A NUMERICAL STUDY OF CONVECTION FLOW IN AN OPEN WATER CAVITY

Paulo Cesar Mioralli[a], mioralli@ifsp.edu.br
Vicente Luiz Scalon[b], scalon@feb.unesp.br
Elson Avallone[a], elson.avallone@ifsp.edu.br
Alcides Padilha[b], padilha@feb.unesp.br

[a] Instituto Federal de São Paulo – IFSP, Av. Pastor José Dutra de Moraes, 239, Distrito Industrial Antônio Zácaro, Catanduva-SP, 15808-305, Brazil
[b] Universidade Estadual Paulista "Júlio de Mesquita Filho" – FEB/UNESP, Av. Eng. Luiz Edmundo Carrijo Coube, 14-01, Vargem Limpa, Bauru-SP, 17033-360, Brazil

Abstract. *The development of new technologies of solar energy and its optimization necessarily involves studying the behavior of the energy stored inside the thermal reservoir. A simple way to estimate this behavior is through the analysis of a flow in a cavity with different inlet and outlet of the mass flow. In this study was analized water flow inside the cavity with two openings. The numerical simulation was performed using the computer package open source OpenFOAM®, considering a two-dimensional flow. Simulations were performed considering the effects of the flow field forces and excluding these effects, represented by Richardson Number. The results obtained for the velocity and temperature profiles in the tested cases were presented and the influence of the field forces on the thermal stratification of the water was observed. The results obtained allow to evaluate the influence of the parameters velocity and temperature for the characterization of the obtained thermal stratification inside the solar heater reservoir.*

Keywords*: thermal energy, ventilated cavity, computational fluid dynamics.*

1. NOMENCLATURE

Variables		u, v	*components of velocity (m/s)*
c_p	*constant pressure specific heat (J/kg K)*	U	*velocity (m/s)*
g	*gravitational acceleration (m/s^2)*	x, y	*Cartesian coordinates (m)*
Gr	*Grashof number* $(Gr = g\beta(T - T_{in})L_c^3/\nu^2)$		
H	*height of cavity (m)*	***Greek symbols***	
k	*thermal conductivity (W/m K)*	α	*thermal diffusivity (m^2/s)*
L	*length of cavity (m)*	β	*thermal expansion coefficient (K^{-1})*
L_c	*characteristic length (m)*	ν	*kinematic viscosity (m^2/s)*
p	*pressure (N/m^2)*	ρ	*density (kg/m^3)*
Pr	*Prandtl number* $(Pr = \nu/\alpha)$		
Re	*Reynolds number* $(Re = U_{in}L_c/\nu)$	***Subscripts***	
Ri	*Richardson number* $(Ri = Gr/Re^2)$	in	*inlet*
t	*time (s)*	w	*wall*
T	*temperature (K)*		

2. INTRODUCTION

The fluid flow and heat transfer processes are analyzed in cavities targeting different applications engineering. In recent decades, some researchers have focused on research in ventilated cavities or cavities with openings mass flow, encompassing both the natural and forced convection processes. A literature review shows several heat transfer studies by mixed convection in cavities like Guo and Sharif (2004), Saha et al. (2006), Khanafer et al. (2007), Saha at al. (2007) and Basak et al. (2009) due to the practical significance of the subject for closed or ventilated cavities.

Many works about ventilated cavities are also performed considering any heat generating source or some kind of obstacle in the cavity. In this line of research, highlights are the works of Radhakrishnan et al. (2007), Rahman et al. (2008), Ghasemi and Aminossadati (2008), Rahman et al. (2012) and Belmiloud and Sad chemloul (2015). Ventilated cavities with no elements inside are less common, but no less important, as the studies by Mahmoudi et al. (2010), Sourtiji et al. (2011) and Sourtiji et al. (2014). In general, three kinds of convection, natural, forced and mixed, are present in studies of ventilated cavities without internal elements. The mixed convection is greatly found in studies of atmospheric flows, heat exchangers, lubrication technology, drying technology, electronics cooling and solar energy storage.

Concerning the solar energy, the development of new technologies and their optimization necessarily involves studying the behavior of the energy stored within the thermal reservoir. A simple way to behavior analysis is through the study of the flow in a cavity with different inlet and outlet opening of the mass flow. In this work, the water flow inside a cavity with two openings was numerically studied using the open source computer package OpenFOAM® for simulation. The simulations were performed for different situations, with a predominance of natural convection, forced convection and mixed convection. The velocity and temperature profiles were obtained in the cavity and the influence of these parameters concerning the water thermal stratification was evaluated. The results of such analysis may assist in characterizing the thermal stratification within a solar heater water tank.

3. PROBLEM FORMULATION

Figure 1 illustrates the two-dimensional cavity used in the study. The cavity has square geometry ($H = L = 0.1m$) and has two openings: one for inlet and another for fluid outlet. Both openings have size of 0.01m. The water flow enters through the upper opening cavity and exits through the lower, both located on the left side of the cavity. The four walls of the cavity are maintained at a constant temperature T_w, while the inlet fluid temperature T_{in} is kept constant with a value higher than the wall temperature. The flow is considered incompressible and the fluid is Newtonian. It is negligible the effect of viscous dissipation. The fluid thermophysical properties are constant and the field effects are estimated by the Boussinesq approximation for accounting of buoyancy forces.

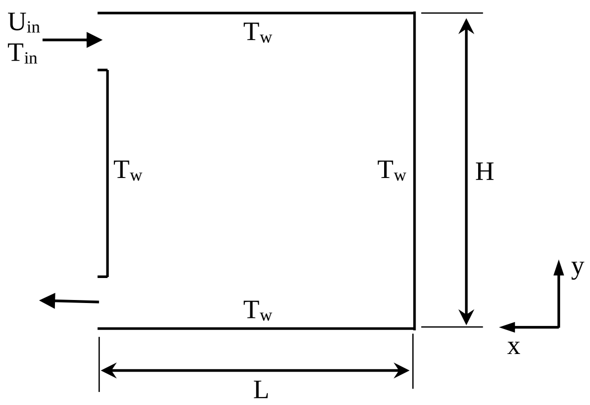

Figure 1. Illustration of cavity with inlet and outlet ports.

For the foregoing considerations, the transport equations are:

Continuity:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

Momentum:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + g\beta(T - T_{in}) \tag{3}$$

Energy:

$$\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

where $\alpha = k/(\rho c_p)$.

The boundary conditions of the problem were established as follows:

Inlet port:

$$\begin{cases} u = U_{in},\ v = 0 \\ T = T_{in} \end{cases} \tag{5}$$

Outlet port:

$$\begin{cases} \frac{\partial u}{\partial x} = 0, \ \frac{\partial v}{\partial x} = 0 \\ \quad \frac{\partial T}{\partial x} = 0, \end{cases} \tag{6}$$

Walls:

$$\begin{cases} u = v = 0 \\ T = T_w \end{cases} \tag{7}$$

The analysis of natural, forced or mixed convection was performed using the Richardson number Ri, which lists the Grashof number Gr and Reynolds number Re, as defined in nomenclature. A characteristic length L_c, necessary for this analysis, was adopted as the value of the height and length of the cavity ($L_c = H = L$).

4. NUMERICAL PROCEDURE

The numerical simulation of the cavity was performed using the computer package open souce OpenFOAM® considering two-dimensional flow. The *buoyantBoussinesqSimpleFoam* model available in OpenFOAM® was used to simulate the cavity to consider the field forces in the problem. Transport equations were solved iteratively by the finite volume method using the SIMPLE algorithm by Patankar (1980). The *upwind* scheme was used for the discretization of the convective terms and the *Gauss Linear* scheme was used for the diffusive terms. The implicit scheme solution (Ferziger and Peric, 1999) was used for the time-differential terms during the transient period. The subrelaxation parameters were used to the convergence of the velocity, pressure and temperature values. These values were set at 0.3, 0.7 and 0.8, respectively. The tolerance for convergence iterative procedure was adjusted as 10^{-4}, 10^{-5} and 10^{-3} for the velocity, pressure and temperature, respectively. The $k - \varepsilon$ model was used in the case of significant turbulence in the flow.

Mesh tests were performed to investigate the mesh independence on the results. Six different mesh sizes were tested and the variation of the temperature profile and velocity along the cavity centerlines (position $x = L/2$ e $y = H/2$) were observed. The tests were performed with the following elements sizes, cells number and points number: 0.01m, 100 cells and 242 points; 0,005m, 400 cells and 882 points; 0,0033m, 900 cells and 1922 points; 0.0025m, 1600 cells and 3362 points; 0.002m, 2500 cells and 5202 points; 0,0017m, 3600 cells and 7442 points. In all cases tested was maintained the cells amount equality in the *x* and *y* directions of the cavity, with inlet velocity set at $0{,}1\ m/s$. Figures 2 and 3 show the results obtained for the velocity components *u* and *v* along the central lines in the cavity. From the results obtained, the mesh with 0.0033m elements size was chosen because it has good relationship between computational simulation time and reliable results. The velocity values shown in Figs. 2 and 3 to the mesh adopted are in good agreement when compared with the values obtained with most refined mesh. The greatest differences were around 15% for some specific values. The greatest variation between temperature values in the tested positions was less than 1% between the meshes most and least refined.

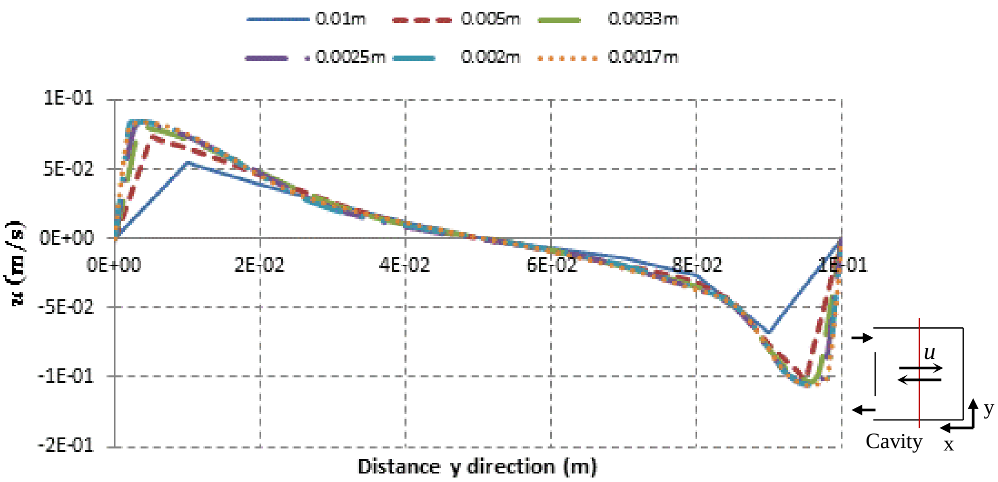

Figure 2. Variation of velocity u along the y direction (for $x = L/2$) for different mesh sizes.

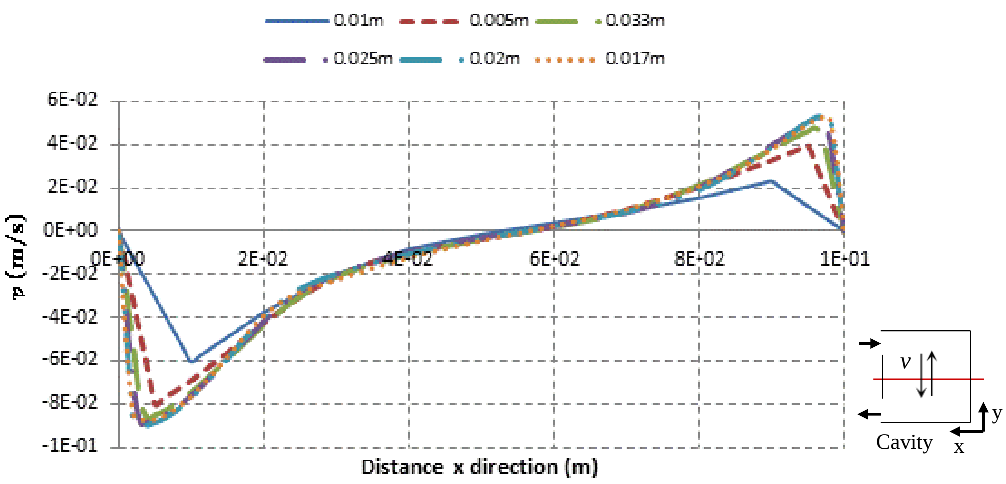

Figure 3. Variation of velocity *v* along the *x* direction (for $y = H/2$) for different mesh sizes.

5. RESULTS AND DISCUSSION

Cavity simulations were performed for nine different cases obtained from the variation of the fluid inlet velocity, with consequent variation of the Richardson number. In all cases the inlet temperature values and also the walls temperature were kept constant, with $T_{in} = 345K$ and $T_w = 300K$. Table 1 shows the velocity values and Richardson numbers for simulated cases. Table 2 shows the thermophysical properties of the water to the reference temperature used in this study.

Table 1. Inlet velocity values and Richardson numbers for tested cases.

U_{in} (m/s)	Ri
$\to \infty$	≈ 0
0.150	1
0.063	5
0.045	10
0.030	20
0.020	50
0.014	100
0.012	150
0.010	200

Table 2. Thermophysical properties of water.

Property	Water[1]
$k\ (W/mK)$	0.64
$\rho\ (kg/m^3)$	989.12
$\alpha\ (m^2/s)$	$1.55x10^{-7}$
$\nu\ (m^2/s)$	$5.83x10^{-7}$
$c_p\ (J/kgK)$	4180
$\beta(K^{-1})$	$4.37x10^{-4}$
Pr	3.77

[1]Reference temperature 320K.

The velocity and temperature fields, with the steady state flow in the cavity, are shown in Figs. 4 and 5 for different values of Richardson number tested. Its is observed in Figs. 4 and 5 that the velocity and temperature profiles do not change significantly to $Ri \leq 1$. Figure 4 shows the flow streamlines in the cavity with the variation of velocity values. It is observed that with the increasing Richardson number, there is an increase of internal regions in the cavity with a significant decrease of the velocity component. The lowest velocity values are observed in the central region of the cavity extending up to the region between the inlet and outlet openings (left side of the cavity), for $Ri \geq 10$. For $Ri = 50$ note is the formation of a recirculation region in the lower right corner of the cavity, which increases with increasing the Richardson number. With increasing Ri also increases up to rotating vortexes in the central cavity region, showing an increase of the effect of field forces and an increase in flow resistance.

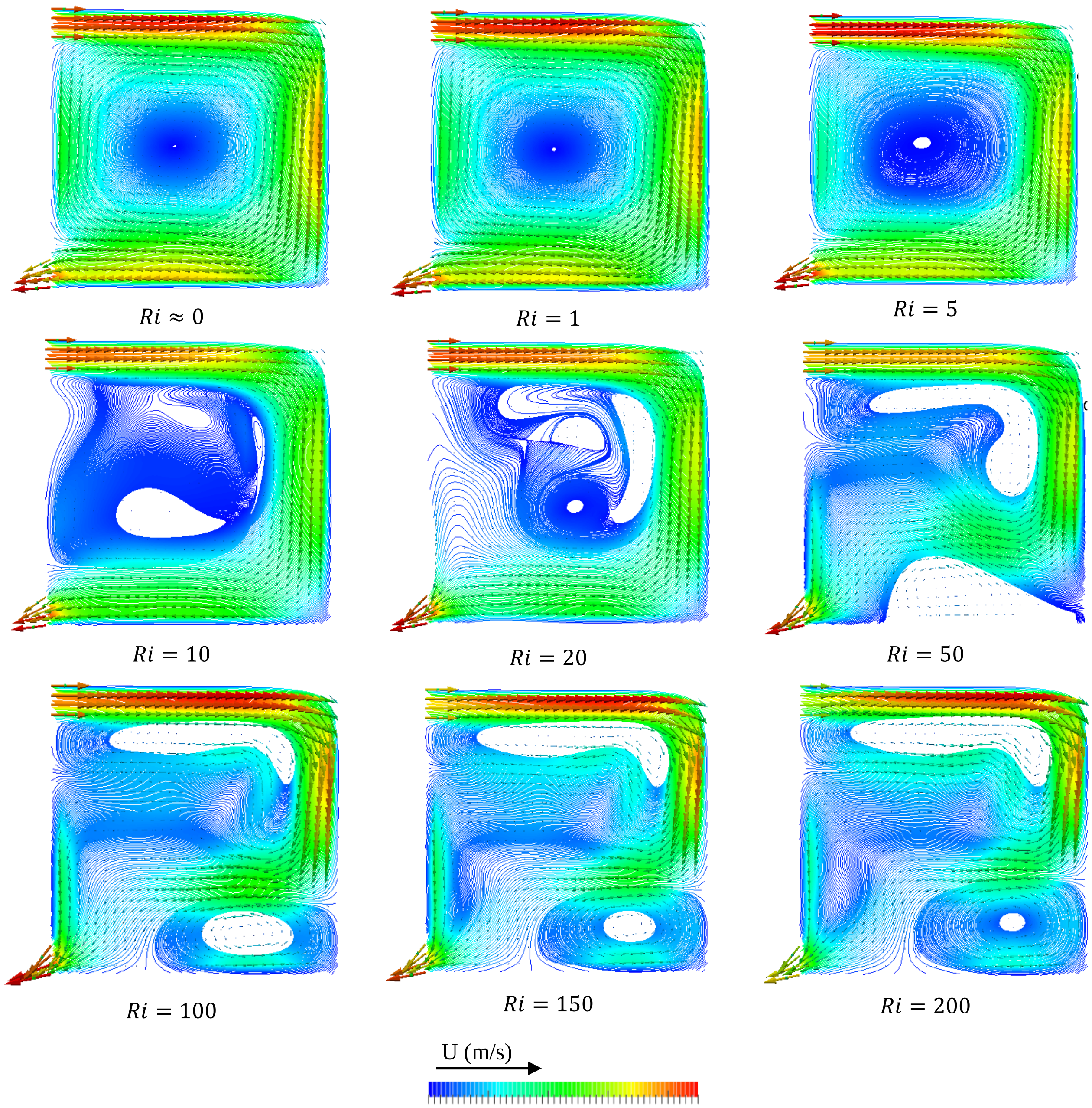

Figure 4. Streamlines with velocity profiles for different Richardson numbers.

Figure 5 shows the flow streamlines in the cavity with the variation of temperature values. It is noted the decreasing temperature values in central and bottom regions of the cavity with increasing Richardson number, which is consistent with the observed in a reservoir of water thermal stratification in a solar heating system. It is even observed that the temperature values inside the cavity are close to the water inlet temperature value (350K) for $Ri \leq 20$. This is due to the fact that there is a significant amount of energy entering the cavity compared to the area of heat exchange. Thus, the thermal capacity of the inlet flow is somewhat affected by the effect of the cavity walls.

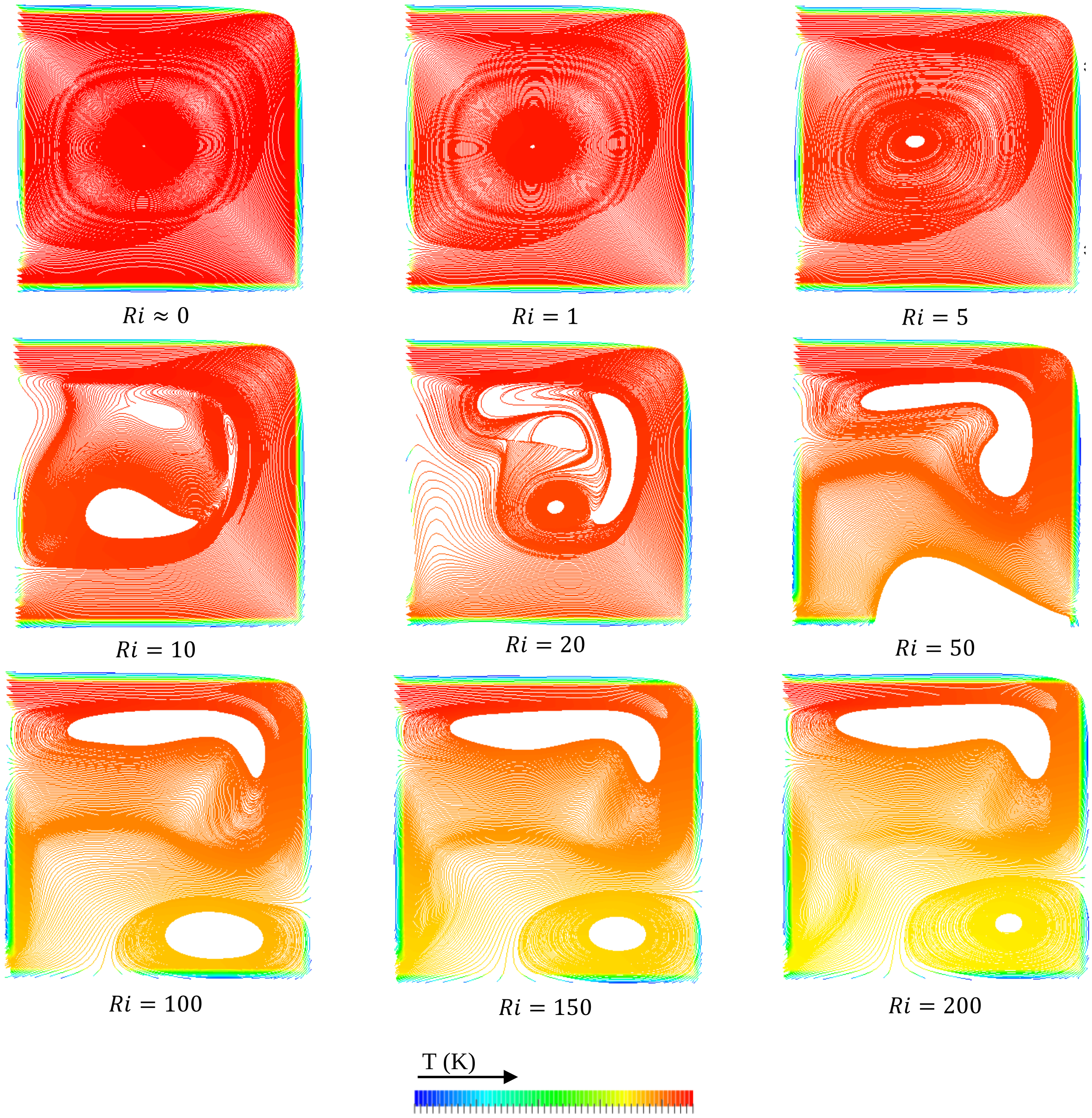

Figure 5. Streamlines with temperature profiles for different Richardson numbers.

An interesting result to be observed are lines of constant temperature values (isotherms) in the cavity, once they (isotherms) provide information on the water thermal stratification. Figure 6 shows the isotherms in the cavity depending on the Richardson number. The main observation is that, as the effects of field forces are enhanced, i.e., increased Ri, the isotherms, especially in the central and bottom regions of the cavity tends to have a horizontal profile. This demonstrates higher thermal stratification in the cavity. This situation is more interesting for thermal reservoir of solar heater. Note that for $Ri \leq 20$ the isotherms show significant slope to the horizontal plane demonstrating lower stratification.

For thermal reservoir of solar heater in many situations is the mixed convection with $Ri \approx 1$. By analyzing the results in the cavity, it is possible to identify how convection affects the thermal stratification within a reservoir. The simultaneity of natural and forced convection may not be beneficial for stratification, although it is observed in many thermal reservoirs. The results and modeling can contribute to simulate a three-dimensional thermal reservoir in the investigation of the water stratification.

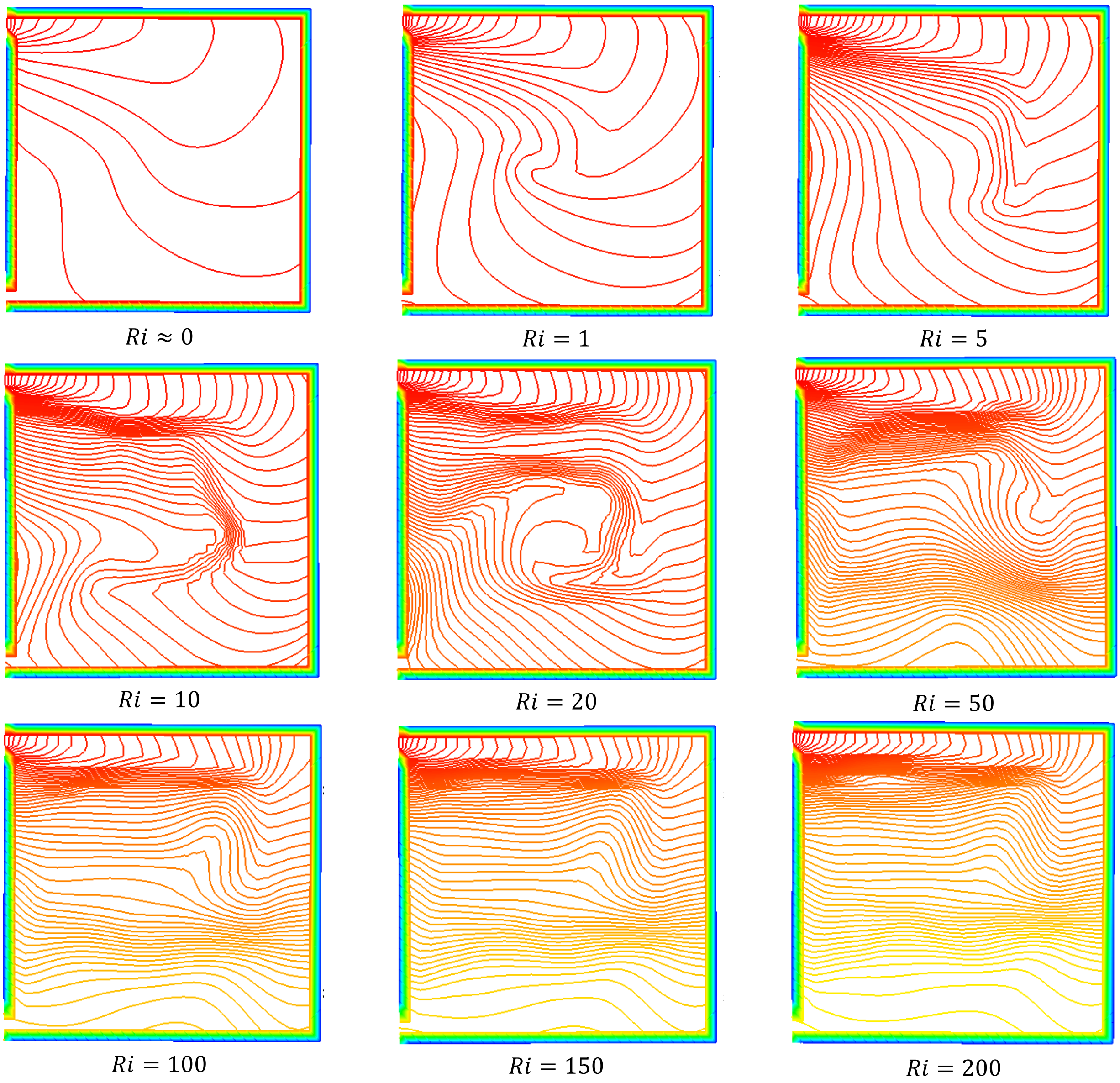

Figure 6. Isotherms for different Richardson numbers.

6. CONCLUSIONS

In this work, a cavity with two openings, one for inlet and one for water outlet, was numerically analyzed. Nine different cases, each one with specified *Ri*, were simulated. The results for the velocity and temperature profiles were obtained in the cavity and they allowed to characterize the thermal stratification. For the simulation of a solar thermal reservoir, a three-dimensional cylindrical geometry, the results of the cavity may contribute to the selection of models available in OpenFOAM® package in the investigation of water thermal stratification in the reservoir.

The study showed that the increase in the flow field forces implies the occurrence of vortex and recirculating regions within the cavity, as well as reducing the temperature values in the central and bottom regions. However, the increase in field forces provides better water thermal stratification in the cavity, which could be observed through the isotherms obtained in the simulation.

7. REFERENCES

Basak, T., Roy, S., Sharma, P. K., Pop, I., 2009. “Analysis of mixed convection flows within a square cavity with uniform and non-uniform heating of bottom wall”. International Journal of Thermal Science, Vol. 48, pp. 891–912.

Belmiloud, M.A., Sad chemloul, N., 2015. “Effect of baffle number on mixed convection within a ventilated cavity”. Journal of Mechanical Science and Technology, Vol. 29, No. 11, pp. 4719-4727.

Ferziger, J. H., Peric, M., 1999. Computational Method for Fluid Dynamics, Springer-Verlag, New York.

Ghasemi, B., Aminossadati, S.M., 2008. “Numerical simulation of mixed convection in a rectangular enclosure with different numbers and arrangements of discrete heat sources”. Arab. J. Sci. Eng., Vol. 33, No. 1B, pp. 189–207.

Guo, G., Sharif, M.A.R., 2004. “Mixed convection in rectangular cavities at various aspect ratios with moving isothermal sidewalls and constant flux heat source on the bottom wall”. International Journal of Thermal Science, Vol. 43, pp. 465–475.

Khanafer, K.M., Al-Amiri, A.M., Pop, I., 2007. “Numerical simulation of unsteady mixed convection in a driven cavity using an externally excited sliding lid”. Eur. J.Mech. B/Fluids, Vol. 26, pp. 669–687.

Mahmoudi, A.H., Shahi, M., Talebi, F., 2010. “Effect of inlet and outlet location on the mixed convective cooling inside the ventilated cavity subjected to an external nanofluid”. International Communications in Heat and Mass Transfer, Vol. 37, pp. 1158–1173.

Patankar, S.V., 1980. Numerical heat transfer and fluid flow, Hemisphere, New York.

Radhakrishnan, T.V., Verma, A.K., Balaji, C., Venkateshan, S.P., 2007. “An experimental and numerical investigation of mixed convection from a heat generating element in a ventilated cavity”. Experimental Thermal and Fluid Science, Vol. 32, pp. 502-520.

Rahman, M.D.M., Alim, M.A., Saha, S., Chowdhury, M.K., 2008. “A numerical study of mixed convection in a square cavity with a heat conducting square cylinder at diferente locations”. J. Mech. Eng. ME39, Vol. 2, pp. 78–85.

Rahman, M.M., Parvin, S., Rahim, N.A., Islam, M.R., Saidur, R., Hasanuzzaman, M., 2012. “Effects of Reynolds and Prandtl number on mixed convection in a ventilated cavity with a heat-generating solid circular block”. Applied Mathematical Modelling, Vol. 36, pp. 2056-2066.

Saha, S., Saha, G., Ali, M., Islam, Md.Q., 2006. “Combined free and forced convection inside a two-dimensional multiple ventilated rectangular enclosure”. ARPN J. Eng. Appl. Sci., Vol. 1 No. 3, pp.23–35.

Saha, S., Saha, G., Ali, M., Islam, Md.Q., 2007. “Combined free and forced convection inside a two-dimensional multiple ventilated rectangular enclosure”. ARPN J. Eng. Appl. Sci., Vol. 2 No. 2, pp.25–36.

Sourtiji, E., Hosseinizadeh, S.F., Gorji-Bandpy, M., Ganji, D.D., 2011. “Effect of water-based Al2O3 nanofluids on heat transfer and pressure drop in periodic mixed convection inside a square ventilated cavity”. International Communications in Heat and Mass Transfer, Vol. 38, pp. 1125–1134.

Sourtiji, E., Gorji-Bandpy, M., Ganji, D.D., Hosseinizadeh, S.F., 2014. “Numerical analysis of mixed convection heat transfer of Al2O3-water nanofluid in a ventilated cavity considering different positions of the outlet port”. Powder Technology, Vol. 262, pp. 71-81.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.