COBEM-2017-0679
HEAT TRANSFER IN A TAYLOR-COUETTE FLOW

Vinícius Hagemeyer Chiumento
Laboratory of Applied Mathematics and Sientific Computing, Departament of Applied Mathematics and Statistics, ICMC - University of São Paulo, São Carlos, São Paulo, Brazil
vinicius.chiumento@usp.br

Vinicius Malatesta
Technological Center of Joinville, Federal University of Santa Catarina, Joinville, Santa Catarina, Brazil
vinicius.malatesta@ufsc.br

Abstract. The Taylor-Couette flow is a centrifugal instability phenomenon characterized by the presence of vortices between two concentric cylinders, which affects the torque exerted between the cylinders as well as the heat transfer from one cylinder to another. The flow was modeled using the software Ansys Fluent, which uses the finite volume method, with the LES method. Various simulations were conducted in order to know the flow behavior in respect to dynamics and the heat transfer at different Reynolds numbers. The results were validated by obtaining quantification of model accuracy with experiments in the literature.

Keywords:Hydrodynamics Instability, Heat Transfer, Taylor-Couette flow

1. INTRODUCTION

The Taylor-Couette flow is a centrifugal instability phenomenon characterized by the presence of counter-rotating vortices between two concentric cylinders, which affects the torque exerted between the cylinders (Fasel and Booz, 1984)(Martínez-Arias *et al.*, 2014) as also as expected the heat transfer from one cylinder to another (Fiebig, 1997). This phenomenon occurs in determined cases when the inter cylinder rotate with a higher angular velocity than the outer cylinder. Taylor-Couette could be an excellent didactic case of hydrodynamics stability theory, and permit to study the behavior of some properties in the presence of vortices.

Some geometrical parameters, which are represented in fig. 1, cam be defined to characterize the flow. The aspect ratio (Γ) is definite by l/d and d is the difference between the radius of both cylinders ($d = R_o - R_i$), the last geometrical parameter is the ratio of radius that is defined by η, $\eta = R_o/R_i$. The Reynolds number cam be defined by $Re = R_i\omega_i d/\nu$ when ω_i is the angular velocity of the inner cylinder and ν is kinematic viscosity.

2. FORMULATION

The *sofware Ansys Fluent* solve the governing equations that are the Navier-Stokes equations (eq. 1)and the Energy equation (eq. 2). The turbulence methods adopted is LES, this method solve the larges vortices and ignore the small vortices, in consequence the computational time required for the simulation are reduced. Navier-Stokes equations are resolved using the numerical procedure SIMPLE (Semi-Implicit Method for Pressure-Linked Equations).The Reynolds number is based on the angular velocity of the inner cylinder and cam be defined by $Re = \omega R_i d/\nu$.

$$\frac{\partial \rho u}{\partial t} + u\frac{\partial \rho u}{\partial x} + v\frac{\partial \rho u}{\partial y} + w\frac{\partial \rho u}{\partial z} = -\frac{\partial P}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

$$\frac{\partial \rho v}{\partial t} + u\frac{\partial \rho v}{\partial x} + v\frac{\partial \rho v}{\partial y} + w\frac{\partial \rho v}{\partial z} = -\frac{\partial P}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

$$\frac{\partial \rho w}{\partial t} + u\frac{\partial \rho w}{\partial x} + v\frac{\partial \rho w}{\partial y} + w\frac{\partial \rho w}{\partial z} = -\frac{\partial P}{\partial z} + \nu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \tag{1}$$
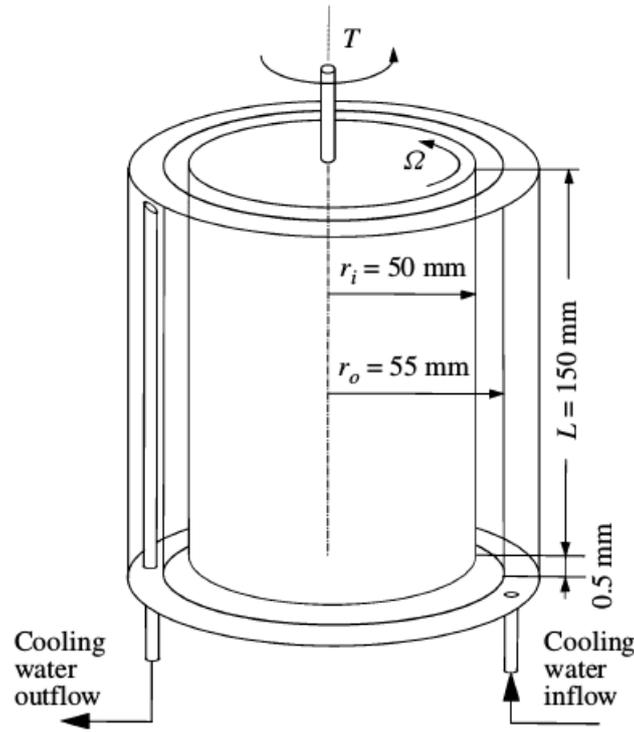
Figure 1. Sketch of the Taylor–Couette system used by Martínez-Arias *et al.* (2014) , drawn to scale.

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

2.1 Heat-transfer

The Nusselt number is a dimensionless number defined by eq. 3 , that number is the total heat transferred divided by the heat transferred only by conduction.

$$Nu = \frac{\dot{Q}_{tot}}{\dot{Q}_{lam}} \tag{3}$$

The quantity of heat transferred only by conduction cam be determined by eq. 4 when T_o and T_i are the temperature of the outer and the inner cylinder respectively, k is the heat conductivity of the water. No one correlation for Nusselt number for Taylor-Couette flow cam be found in literature.

$$\dot{Q} = \frac{2\pi L k (T_o - T_i)}{\ln(r_i/r_o)} \tag{4}$$

2.2 Torque Scaling

A quantitative analogy between Rayleigh-Bénard flow and Taylor-Couette flow cam be utilized to estimate torque between the cylinders. these correlation are proposed by Eckhardt *et al.* (2007) . Eckardt shows that the momentum into Taylor-Couette Flow are transferred between the cylinders in a way very similar to the heat are transferred in the Rayligh-Bénard flow. Is assumed an dimensionless number analog to Nusselt number called ω-Nusselt, these number defined as the Eq. 5 is the total torque between the both cylinders divided by the torque exercised only by an laminar flow. The dimensionless torque, G , defined as Eq 6 when T is the torque exercised by the fluid in the inner cylinder. The laminar dimensionless torque could be estimate by the Eq 7.

$$Nu_\omega = \frac{G}{G_{lam}} \tag{5}$$

$$G = \frac{T}{2\pi l \rho \nu^2} \tag{6}$$

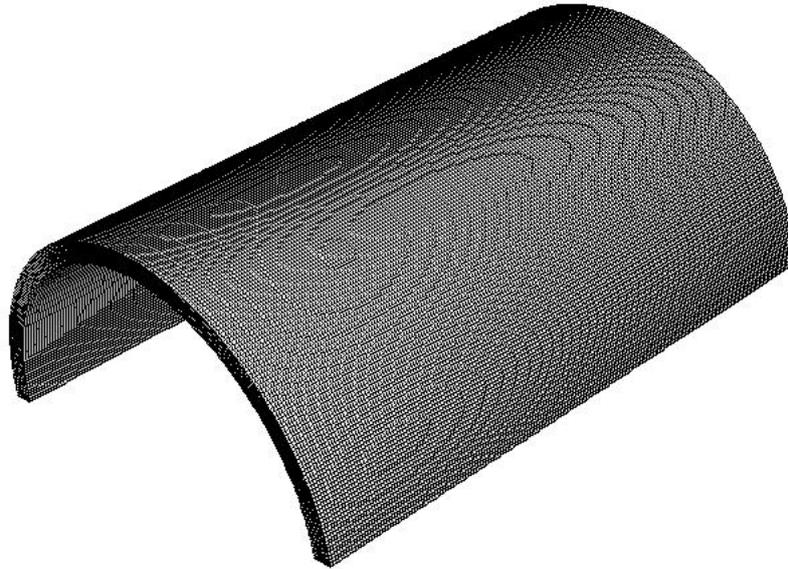$$G_{lam} = \frac{2\eta}{(1+\eta)(1-\eta)} Re \tag{7}$$

Figure 2. Mesh used in simulations.

3. NUMERICAL METHOD

The flow was modeled using the software Ansys Fluent, which uses the finite volume method, with the LES method. Various simulations were conducted in order to know the flow behavior in respect to dynamics and the heat transfer at different Reynolds numbers. The results were validated by obtaining quantification of model accuracy with experiments in the literature.

The mesh, which as represented in fig. 3, utilized has 800 Thousands hexagonal volumes and 848421 nodes, with two hundreds volumes in angular and axial direction and 20 volumes in radial direction. Because a symmetry on the flow, only half of the volume was modeled with the objective do reduce the computational time used in simulations. The geometry of the cylinders are the same as utilized by Martínez-Arias *et al.* (2014), with an aspect ratio ($\Gamma = 30$) and radius ratio $\eta = 0,909$. The time step adopted is 20s and the total time of flow in 40 thousands seconds.

3.1 Boundary Conditions

The extremities of volume are modeled as adiabatic walls. The faces of the cylinder are divided and modeled as radial symmetry boundary conditions. At last the inner cylinder is modeled by a moving wall with a constant velocity and the outer cylinder is is modeled by a stationary wall, both with a constant temperature as defined below:

- $u(r, z, \theta) = 0 (r = R_i, R_e)$;

- $v(r, z, \theta) = 0 (r = R_i, R_e)$;

- $w(r, z, \theta) = V_1 (r = R_i)$ e $0 (r = R_e)$;

- $t(r, z, \theta) = 300k (r = R_i)$ e $350k (r = R_e)$.

4. RESULTS

The torque exerted between the both cylinders grows when the Reynolds number are increased and the heat transfer is a linear function of the Reynolds number. When are compared to experimental results from Martínez-Arias *et al.* (2014). It is observed that the numerical results are higher than experimental results, However experimental and numerical results have the same behavior, which could be observed in fig. 3. There is no paper in literature that could be used to compare the heat transfer in this case. The fig. 4 shows the behavior of Nusselt number in function of Reynolds number. The quantity of heat transferred by conduction are calculated by eq 4, and the laminar torque by eq.7. The number of vortices are not the same when the Reynolds number changes or is used an different mesh.

Analyzing the thermal boundary layer is possible to see in picture 5 that temperature is almost constant inner the vortexes and have a high temperature gradient in the up-wash an down-ash regions. The temperature along a perpendicular line between both cylinders is almost constant in the interior of the vortices, if the line cross an Downwash region or a
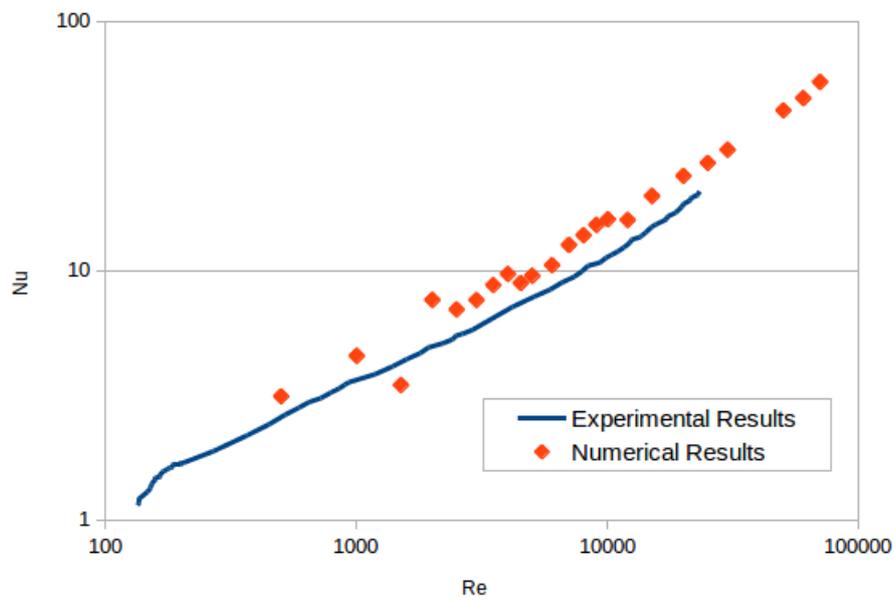
Figure 3. Nu^ω as function of Reynolds number. The numerical results obtained in this work are compared with the experimental results obtained by Martínez-Arias *et al.* (2014).
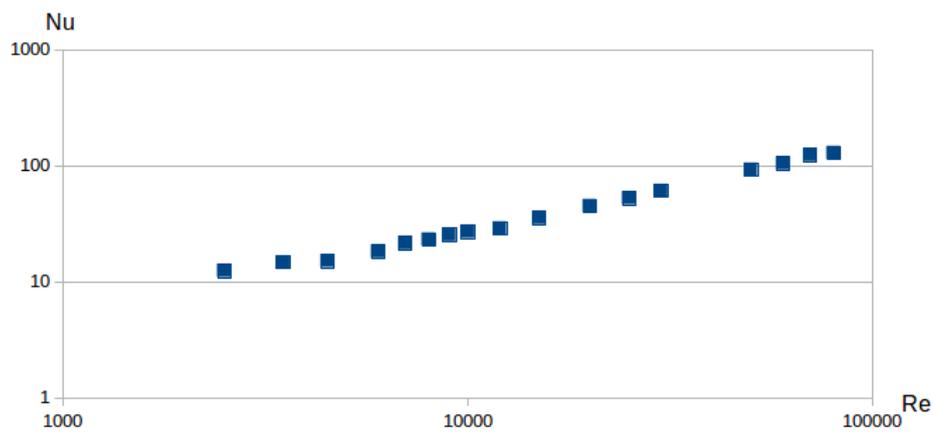

Figure 4. Variation of Nusselt number with Reynolds number.

up-wash region the temperature gradient is very abrupt. They not have much changes for different Reynolds numbers. although the wave length changes, but the average temperature is the same for different cases. Figure 6 show an isotherm surface for temperature igual to 325°C, is possible to see the contour of vortices, as well the upwash and downwash regions.
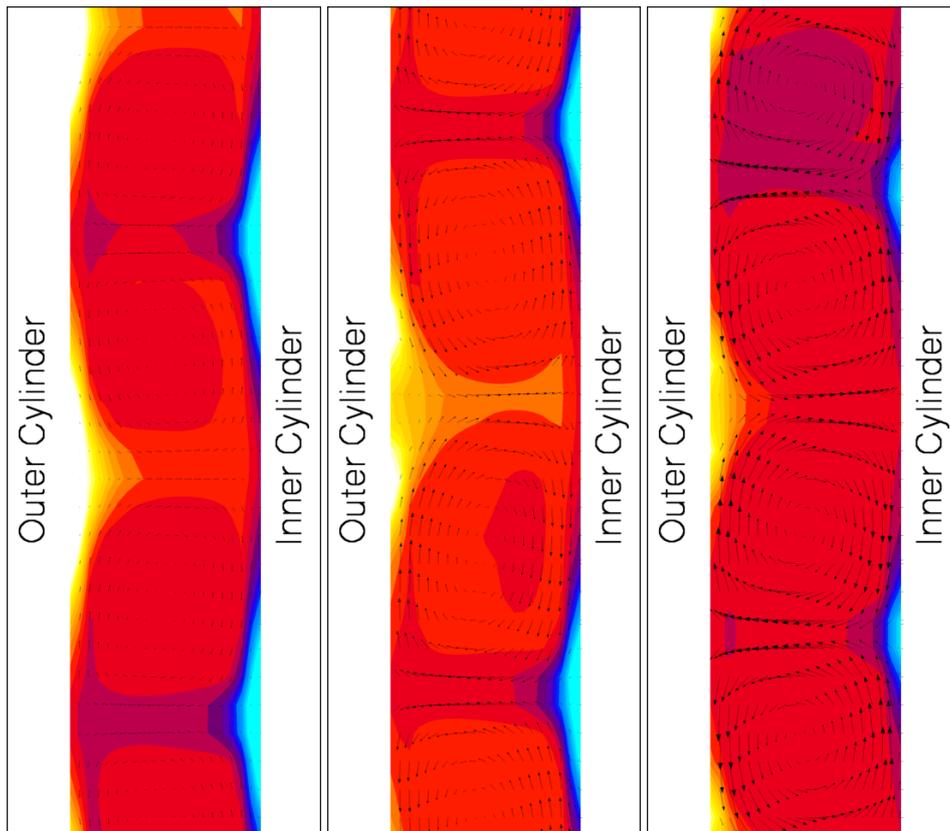
Figure 5. Contours of temperature and vector of velocity magnitude for a section between the cylinders for Reynolds number a)$Re = 2500$ b) $Re = 6000$ c)$Re = 10000$.
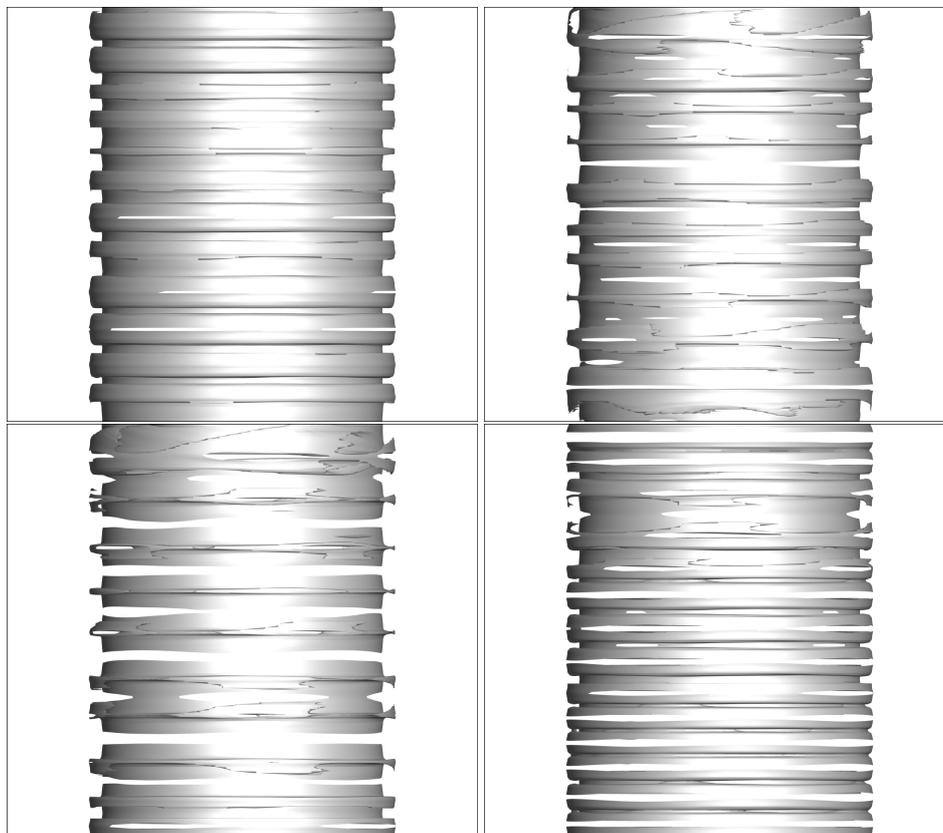

Figure 6. isotherm surface for t = 325°C a) Re = 2500 b) 4500 c) Re = 6000 d) Re = 10000.

5. CONCLUSION

The software Ansys Fluent can simulate the Taylor-Couette flow with satisfactory results and cam represent the behavior very well in a large range of Reynolds number. But when the torque between the both cylinder are compared the results are not very precise. It is observed that the vortices causes a high convection flux of heat between the cylinders and the temperature inside the vortices is almost constant, that result in a high temperature gradient close the cylinders.

6. REFERENCES

Eckhardt, B., GROSSMANN, S. and LOHSE, D., 2007. "Torque scaling in turbulent taylor–couette flow between independently rotating cylinders". *Journal of Fluid Mechanics*, Vol. 581, pp. 221–250. ISSN 1469-7645. doi: 10.1017/S0022112007005629. URL http://journals.cambridge.org/article$_S$0022112007005629.

Fasel, H. and Booz, O., 1984. "Numerical investigation of supercritical taylor-vortex flow for a wide gap". *Journal of Fluid Mechanics*, Vol. 138, pp. 21–52.

Fiebig, M., 1997. "Vortices and heat transfer". *ZAMM-Journal of Applied Mathematics and Mechanics/Zeitschrift für Angewandte Mathematik und Mechanik*, Vol. 77, No. 1, pp. 3–18.

Martínez-Arias, B., Peixinho, J., Crumeyrolle, O. and Mutabazi, I., 2014. "Effect of the number of vortices on the torque scaling in taylor–couette flow". *Journal of Fluid Mechanics*, Vol. 748, pp. 756–767.

7. RESPONSIBILITY NOTICE

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