1/2}$ is the velocity scale, here $g$ is the gravity acceleration and $\beta$ is the thermal expansion coefficient, and the ratio $h/V_b$ is the time scale, the governing equations in the non-dimensional form are written as:
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \sqrt{\frac{Pr}{Ra}} \frac{\partial \Delta u}{\partial x},
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \sqrt{\frac{Pr}{Ra}} \frac{\partial \Delta v}{\partial y} + T
\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \frac{1}{\Pr} \frac{\partial \Delta T}{\partial t}

where \( u \) and \( v \) are the horizontal and vertical velocity components, respectively; \( p \) is the pressure and \( T \) is the temperature. The equations comprise two non-dimensional parameters: the Rayleigh number, \( Ra = \frac{g\beta\Delta T_{ref}h^{3}}{v\alpha} \), and the Prandtl number, \( Pr = v\alpha \), here \( v \) is the kinematic viscosity and \( \alpha \) is the thermal diffusivity. The current contribution considers one set of these parameters: \( Pr = 7, Ra = 10^{3} \).

The initial conditions are \( u = 0, v = 0, T = 0 \). The boundary conditions are as follows. The sloping and flat sections of the bottom are treated as adiabatic walls with the no-slip boundary conditions, so that \( u = 0, v = 0, \partial T/\partial n = 0 \). The same boundary conditions are set at the auxiliary boundary AD. For the isothermal flat free surface AB two cases were considered: (1) the stress free condition, so that \( \partial u/\partial y = 0, v = 0, T = 1 \), and (2) the no-slip condition, so that \( u = 0, v = 0, T = 1 \). The conditions at the distant boundary BC that connects the domain considered with the larger central part of the reservoir were varied. First, an open boundary condition was set at the line BC, with an approximation of the uniform pressure distribution over the boundary; in case of inflow the temperature value \( T = 0 \) is set. Second, a symmetry boundary conditions were set at the line BC, so that \( u = 0, \partial v/\partial x = 0, \partial T/\partial n = 0 \).

The computations were performed using the structured version of the in-house 3D steady/unsteady Navier-Stokes code SINF (Supersonic to Incompressible Flows) being under development since 1993 [10-12]. This version of the code is based on the second-order finite-volume spatial discretization using the cell-centered variable arrangement and body-fitted block-structured grids [10]. The artificial compressibility (AC) technique and/or the SIMPLC method are used for the pressure-velocity coupling in the case of incompressible fluid flows (see [11] for details). The viscous fluxes are evaluated on the basis of central differences. Weighted corrections are calculated on the basis of the Rhie and Chow approach. Three-layer second-order scheme is implemented for physical time stepping. The code is parallelized based on the MPI standard and the domain decomposition strategy according to the grid block structure [12].

In the current contribution, the QUICK scheme was applied to compute the convective fluxes. The SIMPLC technique was used in more stable case of the no-slip surface condition. For the cases with the stress-free surface condition the coupled AC/SIMPLC algorithm was activated.

Figure 2. The effect of solution convergence at each time step on the velocity evolution at the monitoring point located at \( x = 10.16, y = 0.36 \); final scenario is: 5000 iterations per time step from the start to the 2nd time units; 2500 iterations from the 3rd to the 10th time unit, 1500 iterations from the 11th to the 20th time unit, 750 iterations from the 21st to the 50th time unit, then 500 iterations.
The uniform meshes were used for the computations. A mesh dependency study was performed for the case of $FC = 10h$ using seven successively refined grids, the finest grid size was more than 40 thousand cells. The solutions obtained with the mesh of $451 \times 75$ nodes ($33,300$ cells) were accepted as mesh-independent, and all the data presented in the paper were computed using spatial resolution of such quality. The time step of 0.05 time units was chosen after several time step dependency tests. Finally, special attention was paid on the effect of solution convergence at each time step. To provide sufficient convergence during initial period of surface heating, large amount of iterations at each time step was necessary (see figure 2). Later, when the temperature gradients are reduced, it is possible to use less number of iterations, the final scenario of the iteration variation is given in figure 2.

3. Results and discussion

Formation of the buoyancy-induced flow in the region with the sloping bottom is illustrated in figure 3 where temperature and velocity fields are shown for $A = 0.1$ at three successive time instants. A zone with pronounced temperature gradient is formed near the heated free surface. It interacts with the inclined bottom at low $x$-values, and slight thermal forcing is generated in the horizontal direction driving a convective pattern with clockwise water motion. The intensity and extension of this pattern grow in time significantly during the period illustrated, though typical velocity values in the pattern are much less than the scale $V_b$ value, and even at $t = 30$ do not exceed 1% of the buoyancy velocity.

Figure 4 gives quantitative information on time evolution of the convective pattern near the shoreline: the plot shows the maximum value of $x$-velocity that is extracted at each time instant from the vertical line $x = 2$. The velocity values given are very low as the monitoring line is close to the left boundary of the convective pattern. The motion intensity in this region grows during the initial period of about 20 time units, and then velocity values decrease gradually. The results computed correspond to the data obtained previously by Mao et al. [3] also shown in the figure. Some difference between the current results and the data from [3] could be attributed to the auxiliary vertical boundary AD introduced in the current study. Note that to compare the velocity values, the data from [3] were rescaled: $t = t_{(3)}(Ra \cdot Pr)^{1/2}$, $u = u_{(3)}(Ra \cdot Pr)^{-1/2}$.

As mentioned above, a special test case with the no-slip condition at AB was computed. If the no-slip boundary condition is set at the free surface, convection behavior is qualitatively the same as in case of the stress-free condition: the size of the convective cell is comparable in both the cases as the streamline patterns for the no-slip condition (not shown) looks like the cells given in figure 3 a,c,e. However, as expected, the velocity values in the fluid body are approximately two times lower in case of zero velocities at the surface, see blue rhombs in figure 4.

![Figure 3](image)

**Figure 3.** Temperature fields with streamlines (left) and $x$-velocity component distributions (right) at (a, b) $t = 10$, (c, d) $t = 20$, and (e, f) $t = 30$; $A = 0.1$
Figure 4. Evolution of the maximum value of $x$-velocity at $x = 2$; (1) – stress-free surface boundary, symmetry condition at BC; (2) – stress-free surface boundary, pressure condition at BC; (3) – no-slip surface boundary, pressure condition at BC; the computed results are compared with the data from [3]; $A = 0.1$

Figure 5. Temperature fields with streamlines for (a, c) $FC = 10h$ and (b, d) $FC = 15h$: (a, b) symmetry and (c, d) pressure condition at the distant boundary BC; $t = 30$

Figure 6. Temperature fields with streamlines (left) and $x$-velocity component distributions (right) at (a, b) $t = 10$, (c, d) $t = 20$, (e, f) $t = 30$; $A = 0.2$

Figure 4 illustrates also the effect of the condition at the distant boundary BC on the convective pattern in the nearshore region. In reality the nearshore zone is connected with a much larger central part of the reservoir, and the convective pattern could be influenced by the processes far from the shoreline. The current model problem does not consider these long-distance processes, and it is supposed that the open boundary does not affect the flow in the region with the sloping bottom. Figure 4 gives an evidence that at $x = 2$ the solutions with the symmetry and the pressure conditions are the same at least till the 80th time unit. However, as it is visible from figure 5, if the pressure boundary condition is set, the second convective structure is generated near this boundary that was detected also.
in [3]. Remarkably, at $t = 30$ considered in figure 5 two convective cells do not interact. If the length of the region with the uniform water depth is increased, this convective structure is shifted to the right (compare figures 5c and 5d). It could be concluded that the length of the region with the horizontal bottom of $10h$ is enough to avoid the influence of the right boundary on the near shore convection.

Finally, the influence of bottom slope value on the convective cell formation is studied. A comparison of figure 6 where the results for $A = 0.2$ are given with similar plots presented in figure 3 for $A = 0.1$ allows to conclude that the convective cell size increases with the bottom slope reduction. However, for less bottom slope the intensity of the convective pattern is lower than for higher $A$. The reason for it is in higher value of horizontal temperature gradient in case of larger bottom slope.

4. Conclusions
Natural convection in a 2D wedge-shaped reservoir model induced by isothermal surface heating at the water surface was simulated numerically using an in-house Navier-Stokes code SINF. Transient large-scale convective pattern with clockwise water motion was detected in the region with the sloping bottom, typical velocities did not exceed 1% of the buoyancy velocity. It was shown that the length of the region with the horizontal bottom of $10h$ is enough to avoid the influence of the right open boundary on the nearshore convection. The influence of the bottom slope value on the convective cell formation was studied: the convective cell intensity decreases with the bottom slope reduction.

References
[1] James W F and Barko J W 1991 Estimation of phosphorus exchange between littoral and pelagic zones during nighttime convective circulation Limnol. Oceanogr. 36 pp 179–87
[2] Niemann H, Richter C, Jonkers H M and Badran M I 2004 Red sea gravity currents cascade near-reef phytoplankton to the twilight zone Mar. Ecol.-Prog. Ser. 269 pp 91–99
[3] Mao Y, Lei C and Patterson J C 2012 Unsteady nearshore natural convection induced by constant isothermal surface heating J. Fluid Mech. 707 pp 342–68
[4] Yu P, Patterson J C and Lei C 2015 A scaling analysis of transient natural convection in a reservoir model induced by iso-flux heating J. Fluid Mech. 764 pp 219–49
[5] Lei C and Patterson J C 2002 Unsteady natural convection in a triangular enclosure induced by absorption of radiation J. Fluid Mech. 460 pp 181–209
[6] Lei C and Patterson J C 2005 Unsteady natural convection in a triangular enclosure induced by surface cooling Int. J. Heat Fluid Flow 26 pp 307–21
[7] Bednarz T P, Lei C and Patterson J C 2009 Unsteady natural convection induced by diurnal temperature changes in a reservoir with slowly varying bottom topography Int. J. of Thermal Sciences 48 pp 1932–42
[8] Bednarz T P, Lei C and Patterson J C 2008 An experimental study of unsteady natural convection in a reservoir model cooled from the water surface Exp. Therm. Fluid Sci. 32 pp 844–56
[9] Bednarz T P, Lei C and Patterson J C 2009 An experimental study of unsteady natural convection in a reservoir model subject to periodic thermal forcing using combined PIV and PIT techniques Exp. Fluids 47 pp 107–17
[10] Smirnov E M 2000 Solving the full Navier-Stokes equations for very-long-duct flows using the artificial compressibility method Proc. of ECCOMAS 2000 - European Congress on Comp. Methods in Appl. Sciences and Engin. (Barcelona, Spain, 11 - 14 September 2000) pp 1–17
[11] Smirnov E M and Zaitsev D K 2004 Computations of internal flows using an artificial-compressibility solver enhanced with an elliptic pressure-correction procedure Proc. of ECCOMAS 2004 - European Congress on Comp. Methods in Appl. Sciences and Engin. (Jyväskylä, Finland, 24 - 28 July 2004) pp 1–13
[12] Zaitsev D K, Shchur N A and Smirnov E M 2009 Numerical simulation of 3D turbulent flows around bodies subjected to vortex-induced and forced vibration Lecture Notes in Computational Science and Engineering 67 pp 347–54