CILAMCE 2017
Florianópolis - Santa Catarina - Brazil

# NATURAL CONVECTION IN CONCENTRIC ENCLOSED CAVITIES

**Lais Ferreira Oton**

lais-oton@hotmail.com

**Rebeca Pereira Marcondes**

rebecapereiramarc@hotmail.com

Federal University of Ceará

Technology Center, Department of Chemical Engineering, Campus Pici, block 709, 60020-181, Ceará, Fortaleza, Brasil.

**Francisco Marcondes**

marcondes@ufc.br

Federal University of Ceará

Technology Center, Department of Metallurgical Engineering and Material Science, Campus Pici, block 730, 60440-554, Ceará, Fortaleza, Brasil.

***Abstract.*** *Due to the large spectrum of application in engineering, natural convection in enclosed cavities has been investigated that include cooling of fluids, ventilation of buildings, food processing and refrigeration of heated surfaces. This research consists in the study of the heat transfer in a square concentric cavity with an incompressible two-dimensional flow, whose core is heated through the Joule's effect and it is cooled by natural convection by the fluid that resides in the concentric cavity. There are two domains: the nucleus that is governed by heat conduction with an internal source, from the dissipation of energy by the Joule's effect and the concentric cavity that is filled with a fluid and it is characterized by natural convection. For the concentric cavity, we assumed a turbulent natural convection that is modeled with the* $k$-$\varepsilon$ *and* $k$-$\omega$ *models. The governing equations in both domains are solved using the ANSYS CFX software. As the internal temperature of the cavity is one of the unknowns of the problem, we have an iterative procedure to obtain the steady-steady solution in both domains. The results are presented in terms of isotherms, streamlines, local Nusselt and rate heat transfer for several Prandtl numbers.*

***Keywords:*** *Turbulent flow; Natural convection; Concentric enclosed cavities; Element based Finite-Volume Method.*

# 1 INTRODUCTION

The study of natural convection in elliptical and cylindrical concentric cavities has received a lot of attention by the scientific community. However, the square concentric cavity in which the internal cavity is heat by Joule's effect and it is cooled by natural convection by the fluid confined in the concentric cavity has not been investigated yet. This configuration can be viewed as a first approximation of the complex heat transfer phenomenon that takes place in electrical transformers.

The study of heat transfer by natural convection in a cavity containing a conductive solid and a cooling fluid was first analyzed by House et al. (1990). In this work, the authors evaluated the natural convection in a concentric square cavity heated on the left side and cooled on the right side, while the upper and lower horizontal walls were maintained adiabatic.

Oh (1997) carried out studies on a square cavity containing a conductive solid with heat generation per unit of volume. The left lateral wall had a fixed temperature larger than the right wall, while the upper and lower horizontal walls were kept adiabatic. Recently, Lima (2014) carried out studies for the same geometry, but with a constant heat flux in the adjacent vertical walls.

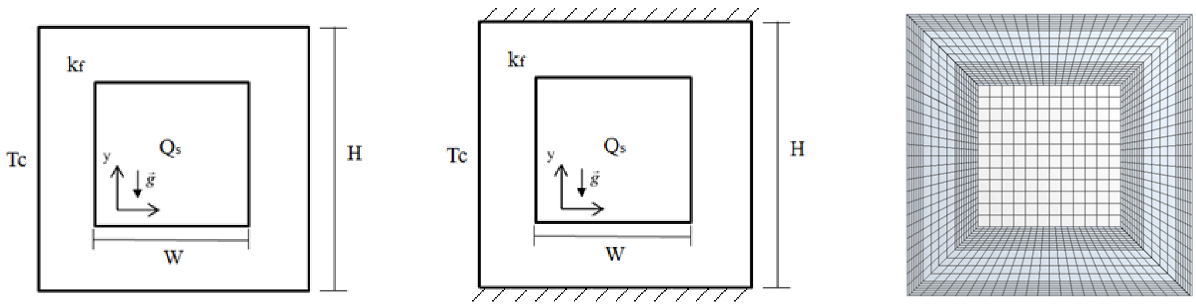
The phenomenon of heat transfer in an enclosed cavity depends on several factors such as the geometric shape of the cavity, the inclination of the cavity, and the boundary conditions on the walls. In this work, the fluid flow and heat transfer in a concentric square cavity will be investigated. The inner cavity is heated by Joules effect and its external walls are cooled by the fluid placed in the concentric cavity. Herein, we investigate the effect the fluid resident into the concentric cavity through the Prandtl number and the boundary conditions on the external walls of the cavity; for this, we assumed two types of boundary conditions: isothermal walls, and isothermal vertical walls with adiabatic horizontal walls.

The cavity models and the mesh were created in ANSYS CFX software. We used the k-$\varepsilon$ and k-$\omega$ turbulent models to represent the turbulent fluid flow in the concentric region. The results are presented in terms of isotherms, streamlines and local and average Nusselt number for three types of fluid (air, water, and oil).

# 2 MATHEMATICAL FORMULATION

The problem studied refers to a two-dimensional square cavity of height H, with a centered square block of length W. The concentric region between the internal block and the external cavity is filled up with a thermal conductivity fluid. We are going to refer from now on in this work, the internal block as domain 1 and concentric region as domain 2.

In domain 1, we have a conductive metal in which we have a generated heat (Qs) due to a current flowing through the conductor nucleus. For cooling of the internal nucleus, we have place a fluid in the concentric cavity that is governed by natural convection. Herein, we investigated the effect of the three fluids for cooling the internal nucleus, Prandtl numbers ($Pr$ = 0.7, 6.13, and 3404.16) that correspond to air, water, and mineral oil. Two types of boundary conditions were used for the external wall of the concentric cavity (domain 2). For the first type of boundary condition, we assumed that the external walls of the cavity were equal to ambient temperature (298.15 K). For the second type of boundary condition, we assumed that the vertical walls were equal to the ambient temperature, but the horizontal walls were adiabatic. Figures 1(a) and 1(b) presents the boundary conditions for the external walls. The dimensions of the cavity as well as the physical parameters of the nucleus are shown in Tab. 1.




**(a) Geometry and boundary conditions of model 1** **(b) Geometry and boundary conditions of model 2** **(c) Mesh used throughout the domain**

**Figure 1: Mesh and boundary conditions of the models**

**Table 1: Dimensions of the cavity and physical properties of the nucleus**

| Nucleus Properties | Value |
|---|---|
| Height of the cavity ($m$) | 2 |
| Width of the nucleus ($m$) | 0.05 |
| Thermal conductivity of the nucleus ($W/m.K$) | 52.9 |
| Specific heat capacity of the nucleus at constant pressure ($J/kg.K$) | 486 |
| Rate of internal heat generation per unit of volume ($W/m^3$) | 6866.62 |

Assuming all fluid properties constant, except the buoyancy term in the y-direction, incompressible fluid flow, steady state regime, thermal radiation negligible, the average equations for mass, momentum and energy along with x and y directions can written as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0 \tag{1}$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_M \tag{2}$$

$$\frac{\partial T}{\partial t} + \frac{\partial U_j T}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{\nu}{Pr} + \frac{\nu_t}{\sigma_t}\right)\frac{\partial T}{\partial x_j}\right] \tag{3}$$

Two models of turbulence were implemented. The k-$\varepsilon$ model, based on the eddy viscosity concept, and Wilcox's (2006), k-$\omega$ model, which assumes that the turbulent viscosity is obtained as a function of kinetic energy and turbulent frequency. Both models relate the mean velocity and turbulent viscosity gradients. The physicals models and the respective constants for turbulent regimes are given by Eqs. 4 though 21.

The equations for k-$\omega$ model are given by

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \beta' \rho k \omega \tag{4}$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \tag{5}$$

$$P_k = \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} - \frac{2}{3}\frac{\partial U_k}{\partial x_k}\left(3\mu_t \frac{\partial U_k}{\partial x_k} + \rho k\right) \tag{6}$$

$$\mu_t = \rho \frac{k}{\omega} \tag{7}$$

$$p' = p + \frac{2}{3}\rho k \tag{8}$$

$$\mu_{eff} = \mu + \mu_t \tag{9}$$

The following constants for the k-$\omega$ model were used:

$$\beta' = 0.09 \tag{10}$$

$$\alpha = 5/9 \tag{11}$$

$$\beta = 0.075 \tag{12}$$

$$\sigma_k = 2 \tag{13}$$

$$\sigma_\omega = 2 \tag{14}$$

The k-$\varepsilon$ model is given by Eqs. (15) through (17). The turbulence production due to viscous forces, is the same used in the k-$\omega$ model, (Eq. 6). The difference is in the turbulence viscosity ($\mu_t$), that is given by Eq. (17).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho \varepsilon \tag{15}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho U_j \varepsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon) \tag{16}$$

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{17}$$

The constants of the k-$\varepsilon$ model are given by:

$$C_{\varepsilon 1} = 1.44 \tag{18}$$

$$C_{\varepsilon 2} = 1.92 \tag{19}$$

$$\sigma_k = 1 \tag{20}$$

$$\sigma_\varepsilon = 1.3 \tag{21}$$

For all the equations described above, $U_i$ represents the Cartesian velocity components ($m/s$); $\rho$ the fluid density ($kg/m^3$); T is the temperature ($K$); $\sigma_t$ is the turbulent Prandtl number; $\omega$ and $\epsilon$ are the specific dissipation and the rate of turbulent dissipation ($m^2/s^2$), respectively; $\nu_t$ is the kinematic turbulent viscosity ($m^2/s$); $\mu$ is the dynamic viscosity (kg/m s); $\nu$ is the kinematic viscosity ($m^2/s$); k is the kinetic turbulent energy ($m^2/s^2$); $P_k$ is the turbulence production rate ($kg/ms^3$); $\mu_t$ is the turbulent viscosity ($kg/ms$).

In the interface of domains 1 and 2, we use the conservative flux, we mean the heat transfer by conduction that arrives at external wall of the nucleus is equal to the advective heat transfer that is transferred to the fluid at this interface. It is important to highlight the iterative nature of the solution of this problem. Since initially the temperature of the external wall of the solid core is unknown, a value is arbitrated for the external temperature of the core and using this temperature and the temperature of the external wall of the concentric cavity, the average temperature of the concentric cavity is calculated. We use this average temperature to evaluate the velocity, pressure, and temperature fields for the concentric cavity. With the values of velocity, and the temperature in the cavity and the temperature of the solid core, we obtain another average temperature for external wall of the nucleus and another iterative cycle starts. We stop this process when the difference between the heat transfer rates evaluated at the external wall of the nucleus and the external wall of the concentric cavity are smaller than 0.5 W. The boundary conditions for the cavity model 1 are given by

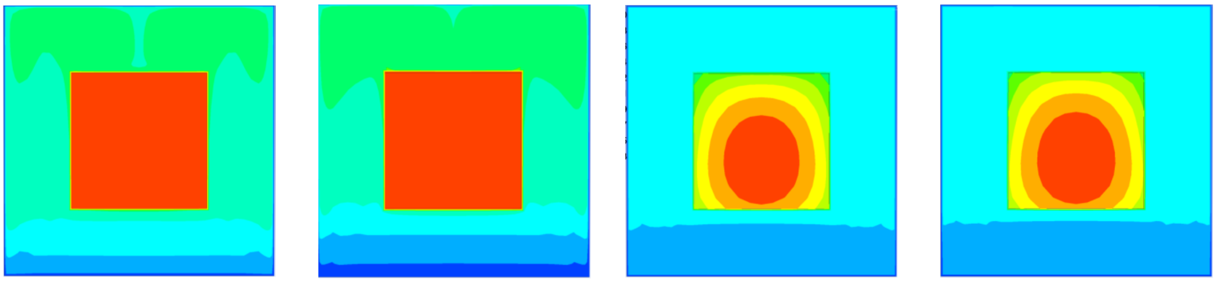
$$u(x=0,y) = \mathrm{v}(x=0,y) = 0 \qquad T(x=0,y) = T_c \tag{22}$$
$$u(x=H,y) = \mathrm{v}(x=H,y) = 0 \qquad T(x=H,y) = T_c \tag{23}$$
$$u(x,y=0) = \mathrm{v}(x,y=0) = 0 \tag{24}$$
$$u(x,y=H) = \mathrm{v}(x,y=H) = 0 \tag{25}$$

The boundary conditions for the cavity model 2 are given by

$$u(x=0,y) = \mathrm{v}(x=0,y) = 0 \qquad T(x=0,y) = T_c \tag{26}$$
$$u(x=H,y) = \mathrm{v}(x=H,y) = 0 \qquad T(x=H,y) = T_c \tag{27}$$
$$\left.\frac{\partial T}{\partial y}\right)_{y=0} = 0 \qquad u(x,y=0) = \mathrm{v}(x,y=0) = 0 \tag{28}$$
$$\left.\frac{\partial T}{\partial y}\right)_{y=H} = 0 \qquad u(x,y=H) = \mathrm{v}(x,y=H) = 0 \tag{29}$$

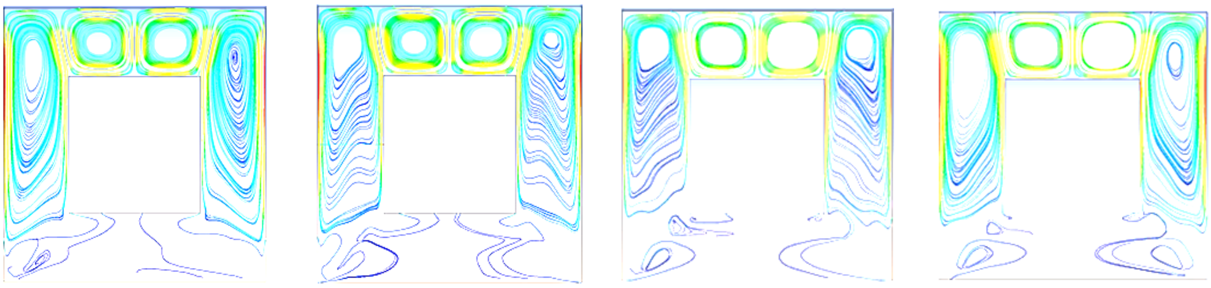
# 3 RESULTS

For the boundary conditions presented in the previous section for model 1 using both turbulence models, the temperature field and streamlines for the natural convection heat transfer occurring between the solid core and the cooling fluid are presented in Figs. 2 and 3, respectively, for air and water. For this cavity model, when the Prandtl number was increased the number of iterations to obtain convergence was largely increased, and therefore we could not obtain a converged solution for the mineral oil. One possible reason for such behavior could be attributed to the thermal instabilities that occurs in the concentric cavity. From the results presented in Figs. 2 and 3, we can verify that, in general, both turbulence models kept the same pattern for the thermal and fluid flow, although some minor differences were observed for the thermal field, especially for small Prandtl number.



**(a)** $k-\omega$, $Pr = 0.7$ **(b)** $k-\varepsilon$ , $Pr = 0.7$ **(c)** $k-\omega$, $Pr = 6.13$ **(d)** $k-\varepsilon$ , $Pr = 6.13$

**Figure 2: Isotherms for model 1.**



**(a)** $k-\omega$, $Pr = 0.7$ **(b)** $k-\varepsilon$ , $Pr = 0.7$ **(c)** $k-\omega$, $Pr = 6.13$ **(d)** $k-\varepsilon$ , $Pr = 6.13$

**Figure 3: Streamlines for model 1.**

The rate of heat transfer and the average temperature of the wall of the solid nucleus are presented in Tab. 2. From this table, we can verify that the temperature of the solid decreases and the rate of heat transfer increases when the Prandtl number increases. From this table, we can also observe that the largest difference between the average temperature and rate of heat transfer were observed for small Prandtl number (air). Such a behavior, can be justified by the local Nusselt number presented in Fig. 4 along the left vertical wall of the external wall of the solid nucleus in the y-direction.

**Table 2: Average temperature and rate of heat transfer - model 1.**

| Prandtl | 0.7 | | 6.13 | |
|---|---|---|---|---|
| Turbulence model | $k-\varepsilon$ | $k-\omega$ | $k-\varepsilon$ | $k-\omega$ |
| Average temperature of the nucleus $(K)$ | 363.8 | 359.9 | 308.7 | 307.52 |
| Rate of heat transfer $(W)$ | 48.2 | 43.2 | 337.8 | 338 |

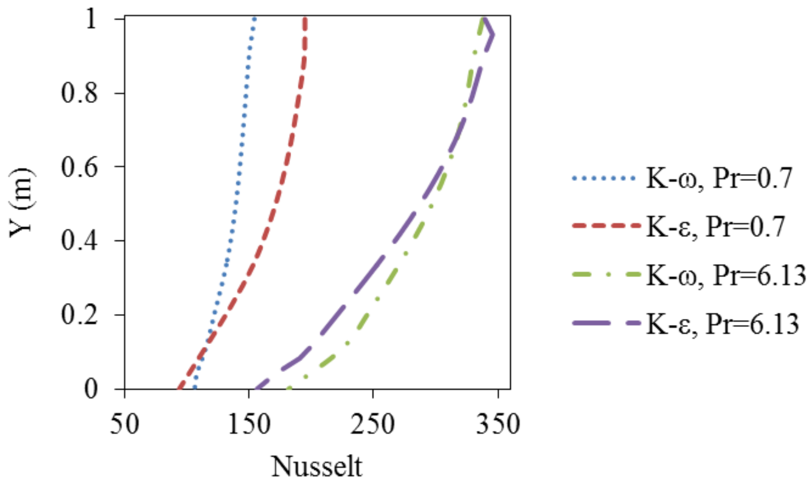
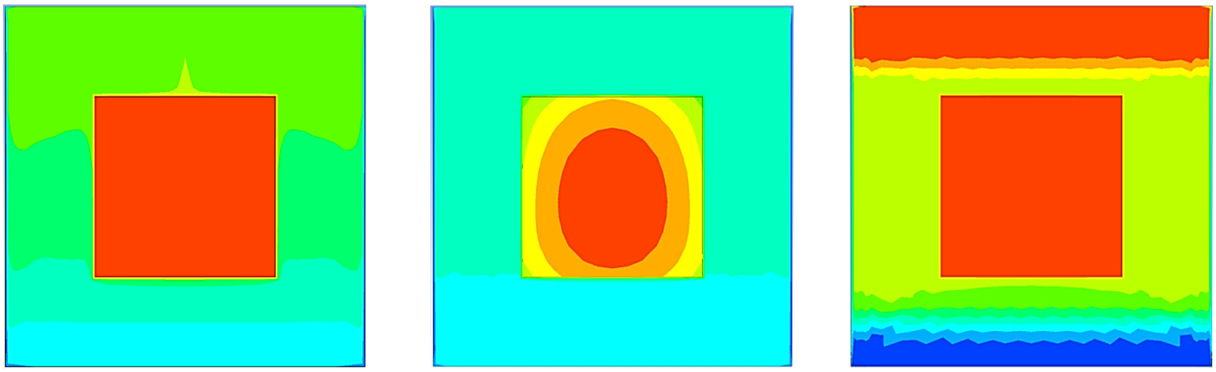


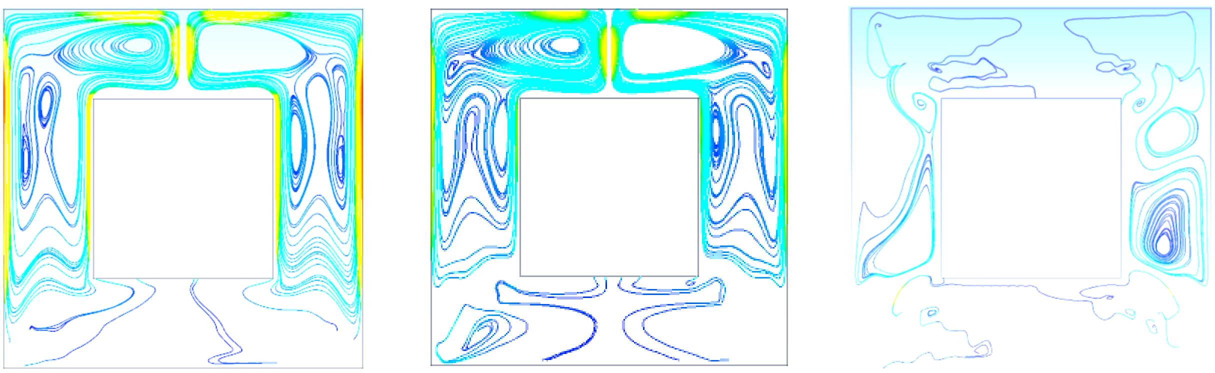

**Figure 4: Nusselt curve along the y-axis - model 1**

The results obtained for the second cavity model are shown next. For this cavity model, only the k-$\omega$ turbulence model was implemented. The temperature field and streamlines for this cavity model, for the three types of fluids are presented in Figs. 5 and 6.



**(a)** $Pr = 0.7$ **(b)** $Pr = 6.13$ **(c)** $Pr = 3404.16$

**Figure 5: Isotherms for model 2**



**(a)** $Pr = 0.7$ **(b)** $Pr = 6.13$ **(c)** $Pr = 3404.16$

**Figure 6: Streamlines for model 2**

**Table 3: Average temperature and rate of heat tranfer - model 2**

| Prandtl | 0.7 | 6.13 | 3404.16 |
|---|---|---|---|
| Average cavity temperature ($K$) | 309.67 | 314.61 | 417.33 |
| Heat flux ($W$) | 12.67 | 337.39 | 215.94 |

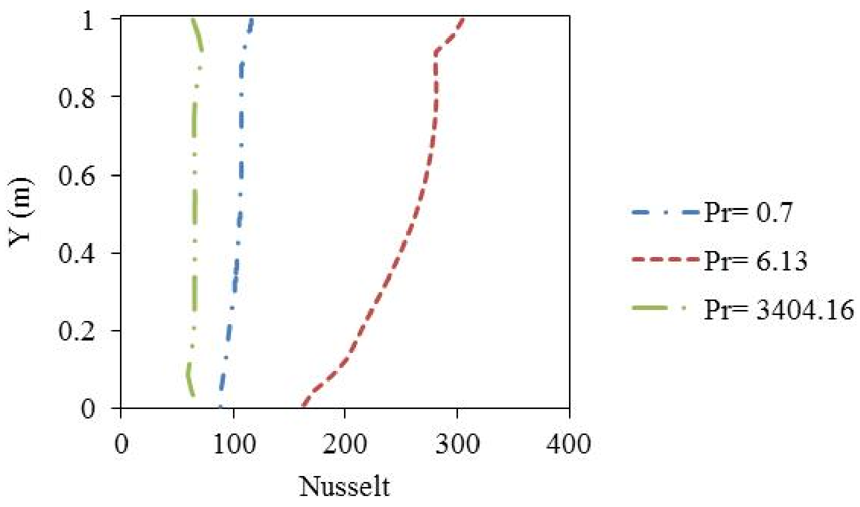



**Figure 7: Nusselt curve along the y-axis - model 2**

From Figs. 5 and 6 it is possible to identify the effect of the dynamic viscosity for the different fluids in the natural convection process. According to Fig. 5, the higher Prandtl fluid less movement in the cavity occurs, even though the rate of heat transfer is high, as it is possible to observe from Tab. 3.

Figure 7 shows the behavior of the local Nusselt number along the y-axis in the left vertical wall of the solid nucleus. According to the graph, it is possible to verify the different behaviors for each fluid. In this case, the fluid with the highest Prandtl number and viscosity did not presented the highest coefficient of heat transfer. The reason for such a behavior will be investigated in the further analysis of this problem.

## 4 CONCLUSIONS

In this study the heat transfer in a concentric enclosed cavity was analyzed by means of the EbFVM (Element based Finite-Volume Method), where the inner solid nucleus is heated by Joules effect and the nucleus is cooled by the fluid placed in the concentric cavity; the effect of three types of fluids (air, water, and mineral oil) in the concentric cavity were investigated. For the external wall of the cavity, we analyzed two types of boundary conditions: all walls of the external cavity kept at the ambient temperature, and the vertical wall kept at the ambient temperature and the horizontal walls adiabatic. These boundary conditions and the fluid placed in the concentric cavity had a large impact in the fluid flow and the heat transfer of the cavity as it was verified by the temperature field, streamlines, and local Nusselt number presented in this work.

# REFERENCES

House, J.M.; Beckermann, C. e Smith, T.F. Effect of a centered conducting body on natural convection heat transfer in an enclosure. *Numerical Heat Transfer*, v. 18, n. 2, 213225, 1990

Lima, Thiago Parente. Natural convection in square cavity, with intern solid, heated and cooled by the adjacent walls  State University of Campinas, Mechanical Engineer.  Campinas-SP , 2014. (In Portuguese).

Oh, J.Y.; Ha, M.Y. e Kim, K.C. Numerical study of heat transfer and flow of natural convection in an enclosure with a heat-generating conducting body. *Numerical Heat Transfer*, Part A Applications, v. 31, n. 3, 289303, 1997.

Wilcox, D.C., "Multiscale model for turbulent flows". *AIAA 24th Aerospace Sciences Meeting*, 1986.