ENCIT 2018

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-XXXX
CFD SIMULATION OF A NATURAL CONVECTION FLOW IN A ENCLOSURE CAVITY WITHIN A HEATED BODY

Valeria S. Franca
Federal University of Rio de Janeiro
Nuclear Engineering Department, Polytechnic School/UFRJ
valeria.f@poli.ufrj.br

Felipe P. Ribeiro
Jian Su
Nuclear Engineering Department, COPPE-POLI
Universidade Federal do Rio de Janeiro
Av. Horácio Macedo, 2030, G206, Centro de Tecnologia, Cidade Universitária,
21941-914 Rio de Janeiro, Brazil
felipeportor@poli.ufrj.br
sujian@nuclear.ufrj.br

Abstract. *Natural convection plays a relevant role in the industry field. In nuclear engineering, it can be encountered in the passive cooling systems in the dry casks used to storage spent nuclear fuel. The aim of the present work is to analyze the Rayleigh-Bernard convection in a square cavity containing an isothermal hot body, in order to investigate the influence of the hot body for different Rayleigh numbers (Ra), varying in the range of* 10^3 *to* 10^{11}*. A two dimension solution for the physical model proposed is obtained from numerical simulations carried out in the Computational Fluid Dynamics (CFD) code ANSYS- Fluent. For small and intermediate* Ra *the solutions present a laminar steady state with difference in the symmetry behavior of the flow and temperature. Temperature and velocity contours are symmetric at Ra=*10^3 *and non-symmetric from Ra=*10^4*. At high* Ra*, flow and temperature become unsteady and rehearse to reach a symmetric standard again.*
Keywords: *ANSYS-Fluent, Natural Convection, Rayleigh.*

1. INTRODUCTION

The study of natural convection flow in enclosures cavities plays an important role in the industry due to its wide thermal engineering applications, such as heat exchanger, solar panels and cooling devices for electronic instruments.

In the nuclear field, natural convection flows can be encountered to cool dry casks used for spent nuclear fuel storage. Cooling systems for these casks operate on the basic principles of buoyancy driven natural convection, where cooler air enters the air passage near the bottom of the system, absorbs decay heat, and exits the system near the top with high temperature and lower density.

Buoyancy-driven flow of a fluid heated from below and cooled from above is the so called Rayleigh-Bernard convection. Deardorf and Willis (1965) studied a two-dimensional flow between two horizontal plates for high Rayleigh numbers (Ra) observing the development of buoyancy in a short period of turbulence. Their results show that the cells almost do not presented variation as their wavelengths increased.

The transition from the laminar to turbulent regime was also investigated by Deardorf and Willis (1965), for Ra ranging from 10^3 to 10^6. It was discovered that a transition region of almost constant convection to the turbulent regime for Ra values between 10^3 and 10^4 and for high Ra a dominant wavelength was observed.

Ha *et al.* (2002) analyzed the natural convection in a 2D square cavity with a heated body located between a cold up wall and a hot down one, keeping an isothermal condition. Different Ra ranging between 10^3 and 10^6 were used to verify the influence of the body in the natural convection process, reaching the conclusion that for low Ra, such as 10^3, the temperature profile presents a standard symmetry behavior. In the case of intermediate Ra (in the order of 10^4), the symmetry is lost but still presents a steady-state. For higher Ra the flow and temperature become time dependent.

The present work analyzes the Rayleigh-Bernard convection in a square cavity containing an isotherm heated body located at the center of the domain. The objective is to investigate the influence of the heated body according to the Rayleigh number (Ra) variation in a laminar regime. Boussinesq model was used to perform the fluid behavior and the solution were obtained by using the commercial CFD code ANSYS-Fluent.

2. MATHEMATICAL MODELS AND METHODS

A two-dimensional CFD model of a square cavity has been developed allowing the study of Rayleigh-Benard convection as a result of Rayleigh number variation. The simulation mathematical model includes

Continuity Equation:

$$\frac{\partial u_i{}^*}{\partial x_j{}^*} = 0; \tag{1}$$

Navier Stokes Equation:

$$\frac{\partial u_i{}^*}{\partial t_j{}^*} + u_i{}^* \frac{\partial u_i{}^*}{\partial x_j{}^*} = -\frac{\partial P^*}{\partial x_j{}^*} + Pr\frac{\partial^2 u_i{}^*}{\partial x_j{}^* \partial x_j{}^*} + RaPr\theta\delta; \tag{2}$$

Energy Equation:

$$\frac{\partial \theta}{\partial t^*} + u_j \frac{\partial \theta}{\partial x_j{}^*} = \frac{\partial^2 u_i{}^*}{\partial x_j{}^* \partial x_j{}^*}; \tag{3}$$

2.1 Dimensionless Numbers

For this problem, dimensionless numbers are defined as:

$$t* = \frac{t\alpha}{L^2},\ \ x_i^* = \frac{x_i}{L},\ \ u_i^* = \frac{u_i L}{\alpha},\ \ P^* = \frac{PL^2}{\rho\alpha^2},\ \ \theta = \frac{T - T_c}{T_h - T_c}; \tag{4}$$

where, ρ, α, u_i, P and θ are, respectively, density, thermal diffusivity, velocity, pressure and temperature.
The above equations result in two dimensionless parameters:

$$Ra = \frac{g\beta L^3 (T_h - T_c)}{\nu\alpha}; \tag{5}$$

$$Pr = \frac{\mu C_p}{k} = \frac{\nu}{\alpha} \tag{6}$$

2.2 CFD MODEL

The physical model and its boundary conditions are outlined in Figure 1, representing a $2D$ square cavity of dimension L, within a heated isotherm body with dimension $W = L/3$. External vertical walls of the cavity are assumed to be adiabatic ($q = 0$), while horizontal walls are assumed to be isothermal, with the upper one at cold temperature, T_c, and the bottom one at hot temperature, T_h.

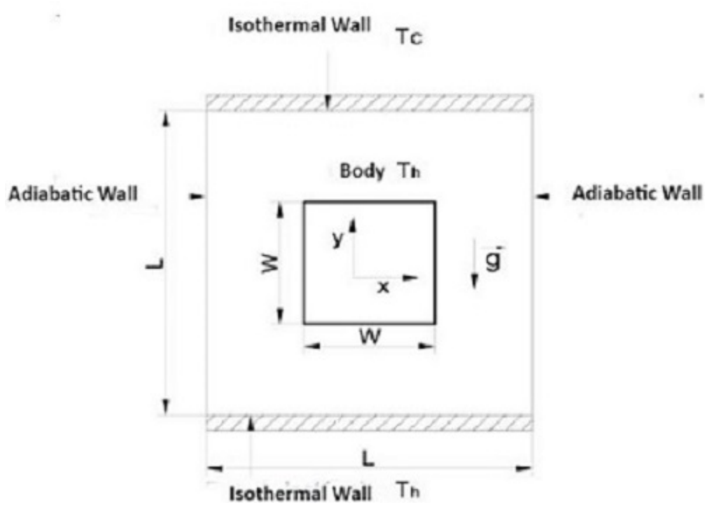


Figure 1: **Simulation domain and boundary conditions (Rodrigues and Su (2016)).**

The fluid used in the domain is the air with constant properties, except for the density that follows the Boussinesq model.

Commercial CFD code ANSYS-Fluent 18.2 was used for the analysis and the solution was obtained for the steady and unsteady state incompressible Navier-Stokes equations. Pressure based solver was last square cell based, using an implicit scheme for convergence and the SIMPLEC algorithm was used to pressure-velocity coupling.

2.2.1 Mesh sensitivity study

A mesh sensitivity study was performed by Rodrigues and Su (2016), who established that the 400x400 mesh has a better results between time and precision of calculation. Therefore, this is the mesh used for simulations presented in this paper.

Ra	Malha(200×200)	Malha(400×400)	Malha(800×800)
10^3	0,1134	0,1133	0,1133
10^4	0,4144	0,4147	0,4147

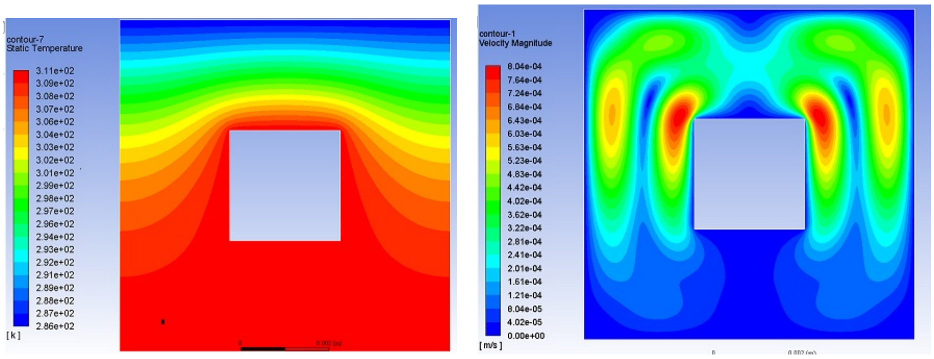
Figure 2: Nu_h **values for the mesh sensitivity study (Rodrigues and Su, 2016).**

3. RESULTS

For Rayleigh number 10^3 the natural convection is in steady state. Figures 3 (a) and (b) show the contours for temperature and velocity field, respectively.

With the temperature difference between the cold top wall of the cavity and the heated body, a strong thermal gradient appears in the upper part of the cavity, while the lower part keeps the temperature of the hot bottom wall.

As shown, the isotherms and streamlines are symmetric across the center line, demonstrating that the heat is transferred primarily by conduction.

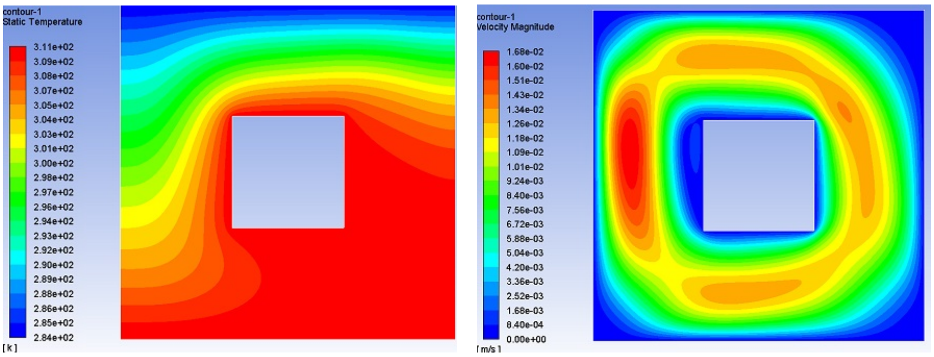


(a) Temperature Field for Ra=10^3

(b) Velocity Field for Ra=10^3

Figure 3: **Temperature and velocity contours for Ra=**10^3

When the Rayleigh Number is increased to Ra=10^4, the temperature contour lost its symmetry and starts to concentrate at the left part of the cavity. The velocity field increases due to the effect of heat transfer from the heated body. When compared to the velocity obtained from Ra=10^3 it is perceptible that the two vortex formed in the previous case, join in a single cell because its isotherms lines follow the fluid flow, rotating in a counter-clockwise, and so, changing to a non-symmetric thermal gradient, showing the presence of heat transfer by conduction. See Figures 4 (a) and (b).

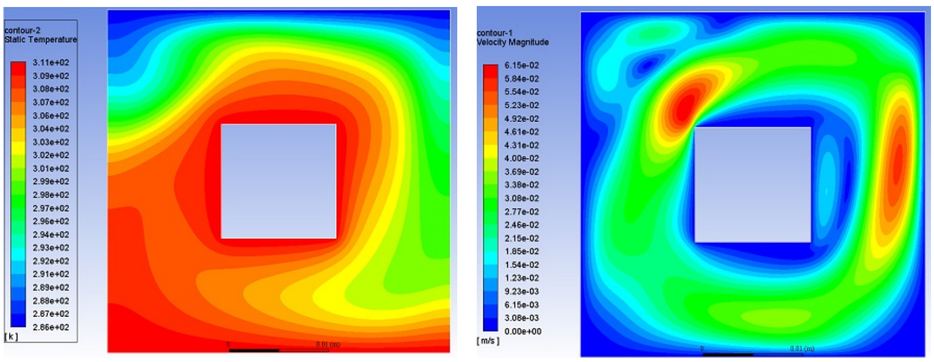


(a) Temperature Field for Ra=10^4 (b) Velocity Field for Ra=10^4

Figure 4: **Temperature and velocity contours for Ra=**10^4

For Rayleigh Number from 10^5, the flow starts to present a time dependent behavior since there is no convergence for the steady state. When it reaches to Ra=10^5 it is perceived that the thermal gradient increases in the upper and bottom wall, and also around the heated body with the increase of Ra. This thermal profile presents an almost symmetric behavior, except for the thermal gradient at the right side.

The increase of the gradients in the cavity results in the increase of heat transfer. Which, in turn, keeps increasing the velocity field due to convection. Two vortex appear, one at the upper left corner and the other at the right side of the body as shown in Figures 5(a) and (b).

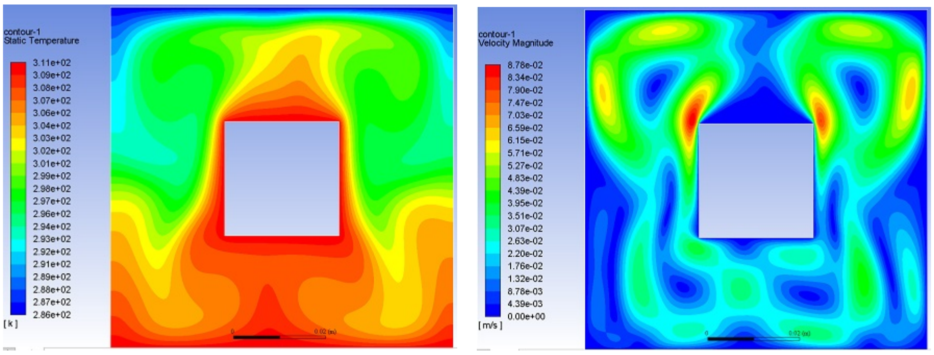


(a) Temperature Field for Ra=10^5 (b) Velocity Field for Ra=10^5

Figure 5: **Temperature and velocity contours for Ra=**10^5

For Ra=10^6 it is seen that the thermal gradient increases at the upper wall and around the heated body due to the increase of the Ra. The stream lines get back to the symmetric form, appearing four vortices in the enclosure. Two large ones at the upper part, where the fluid flow has predominance due to the presence of the heated body, and two small ones at the lower part of the cavity.

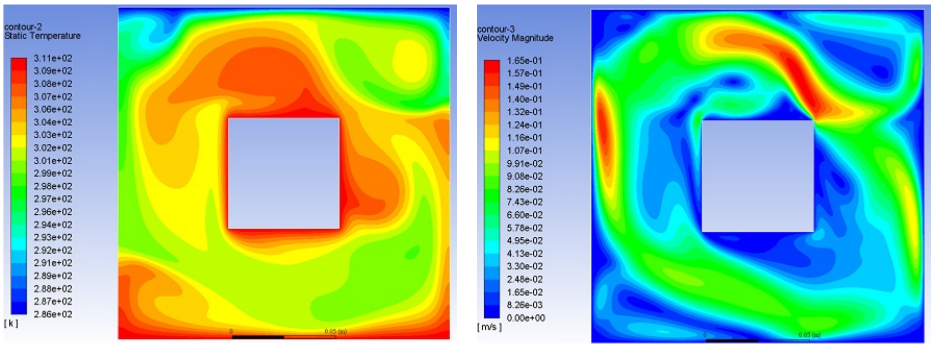
The isotherm lines rehearsing to get back to the symmetry contour due to the increase of transfer by convection, as shown in Figures 6 (a) resulting also in an increasement of the velocity of the flow that concentrates at the upper corner of the cavity the highest average values of velocity, showed in Figures 6 (b).



(a) Temperature Field for Ra=10^6 (b) Velocity Field for Ra=10^6

Figure 6: **Temperature and velocity contours for Ra=**10^6

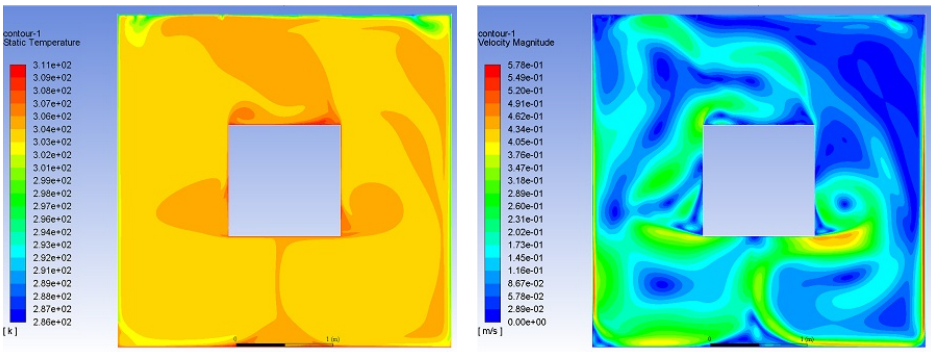
For $Ra = 10^7$, it is seen the predominance of the temperature average in the cavity due to the velocity increases, Figures 7 (a). This increasement in the velocity makes the four vortex in the previous case rehearsing to join in a single cell, similar to the case Ra=10^4, see Figures 7 (b).



(a) Temperature Field for Ra=10^7 (b) Velocity Field for Ra=10^7

Figure 7: **Temperature and velocity contours for Ra=**10^7

Results for $Ra = 10^8$, $Ra = 10^9$ and $Ra = 10^{10}$ do not present a significant difference from the previous case. The simulations for $Ra = 10^{11}$ did not converge, showing that the actual model is not sufficient. There is a thermal equilibrium in the entire cavity and the streamlines do not present any symmetry. See Figures 8(a) and (b).



(a) Temperature Field for Ra=10^{11} (b) Velocity Field for Ra=10^{11}

Figure 8: **Temperature and velocity contours for Ra=**10^{11}

Figure 9 presents the surface-average Nusselt at the hot wall as the function of Rayleigh number, as shown, the Nusselt number increase with Ra, due to the increase of heat transfer by convection. The fourth point shows a discrepancy of the other points because it has more than one solution.

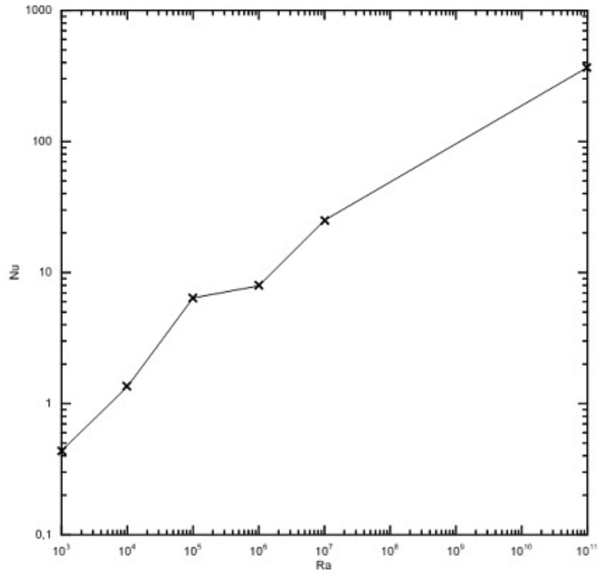


Figure 9: **Averaged-Nusselt number as function of Rayleigh number**

4. CONCLUSION

The present work studied the Rayleigh-Bernard convection of air inside a two-dimensional cavity containing a heated body. For $Ra = 10^3$ it is observed a symmetric pattern for the isotherm and stream lines besides a steady behavior. When Ra reaches to 10^4, the symmetric behavior it is lost and it is observed the presence of heat transfer by convection. At $Ra = 10^5$ the regime flow become unsteady and it is also noticed an increase of convection. For the following value of Ra, 10^6, convection become the predominant way of heat transfer and so do $Ra = 10^7$; For both contours of temperature and velocity it is possible to realize the return for a symmetric shape. At $Ra = 10^{11}$ the temperature become uniform along the cavity and the velocity contour did not show any pattern. Moreover, the residuals did not converge, showing the need to implement a turbulence model. Regarding to the Nusselt number, it is perceptible that the Nu increases in function of the convection and also indicates that the geometry of the body has influence in the heat transfer and consequently in the results of Nu.

5. ACKNOWLEDGMENTS

The authors are grateful for the financial support provided by Brazilian funding agencies: CNPq, FAPERJ and CAPES.

6. REFERENCES

Deardorf, J. and Willis, G., 1965. “The effect of two-dimentionality on the suppression of thermal turbulence”. In *Journal of Fluid Mechanics*. Vol. 23, pp. 337–353.

Ha, M.Y., Kim, I.K. and Yoon, H.S., 2002. “Two-dimensional and unsteady natural convection in a horizontal enclosure with a square body”. In *Numerical Heat Transfer*.

Rodrigues, V.J.N. and Su, 2016. “Computer simulation of natural convection in a square cavity containing an heated body”.

7. RESPONSIBILITY NOTICE

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