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

ENCIT-2018
TWO-DIMENSIONAL COMPUTATIONAL SIMULATION IN A CYLINDER BY NATURAL CONVECTION

Beatriz Machado dos Santos
Wallthynay Ferreira de Arruda
Federal University of Rio de Janeiro - UFRJ
bmachado@nuclear.ufrj.br, warruda@nuclear.ufrj.br

Jian Su
Federal University of Rio de Janeiro - UFRJ
sujian@nuclear.ufrj.br

Abstract. *In this paper, a preliminary two-dimensional study on conjugate natural convection in horizontal cylinder with a hot square body wrapped in a fluid, air, whose Prandt value is 0.7 will be developed. The laminar regime was adopted in the first simulation in order to simplify the problem. The governing equations were obtained at the literature and the simulations were developed numerically by the CFD ANSYS Fluent 18.2 package. The quality of the mesh was verified and the preliminary results are presented in this paper.*

Keywords: *conjugate natural convection, air, heat transfer*

1. INTRODUCTION

The natural convection heat transfer has been widely used in research due to its technological applications in nuclear reactors, being a low cost and reliable method. The natural convection has received special attention in the literatures. Several works whose boundary conditions is isothermal walls were published.

Farouk and Guçeri (1982) have presented numerical solutions for the steady-state, two-dimensional natural convection in the annulus between two horizontal concentric cylinders which were held at different constant temperatures.

Shu and Xue (2001) have numerically studied the natural convective heat transfer in a horizontal eccentric annulus between an isothermal square outer and a heated isothermal circular inner cylinder using differential quadrature (DQ) method.

Mizushima *et al.* (2001) investigated theoretically by assuming two-dimensional and incompressible flow fields, transitions of natural convection in an annulus between horizontal concentric cylinders. It was assumed that the inner cylinder is kept at a higher temperature than the outer cylinder, and was confirmed by numerical simulations that dual stable steady solutions exist for Rayleigh numbers larger than a critical value.

Yoo (2003) investigated natural convection in a horizontal annulus with a constant heat flux wall for the fluids of $0.2 \leq Pr \leq 1$. The outer cylinder was kept at a constant temperature, and the inner cylinder was heated with a constant heat flux. By using a numerical approach in solving the unsteady governing equations of flow and temperature fields, it was shown that dual steady solutions exist above a critical Rayleigh number.

Few studies have considered isoflux inner wall boundary condition. Kumar (1988) and Kumar and Keyhani (1990) have studied the natural convection heat transfer in horizontal annuli with inner and outer cylinders maintained at constant heat flux and temperatures, respectively.

The objective of this paper is to study a geometry with focus on the analysis of dry storage in order to solve a real problem in Brazil, the lack of space in the pools of the Angra Nuclear Power Plants.

2. GOVERNING EQUATIONS

According to Sambamurthy *et al.* (2008) the flow and temperature distributions are assumed to be two-dimensional and are governed by continuity, Navier−Stokes, and fluid and solid energy equations.

$$\frac{\partial \omega^*}{\partial t^*} + \frac{\partial}{\partial x^*}(u^*\omega^*) + \frac{\partial}{\partial y^*}(v^*\omega^*) = \left(\frac{\partial^2 \omega^*}{\partial x^{*2}} + \frac{\partial^2 \omega^*}{\partial y^{*2}}\right) + Gr\frac{\partial T_f^*}{\partial y^*}cos\Phi x - Gr\frac{\partial T_f^*}{\partial x^*}cos\Phi y \quad (1)$$

$$\frac{\partial^2 \Psi^*}{\partial x^{*2}} + \frac{\partial^2 \Psi^*}{\partial y^{*2}} = -\omega^* \quad (2)$$

$$\frac{\partial T_f^*}{\partial t^*} + \frac{\partial}{\partial x^*}(u^* T_f^*) + \frac{\partial}{\partial y^*}(v^* T_f^*) = \frac{1}{Pr}(\frac{\partial^2 T_f^*}{\partial x*^2} + \frac{\partial^2 T_f^*}{\partial y*^2}) \tag{3}$$

$$\rho_s^* c_s^* \frac{\partial T_s^*}{\partial t^*} = \frac{\lambda_s^*}{Pr}(\frac{\partial^2 T_f^*}{\partial x*^2} + \frac{\partial^2 T_f^*}{\partial y*^2}) + \frac{1}{Pr} \tag{4}$$

where λ_s^* is the solid-to-fluid thermal conductivity ratio and

$\omega^* = \frac{\partial v^*}{\partial x^*} - \frac{\partial u^*}{\partial x^*}$ $\quad u^* = \frac{\partial \Psi^*}{\partial y^*}$ $\quad v^* = \frac{\partial \Psi^*}{\partial x^*}$

The system of non-dimensionalisation is:

$t^* = \frac{t\nu_f}{R_0^2}$ $\quad X^* = \frac{x}{R_0}$ $\quad Y^* = \frac{y}{R_0}$

$u^* = \frac{uR_0}{\nu_f}$ $\quad v^* = \frac{vR_0}{\nu_f}$ $\quad \omega^* = \frac{\omega R_0^2}{\nu_f}$

$\Psi^* = \frac{\Psi}{\nu_f}$ $\quad \Psi^* = \frac{\Psi}{\nu_f}$ $\quad \rho_s^* = \frac{\rho_s}{\rho_f}$

$c_s^* = \frac{c_s}{\rho_{p,f}}$ $\quad \lambda_s^* = \frac{\lambda_s}{\lambda_f}$ $\quad T^* = \frac{(T-T_r)}{\Delta T}$

$\Delta T = \frac{Q_v R_0^2}{\lambda_f}$ $\quad Gr = \frac{g\beta \Delta T R_0^3}{\nu_f^2}$ $\quad Pr = \frac{\eta_f^2 c_{p,f}}{\lambda_f}//$

3. METHODOLOGY

The physical model for the present work is shown in Fig. 1. A circular cylinder, of diameter D, is located in the origin of the x and y axes as shown. Inside the cicumference there is a solid square cylinder of side L located concentrically inside the circular cylinder.

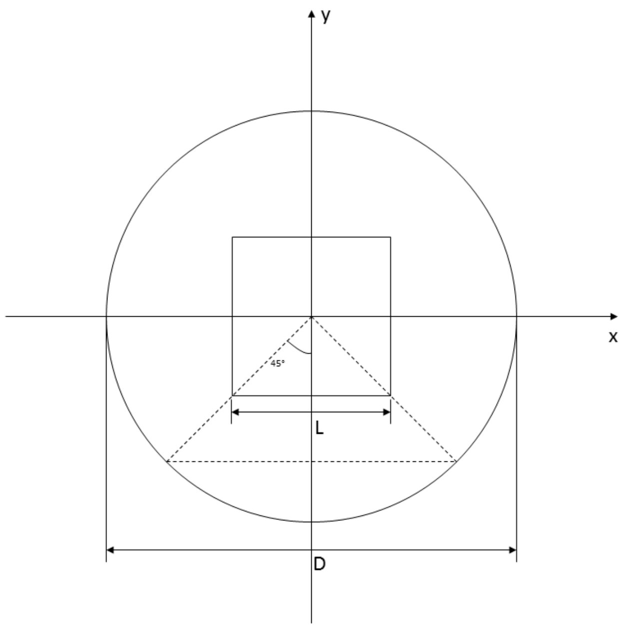

Figure 1. Configuration.

The mathematical model was solved numerically using the commercial CFD ANSYS FLUENT 18.2 package, which uses finite volume techniques to solve problems in non-uniform and uniform, unstructed and hybrid structured meshes. Therefore, its possible to obtain a numerical solution of discrete momentum and mass balance equations. As boundary conditions for this study was assumed that the walls are isothermal (T=300K).

For natural convection to occur, it is necessary that the side of the square (L) and the diameter of the circumference (D) are related up to a maximum value of 0.717, according to Eq. 5.

$$L_{max} = \frac{\sqrt{2}}{2} D \tag{5}$$

Therefore, the ratio of L and D must be less than the one given in Eq. 6.

$$\left(\frac{L}{D}\right)_{max} = \frac{\sqrt{2}}{2} \tag{6}$$

In this study, for simplicity, a value of 0.5 was chosen for the L / D ratio.

Fig. 2 is the geometry inserted in the commercial software. The square and the circumference are considered different bodies, in order to obtain the most suitable mesh for both.

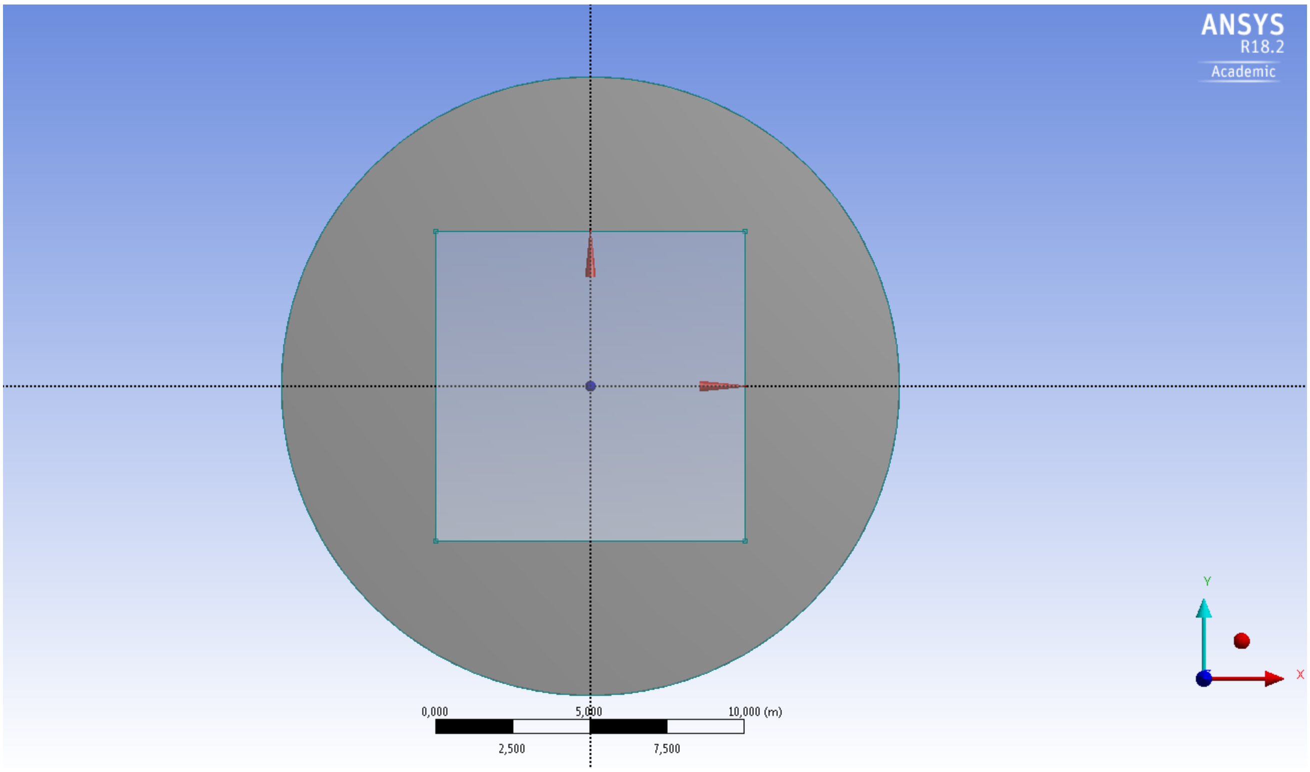

Figure 2. Computational Fluid Dynamics (CFD) geometry.

Table 1 presents the reference properties for the air used in the simulations.

Table 1. Reference fluid properties.

Parameters		
k	Thermal conductivity (W/m.K)	0.00263
β	Volumetric expansion coefficient (K^{-1})	0.00333
ρ	Density (kg/m^3)	1.1614
μ	Viscosity (kg/m.s)	1.846×10^{-5}
cp	Specific heat (J/kg.K)	1007
α	Thermal diffusivity (m^2/s)	2.248×10^{-5}

4. RESULTS

Different refinements for the mesh were verified, in order to guarantee the quality of the simulations results and the mesh convergence. Table 2 shows the meshes used for the convergence study and their respective number of nodes and elements.

Table 2. Number of nodes and elements for each mesh.

Mesh	Number of nodes	Number of elements
1	9032	8906
2	16158	15990
3	29720	29493
4	36168	35918

Fig.3 shows the solutions obtained for different meshes.

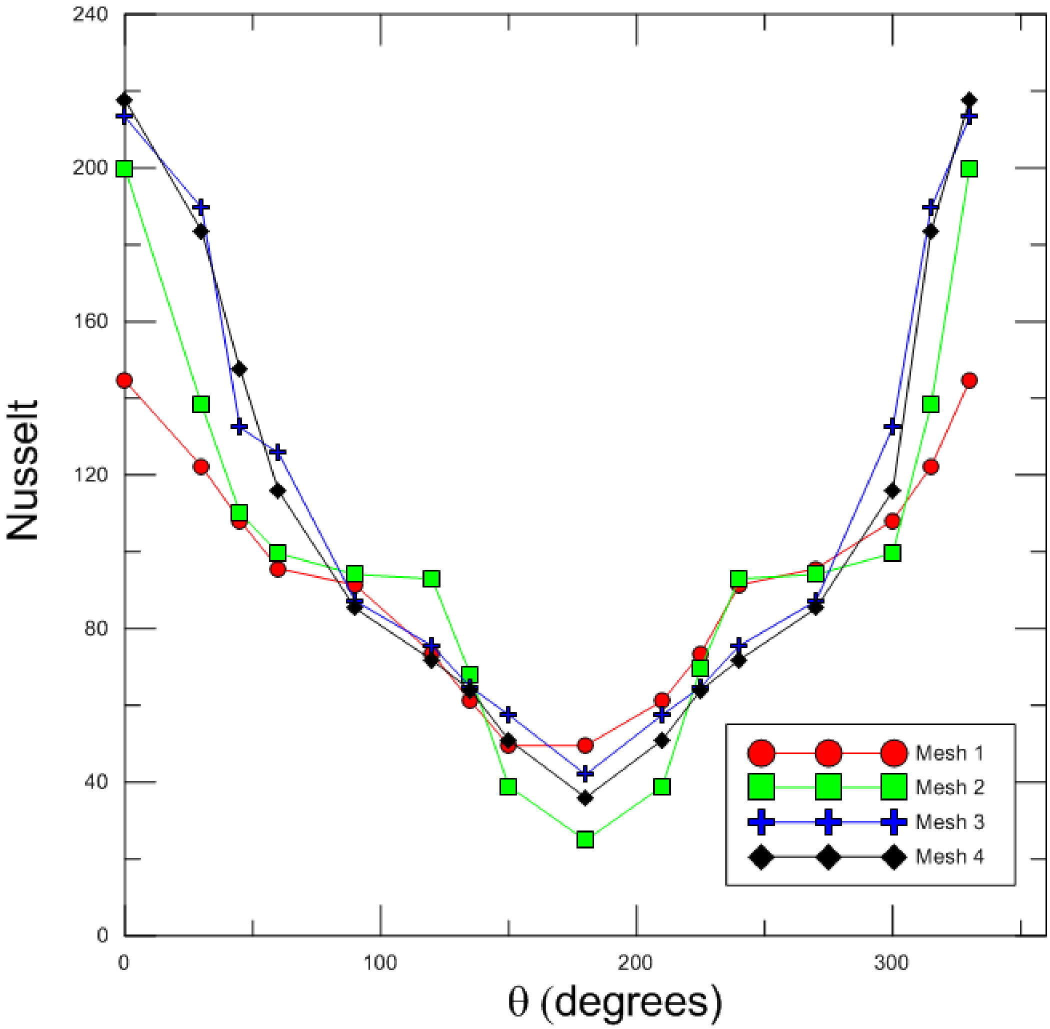

Figure 3. Distribution of Nusselt number the geometry of the cavity.

The results show that mesh 3 is the most appropriate to simulate Rayleigh number variation. It provides reasonably accurate solutions since it is close to the results obtained by mesh 4, but requires less computational effort since it has a smaller number of elements and nodes.

Results to the variation of Rayleigh number from 10^3 to 10^7, considering the laminar regime, were obtained from the simulation performed in the CFD ANSYS Fluent software using the mesh 3. The comparison between the Nusselt number and the Rayleigh number is presented at Fig. 4.

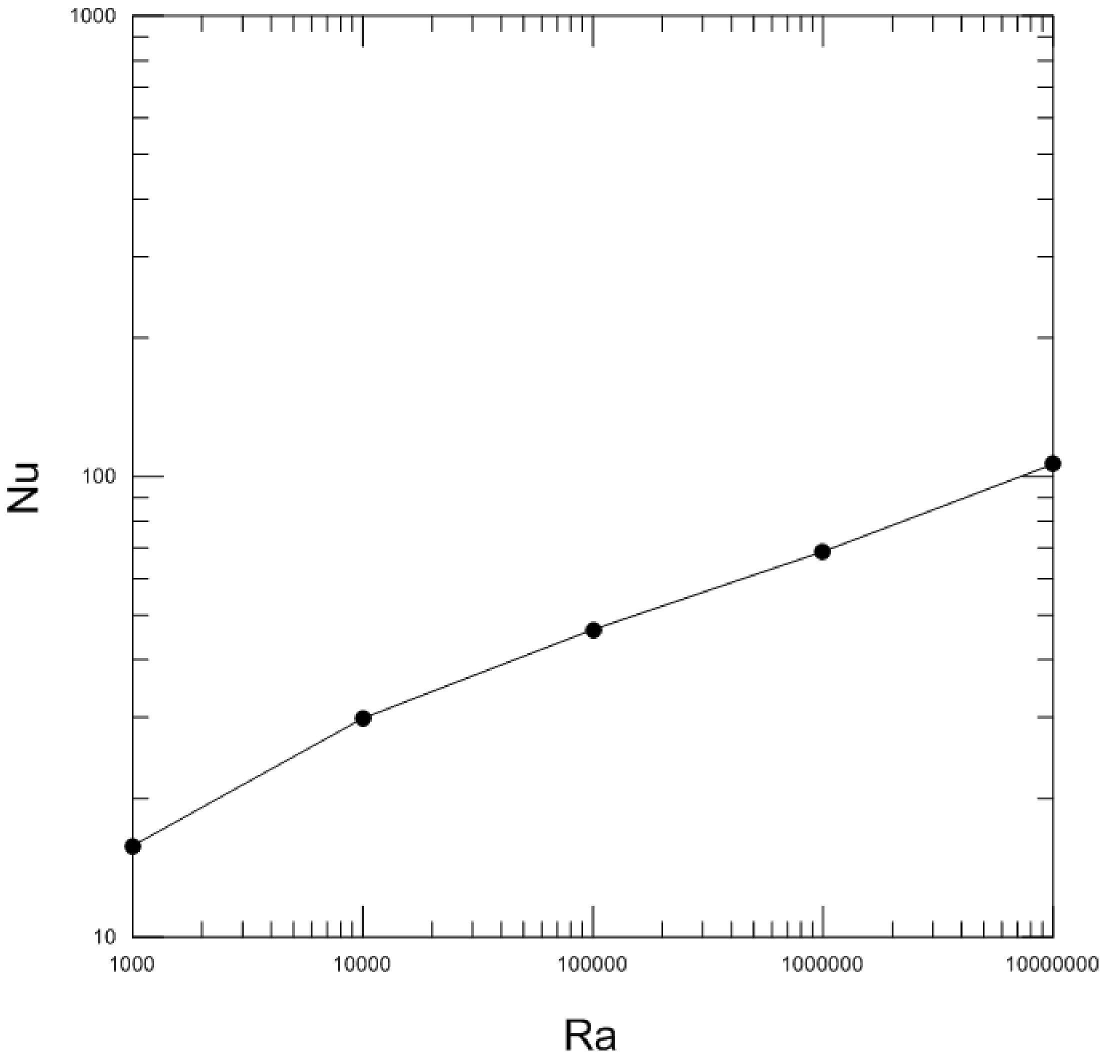

Figure 4. Nusselt number as a function of Rayleigh number.

5. CONCLUSION

We can conclude with these results that simulations are satisfactory according the literature. As we can see, the Nusselt number increase with the Rayleigh number.

The next steps are the variation of the L / D ratio, compare with literature correlations and apply the adequate turbulence model.

6. ACKNOWLEDGEMENTS

The authors are grateful to CNPq and FAPERJ for the financial support.

7. REFERENCES

Farouk, B. and Guçeri, S.I., 1982. "Laminar and turbulent natural convection in the annulus between horizontal concentric cylinders". *Journal of Heat Transfer*, Vol. 104, No. November 1982, pp. 631–636. ISSN 00221481. doi: 10.1115/1.3245178. URL `http://link.aip.org/link/?JHTRAO/104/631/1`.

Kumar, R., 1988. "Study of natural convection in horizontal annuli". *Int. J. Heat Mass Transfer*, Vol. 31(6), pp. 1137–1148.

Kumar, R. and Keyhani, M., 1990. "Flow visualization studies of natural convective flow in a horizontal cylindrical annulus". *Trans. ASME J. Heat Transfer*, Vol. 112, pp. 784–787.

Mizushima, J., Hayashi, S. and Adachi, T., 2001. "Transitions of natural convection in a horizontal annulus". *Int. J. Heat Mass Transfer*, Vol. 44, pp. 1249–1257.

Sambamurthy, N., Shaija, A., Narasimham, G. and Murthy, M.K., 2008. “Laminar conjugate natural convection in horizontal annuli”. *International Journal of Heat and Fluid Flow*, Vol. 29, No. 5, pp. 1347 – 1359. ISSN 0142-727X. doi:https://doi.org/10.1016/j.ijheatfluidflow.2008.04.003. URL `http://www.sciencedirect.com/science/article/pii/S0142727X08000714`.

Shu, C. and Xue, H.and Zhu, Y., 2001. “Numerical study of natural convection in an eccentric annulus between square outer cylinder and a circular inner cylinder using a DQ method”. *Int. J. Heat Mass Transfer*, Vol. 44, pp. 3321–3333.

Yoo, J.S., 2003. “Dual free-convective flows in a horizontal annulus with a constant heat flux wall”. *Int. J. Heat Mass Transfer*, Vol. 46, pp. 2499–2503.

8. RESPONSIBILITY NOTICE

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