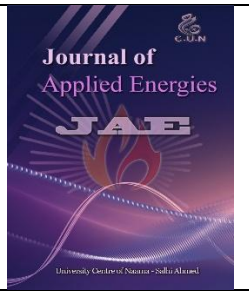
Journal of Applied Energies 01 (2022) 40-48

Journal of Applied Energies



Study on the passage of a cold fluid for two successive chambers with heated lower walls

Houssem Laidoudi[1,*], Houari Ameur[2]

[1]Laboratory of Sciences and Marine Engineering (LSIM), Faculty of Mechanical Engineering, USTO-MB, BP 1505, El-Menaouer, Oran 31000, Algeria

[2]Department of Technology, University Centre of Naama, P.O. Box 66, Naama 45000, Algeria.

*Correspondence. Email address: hichemsoft19@gmail.com

Abstract: Through this work, we want to evaluate the behavior of the fluid inside two successive chambers equipped with an opening for the fluid to enter the flow and the other for its exit. The fluid has a low temperature, while the lower wall of the two chambers has an elevated temperature. The results of this research show the effect of velocity and thermal buoyancy intensity on the flow path and thermal distribution inside the two chambers. Also, the values of *Nu* are computed in terms of the studied parameters.

Keywords: Heat transfer; Successive chambers; Fluid flows; Thermal buoyancy.

1. INTRODUCTION

Recently, a group of researches was conducted to study the behavior of the fluid in an open and closed chamber. This interest is due to the presence of these rooms in various applications

in the field of industry, including the following: the heat exchangers, refrigeration systems, nuclear reactors, cooling systems of electronic applications, and so on.

Manca et al. [1] studied the passage of incompressible flow through an open cavity. The aim of the study was to test the position of the heated wall of the cavity on the flow and thermal comportments. The intensity of thermal buoyancy was defined by the Ri number (0.1). On the other hand, the flow velocity corresponded to the values 100 and 1000 of Reynolds number. Mahmood et al. [2] studied the same physical model with an obstacle over the cavity and used the non-Newtonian fluid as a working fluid. This research was performed for the process of metallurgical applications. Brahmi and Hadjadj [3] studied a flow within an open channel containing a semi-circular cavity to test the weak shock waves. Hussain et al. [4] simulated a flow through a single cavity using nanoparticles of Al_2O_3. The results showed that the presence of nanoparticles in basic fluid (water) improves heat transfer. Yamamoto et al. [5] achieved three-dimensional simulations of viscoelastic flow inside a channel of rectangular shape with a cavity. The governing equations were solved using the Phan-Thien–Tanner (PTT) model. The behavior of the flow in the cavity was the purpose of this research. Zamzari et al. [6] studied the entropy generation and its effect on heat transfer of the air inside an opening channel with a single cavity. The cavity was a square cross-section. The numerical simulations were in 2D. The flow at the channel inlet was in pulsation mode of different frequencies and amplitude. In addition to all this, other works in this context studied the fluid inside rooms, including those in Refs. [7-12].

Through our survey of all these and other works, it appears that most of the outcomes depended on one room. This reason made us suggest studying the fluid behavior within two successive chambers to enhance the results for this field.

2. DESCRIPTION OF THE STUDIED GEOMETRY

The physical model studied in this paper is schematically represented in Figure 1. The domain consists of an open channel with two equal-sized successive chambers, and the rooms are square cross-sections. The fluid enters the channel with an initial velocity (u_{in}) and low temperature (T_{in}) and then crosses the two rooms where the lower wall has a high temperature (T_w). The diameter of the channel (d) is equal to the gap spacing between the chambers.

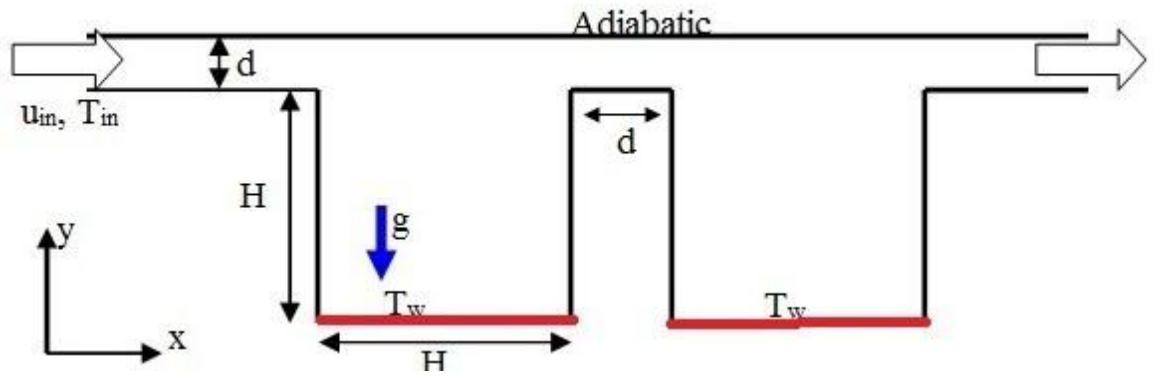


Figure 1: Simple representation of the studied domain

3. MATHEMATICAL FORMULATION AND SIMULATION STEPS

The appropriate equations for the present work are continuity, momentum, and energy. These equations are written in dimensionless form as:

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \tag{1}$$

$$U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right) \tag{2}$$

$$U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial Y}+\frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right)+Ri\theta \tag{3}$$

$$U\frac{\partial \theta}{\partial X}+V\frac{\partial \theta}{\partial Y}=\frac{1}{PrRe}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right) \tag{4}$$

Reynolds number (*Re*), Grashof number (*Gr*), Richardson number (*Ri*), Prandtl number (*Pr*) are expressed as follows:

$$\mathrm{Re}=\frac{u_{in}d}{\nu},Gr=\frac{g\beta_T\Delta T d^3}{\nu^2},Ri\frac{Gr}{\mathrm{Re}^2},Rr=\frac{\nu}{\alpha} \tag{5}$$

where the dimensionless variables in the main equations were determined as:

$$(X,Y)=\frac{(x,y)}{d},(U,V)=\frac{(u,v)}{u_{in}},P=\frac{p}{\rho u_{in}^2},\theta=\frac{(T-T_{in})}{T_w-T_{in}} \tag{6}$$

where u_{in}, g, ν , βT are the inlet velocity* flow, gravitational acceleration, kinematic velocity, and volume expansion coefficient, respectively.

The boundary conditions for the present simulation are:

- At the inlet of the opening channel:

$$U = 1, V = 0, \theta = 0 \tag{6}$$

- At the outlet of the channel:

$$\frac{\partial U}{\partial X} = 0, \frac{\partial V}{\partial X} = 0, \frac{\partial \theta}{\partial X} = 0 \tag{7}$$

- On bottom walls:

$$U = 0, V = 0, \theta = 1. \tag{8}$$

- On the rest cavity and channel walls:

$$U = 0, V = 0, \frac{\partial \theta}{\partial n} = 0 \tag{9}$$

The local and average Nusselt number expressions are:

$$Nu_l = \frac{hd}{k} = -\frac{\partial \theta}{\partial n_s} \tag{10}$$

$$Nu = \frac{1}{S}\int_s Nu_l ds \tag{11}$$

where h and n_s are the local surface heat transfer coefficient and the normal direction to the cylinder surface, respectively.

The studied geometry, as well as the grid, have been achieved by Gambit. The form obtained at this stage is shown in Figure 2. This grid is received after the grid independence test, and the entire system contains 122,500 elements. The grid elements have a structural form. However, the numerical simulations were done by ANSYS-CFX. The fluid velocity at the inlet was inferred from the equivalent value of Reynolds number. On the other hand, the intensity of the thermal buoyancy was inferred from the value of *Ri*. Finally, thermo-physical characteristics of the fluid are determined through the expression of Prandtl number (*Pr* = 6.01, which is the water).

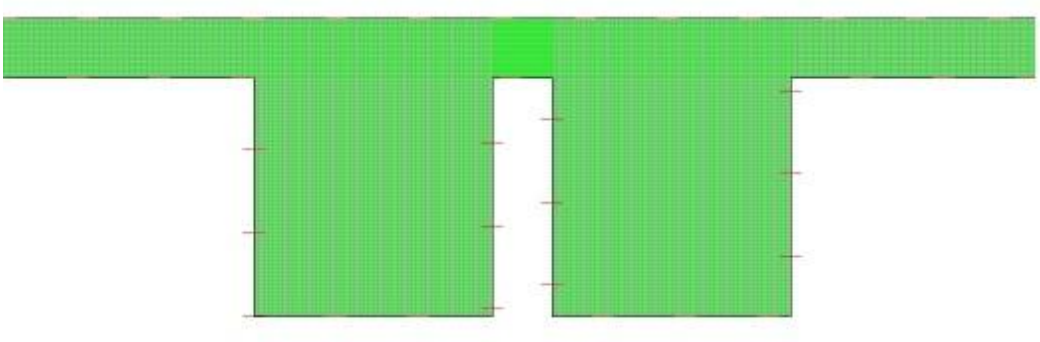


Figure 2: Appropriate grid for the present study.

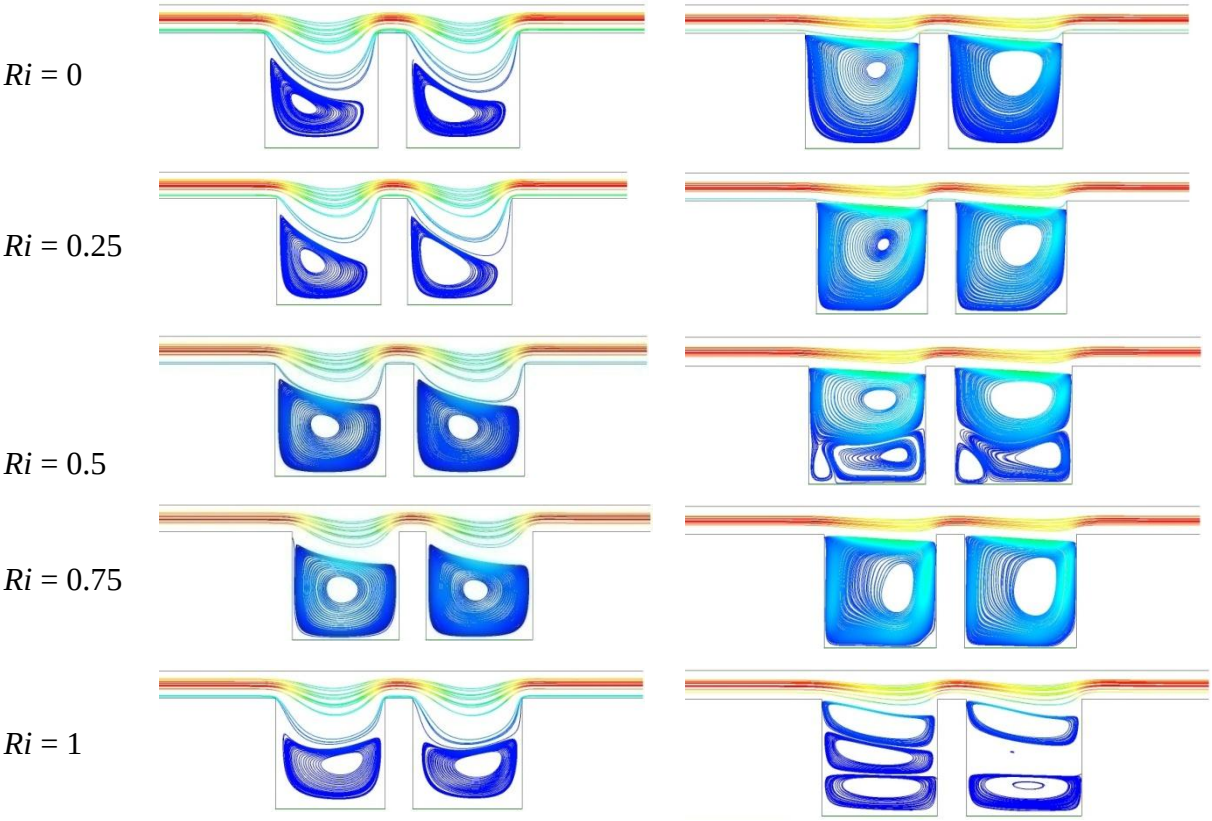


Figure 3: Streamlines in the studied domain for different values of *Ri* and *Re*.

A validation test was achieved to prove the accuracy of our numerical method. The presentation of the validation test is presented in our previous works [11, 12].

4. RESULT AND DISCUSSION

Figure 3 shows the flow behavior in the studied domain in terms of *Ri* and *Re*. When *Ri* = 0, it means no effect of thermal buoyancy. In this case, a recirculation zone appears in each chamber. The size of both zones increases with increasing the velocity of flow at the inlet.

Also, the center of the vortices moves up with increasing *Re*. In addition, the flow stability in the chamber decreases with the increasing intensity of the buoyancy. The new form and other steady vortices appear in both chambers. The effect of buoyancy becomes very significant for the high value of *Re*. The presence of these vortices may reduce the ratio of heat transfer.

Figure 4 shows the isotherms (dimensionless temperature) in the domain. The dissipation of temperature contours depends on both studied numbers *Re* and *Ri*. Also, the temperature gradient near the bottom walls increases with increasing *Re* and *Ri*. That means the heat transfer mainly depends on *Ri* and *Re*. Also, along the bottom walls. The distribution of gradient temperature is not homogeny. Furthermore, the heat transfer of the first chamber is greater than the second chamber due to the value of flow velocity, which is vital in the first chamber. The flowing movement increases the evacuation of heat energy and, accordingly, improves heat transfer.

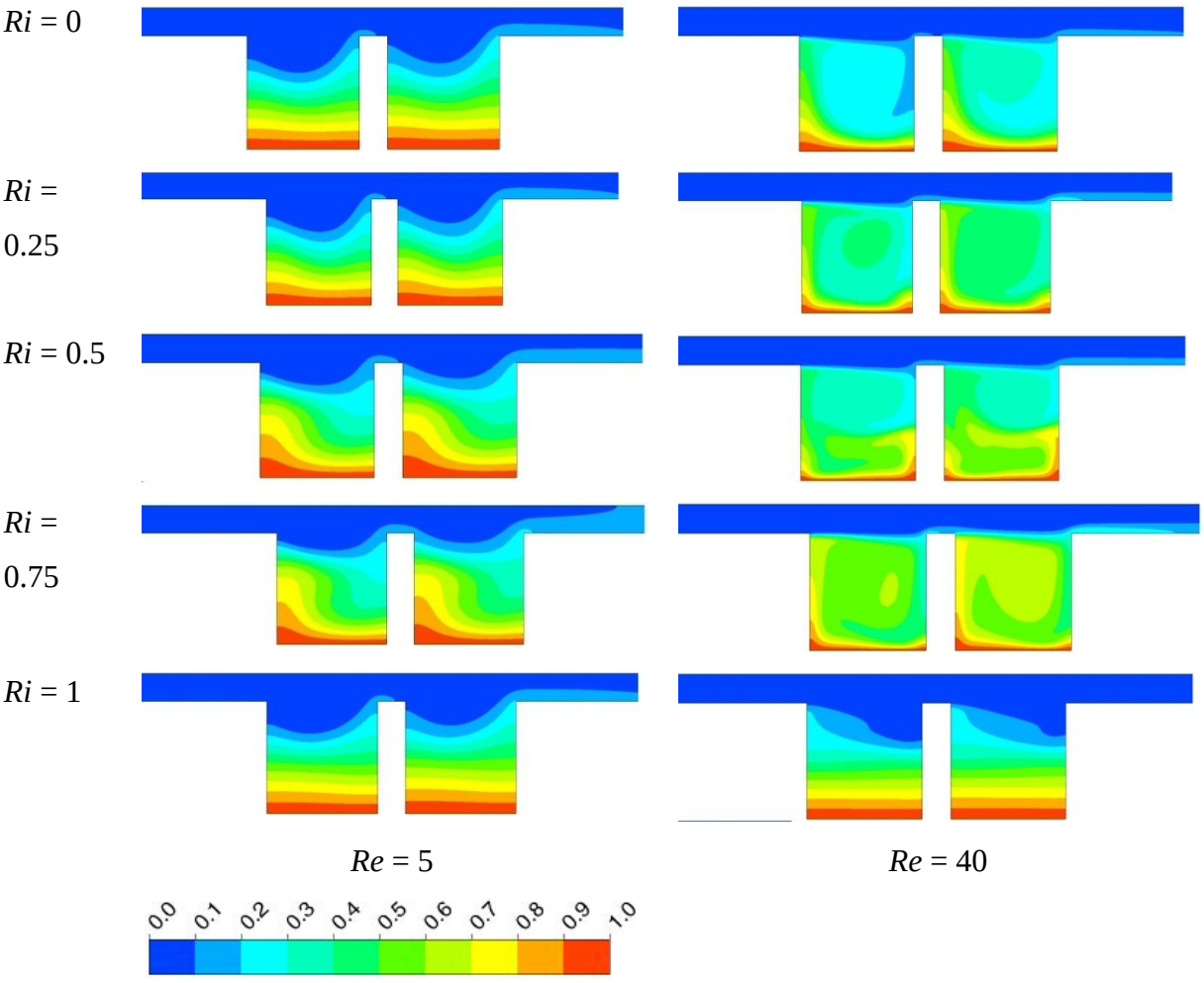


Figure 4: Isotherms in the studied domain for different values of *Ri* and *Re*

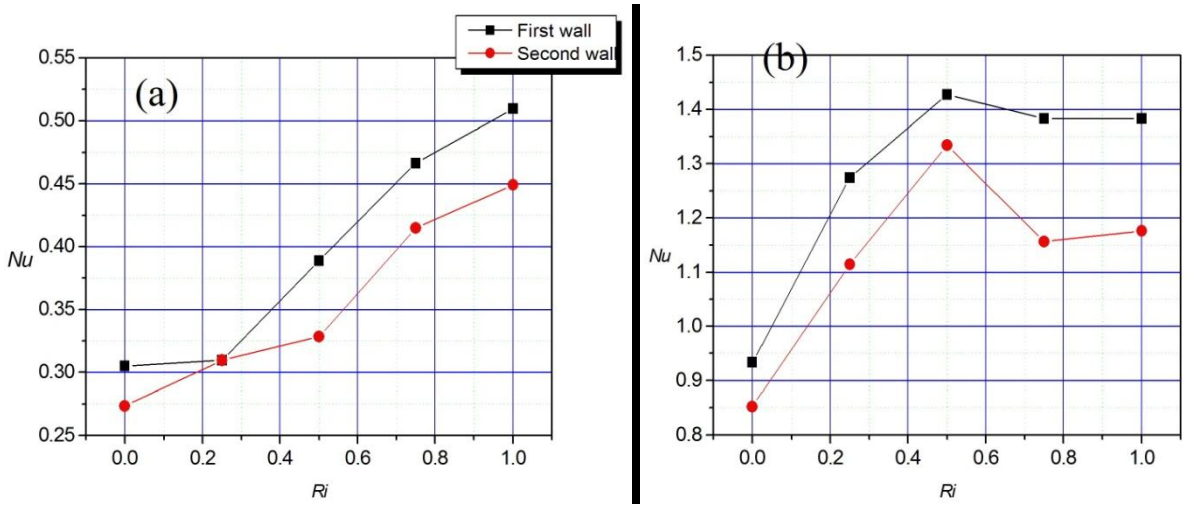


Figure 5: Variation of *Nu* versus *Ri* for both bottom walls, (a) *Re* = 5, (b) *Re* = 40.

Figure 5 shows the evolution of the average Nusselt number versus *Ri* and *Re* for the first wall and the second wall. As expected, the increase in the value of *Re* increases *Nu*. For *Re* = 5, increasing *Ri* number increases the average *Nu* number. On the other hand, for *Re* = 40, the increased *Ri* increases *Nu* up to the value 0.5 and decreases again. It is mainly due to vortices' appearance, which reduces the impact of flow velocity on the evacuation of heat transfer.

5. CONCLUSION

The present work is a numerical simulation of incompressible fluid in an opening channel equipped with two equal-size successive chambers. The study's objective is to understand the behavior of hydrodynamic flow in the domain and determine its effect on thermal performance. The principal points drawn from this study are:

- The increase in *Re* number increases the heat transfer for both walls.
- The effect of thermal buoyancy is significant for *Re* = 40.
- The increase in *Ri* number for *Re* = 40 leads to a new form of steady vortices in both chambers.
- *Ri* = 0.5 is the optimal value for heat transfer when *Re* = 40 and *Ri* = 1 for *Re* = 5.

REFERENCES

[1] Manca, O., Nardini, S., Khanafer, K., Vafai, K. (2003). Effect of heated wall position on mixed convection in a channel with an open cavity. *Numerical Heat Transfer: Part A: Applications*, *43*(3), 259-282.

[2] Mahmood, R., Bilal, S., Majeed, A. H., Khan, I., Nisar, K. S. (2020). Assessment of pseudo-plastic and dilatant materials flow in channel driven cavity: application of metallurgical processes. *Journal of Materials Research and Technology*, *9*(3), 3829-3837.

[3] Brahmi, N., Hadjadj, A. (2020). Achieving an optimal shock-wave mitigation inside open channels with cavities for weak shock waves: A computational study. *Theoretical and Applied Mechanics Letters*, *10*(5), 354-365.

[4] Hussain, S., Mehmood, K., Sagheer, M., Farooq, A. (2017). Entropy generation analysis of mixed convective flow in an inclined channel with cavity with Al2O3-water nanofluid in porous medium. *International Communications in Heat and Mass Transfer*, *89*, 198-210.

[5] Yamamoto, T., Ishiyama, M., Nakajima, M., Nakamura, K., Mori, N. (2003). Three-dimensional viscoelastic flows through a rectangular channel with a cavity. *Journal of non-Newtonian Fluid Mechanics*, *114*(1), 13-31.

[6] Zamzari, F., Mehrez, Z., El Cafsi, A., Belghith, A., & Le Quéré, P. (2017). Numerical investigation of entropy generation and heat transfer of pulsating flow in a horizontal channel with an open cavity. *Journal of Hydrodynamics*, *Ser. B*, *29*(4), 632-646.

[7] García, F., Treviño, C., Lizardi, J., Martínez-Suástegui, L. (2019). Numerical study of buoyancy and inclination effects on transient mixed convection in a channel with two facing cavities with discrete heating. *International Journal of Mechanical Sciences*, *155*, 295-314.

[8] Wang, L., Luo, X., Zhang, J., He, B., Peng, Z. (2021). Flow boiling characteristics of minichannel heat sink with artificial conical cavities array under electric field. *International Journal of Heat and Mass Transfer*, *173*, 121286.

[9] Huang, B., Li, H., Xia, S., Xu, T. (2020). Experimental investigation of the flow and heat transfer performance in micro-channel heat exchangers with cavities. *International Journal of Heat and Mass Transfer*, *159*, 120075.

[10] Sáchica, D., Treviño, C., Martínez-Suástegui, L. (2020). Numerical study of magnetohydrodynamic mixed convection and entropy generation of Al2O3-water nanofluid in a channel with two facing cavities with discrete heating. *International Journal of Heat and Fluid Flow*, *86*, 108713.

[11] Laidoudi, H., Makinde, O. D. (2021). Study on the effect of flow velocity and thermal buoyancy on the dynamic behaviour of flow inside a two-hole chamber. *International Journal of Ambient Energy*, 43(1), 5393-5404.

[12] Laidoudi, H., Makinde, O. D. (2021). Computational study of thermal buoyancy from two confined cylinders within a square enclosure wifth single inlet and outlet ports. *Heat Transfer*, *50*, 1335-1350.