Fluid-structure interaction problem in Taylor-Couette flow

Jonatas Emmanuel Borges, jonataseborges@gmail.com
Elie Luis Martínez Padilla, epadilla@mecanica.ufu.br
Marcos Antonio de Souza Lourenço, lourenco@dem.feis.unesp.br
Aristeu da Silveira Neto, aristeus@mecanica.ufu.br
Federal University of Uberlândia,
School of Mechanical Engineering,
38400-902, Uberlândia, MG, Brazil

Andre Leibsohn Martins,
Petrobras

Abstract. *As new challenges arise in the deep and ultra-deep exploration of water oilfields by Petrobras, more knowledge and research are needed, so that developing tools to assist in the critical operations is very important to make things practicable. In the context of the drilling process, the complexity of the fluid flow inside the riser is associated with some particularities like the nature of the non-Newtonian flow, the immersed solid particles, a variable eccentricity of the inner cylinder and the superimposed traveling azimuthal waves on the inflow and outflow boundaries of the Taylor vortices. As an attempt to takes the actual operating conditions into account, in the present work we present a study of the fluid-structure interaction problem, where the eccentricity, due the oscillation movement, behave according a springs system acting in the inner cylinder. Using the Navier-Stokes equations, a finite volume discretization method, with second order accuracy in both time and space, was utilized to simulate the Newtonian, single-phase incompressible fluid flow. The circular walls of the inner and outer cylinders are represented by an immersed boundary method, based on the direct multi-forcing model. The determined results allowed to verify the flow structures in a very qualitative way.*

Keywords: *Finite volume method, incompressible fluid flow, fluid-structure, Taylor-Couette.*

1. INTRODUCTION

The flows between rotating concentric cylinders was first studied by Taylor (1923), who investigate experimentally and analytically in the configuration of small gaps between the cylinders (compared with the radius of the internal cylinder). In this particular case, the problem simplifies and becomes dependent only on the Taylor number Ta. It was be shown that when this parameter increases above its critical value, counter-rotating axisymmetric vortices of toroidal shape arises in the flow, also referred as Taylor-Couette instabilities. Later, many other researches had been carried out (Davey, 1962; Eagles, 1977; Wereley and Lueptow, 1998) due to the great number of applications in engineering, mainly due to its important mechanisms of transport and mixing. The Taylor-Couette flow with superposed axial flow, also has been object of many investigations, for same reasons previously mentioned.

Some works with simplified forms of analysis are found in literature, as in: Lockett *et al.* (1992) and Escudier and Gouldson (1995) for concentric configurations and non-Newtonian fluid, Escudier *et al.* (2002a) and Escudier *et al.* (2002b) for fixed eccentric configurations and non-Newtonian fluid. More complex investigations of the Taylor-Couette flow are given for superposed Poiseuille flow (Kaye and Elgar, 1957; DiPrima, 1960; Lueptow *et al.*, 1992) as well as superposed Couette flow (Ludweig, 1964; Weisberg *et al.*, 1997; Hwang and Yang, 2004). In particular, all of these approximations are of great interest to well drilling engineering in the oil and gas production systems.

However, features like the eccentric movement, due the interaction between the internal and external flows, (related to internal channel) and fluids with changeable viscosity due the stress rate (non-Newtonian fluids) or with the presence of particles immersed in the flow makes the numerical and experimental approaches of such flows very difficult to simulate. Due to its complexity, mathematical approximations and physical experiments do not give sufficient detail about the problem, and many numerical approaches have been proposed in the literature. Recently, Hwang and Yang (2004) obtained good results using the finite volume method with second order in space and third order in time discretizations in cylindrical coordinates. Although the cylindrical system of coordinates fit very well in the particular configuration utilizing static cylinders, this could became a limitation when dealing with fluid-structure like simulations.

In this context, the immersed boundary method is a computational cheap, but very efficient alternative to represent the boundaries of a geometrically complex body while using a Cartesian mesh as the Eulerian domain Peskin (1977). Some elaborated models of immersed boundary are the *Physical Virtual Model* by Lima e Silva *et al.* (2003) and, more recently, the *Multi-Direct Forcing* by Wang *et al.* (2007).

In the present work the finite volume method was utilized in the discretization using the immersed boundary method to represent the inner and outer cylinders. Global second order (except near the boundaries of the immersed boundary) was utilized: spatially, with the central difference scheme and using a fractional step method in time. The time accuracy utilized appeared to be quite sufficient for the problems focused in this study. To solve the Poisson's equation in pressure correction step, we choose to utilize the *Strongly Implicit Procedure* (SIP) by Stone (1968).

2. MATHEMATICAL AND NUMERICAL MODELLING

An incompressible and isothermal fluid, with constant physical properties, was considered. The computational modelling is built upon the continuity and Navier-Stokes equations given in dimensional form and Cartesian coordinates as:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{f_i}{\rho} \tag{2}$$

were u_i and p are the velocity components and pressure field respectively, ρ is the density and ν is the kinematic viscosity. The source term f_i include the Eulerian force due the immersed boundary contribution to represent the immersed bodies in the flow. The force field representation are made in a mathematical manner using the auxiliary Dirac delta function $\delta(x)$, as in Eq. (3):

$$\vec{f}(\vec{x}, t) = \int_{\Gamma} \vec{F}(\vec{x}_k, t)\, \delta(\vec{x} - \vec{x}_k)\, d\vec{x}_k \tag{3}$$

where the k denotes a Lagrangian variable and $\vec{F}(\vec{x}_k, t)$ is the Lagrangian force, determined in the points of the solid interface. The Fig. (1) shows the Lagrangian and Eulerian domains representation in yellow and green respectively.

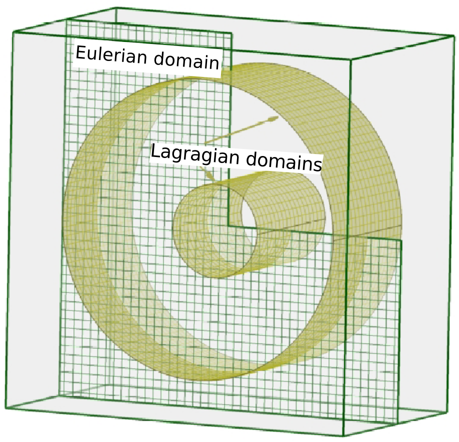

Figure 1: Eulerian and Lagrangian domains representation.

The Lagrangian force is determined using the *multi-direct forcing method* proposed by Wang *et al.* (2007). The model dynamically estimates the fluid force on the solid immersed body surface in the flow. Adding the temporal parameter u^* to Eq. (2) give:

$$\frac{u_i^* - u_i^t}{\partial \Delta t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{f_i}{\rho} \tag{4}$$

It could be seen here that u^* is a mathematical term satisfying the momentum conservation equation. Now the Lagrangian force can be calculated as:

$$\frac{F_k}{\rho} = \frac{u_k - u_i^*}{\Delta t} \tag{5}$$

Determined the force F_k, it is distributed across the Eulerian points around the index k using Eq. (3). In this paper, the discretization of the governing equations Eqs. (1-5), for the Eulerian field uses the finite volume method Patankar (1980) in a staggered computational grid, using the Adams-Bashforth scheme in time and central-differences scheme in space, both of second order. The velocity pressure coupling uses a two step fractional step method by Kim and Moin (1985) with the *Strongly Implicit Procedure* SIP (Stone, 1968) as the Poisson pressure correction solver.

3. PROBLEM DESCRIPTION

In Fig. (2), R_o and R_i are the extern and internal cylinders radius and R_{ex} is the eccentricity radius. Additionally, the channel has length L in the axial direction (perpendicular to figure plane), gap $E = R_o - R_i$, eccentric velocity ω_{ex}, and the following non-dimensional parameters: radius ratio $R = \frac{R_o}{R_i}$, aspect ratio $A = \frac{L}{R_o}$ and the Taylor number $Ta = \frac{\omega R_i E}{\nu}$. For the boundaries in the axial direction it was imposed the periodicity boundary condition.

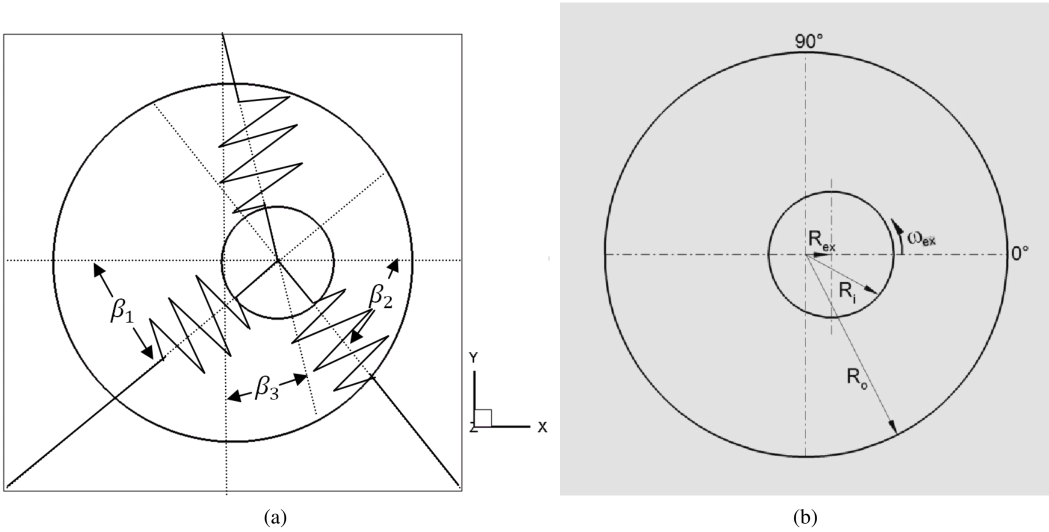

Figure 2: Schematics of the inner cylinder *(a)* supported by three strings and the corresponding β angles and *(b)* at rest.

The fluid-structure interaction simulated in this work has the objective of better realize the operational conditions occurring in real problems. The tests were performed with the inner cylinder sustained by a set of three springs in an arbitrary initial position, as depicted in Fig. 2a. Here was considered that the springs are perfectly elastic, with negligible mass and drag properties. The moment of inertia of inner cylinder has not be considered as well as the forces applied on z direction. Using the Newton's Second Law and Hooke's Law we have the following equations for the resultant forces in each direction:

$$\sum F_X^n = F_{mx}^{n-1} - F_x^n \tag{6}$$

$$\sum F_Y^n = F_{my}^{n-1} - F_y^n \tag{7}$$

where,

$$F_{mx}^{n-1} = -F_1^{n-1}\cos(\beta_1^{n-1}) + F_2^{n-1}\cos(\beta_2^{n-1}) - F_3^{n-1}\sin(\beta_3^{n-1}) \tag{8}$$

$$F_{my}^{n-1} = -F_1^{n-1}\sin(\beta_1^{n-1}) - F_2^{n-1}\sin(\beta_2^{n-1}) + F_3^{n-1}\cos(\beta_3^{n-1}) \tag{9}$$

and were F_i are the total Lagrangian forces acting in the inner cylinder and F_{mi} are the forces applied by the springs. Using the calculated forces is possible to determine the components of acceleration:

$$\begin{aligned} \ddot{x}^n &= \frac{\sum F_X^n}{m} \\ \ddot{y}^n &= \frac{\sum F_Y^n}{m} \end{aligned} \tag{10}$$

whose can be integrate to get the velocities:

$$\begin{aligned} \dot{x}^n &= \dot{x}^{n-1} + \ddot{x}^n \Delta t \\ \dot{y}^n &= \dot{y}^{n-1} + \ddot{y}^n \Delta t \end{aligned} \tag{11}$$

and integrated one second time to obtain the centroid position:

$$\begin{aligned} x^n &= x^{n-1} + \dot{x}^n \Delta t \\ y^n &= y^{n-1} + \dot{y}^n \Delta t \end{aligned} \tag{12}$$

The steps can be summarized as follow:

1. With the initial cylinder position (x_0, y_0) we solve the flow equations;
2. Calculates the Lagrangian force acting on the inner cylinder surface;
3. Calculates the resulting forces using Eqs. (6 and 7);
4. Using the forces calculates the acceleration, velocities and the position of the inner cylinder centroid (x^n, y^n);
5. Update the forces of the springs and go to next time step.

The initial position of the inner cylinder centre was $(x_0 = 0.4, y_0 = 0.6)m$ in all tests.

In order to verify the actual application built using the multi-direct forcing model, was made a comparison with a cylindrical coordinates based code *CCCil3D*, developed by Padilla (2004) and with the last version of the application using the physical virtual model. The CCCil3D has been verified based in the experimental work by Wereley and Lueptow (1999), the numerical simulation by Hwang and Yang (2004) and an analytical approximation by Davey (1962).

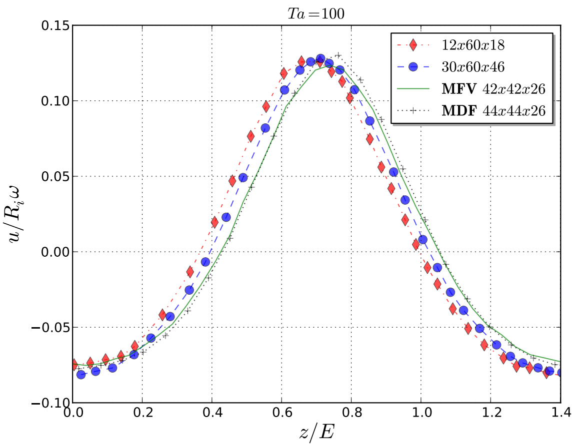

Figure 3: Non-dimensional radial velocity distribution along axial direction to $Ta = 100$.

The compared results are shown in Fig. 3 were we have radial velocity profiles along the z direction in the position $(x = 0.5, y = 0.76)m$, exactly in $\frac{E}{2}$. In figure is evident the better agreement with the mesh refinement, despite the fact of a numerical thickness in the annulus, resultant from the distribution function utilized.

4. RESULTS

We start by releasing the inner cylinder in the initial position (x_0, y_0) and with constant angular velocity ω. The Fig. 4 depicts the time comparison between the development of the velocity components in the classical and fluid-structure interaction cases of the Taylor-Couette flow.

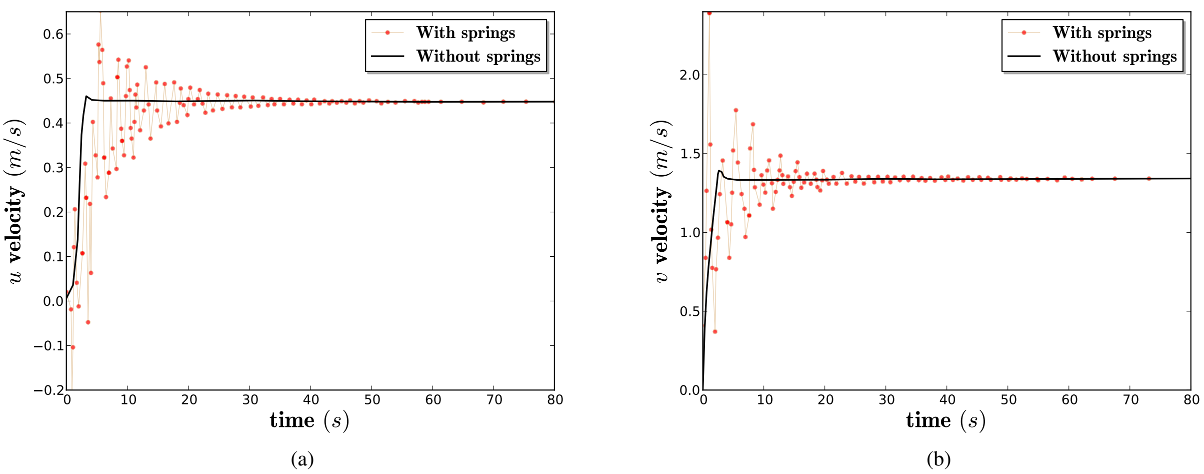

Figure 4: Velocities comparison in the Taylor-Couette flow with (black) and without (gray) fluid-structure interaction; (a) u velocity component and (b) v velocity component.

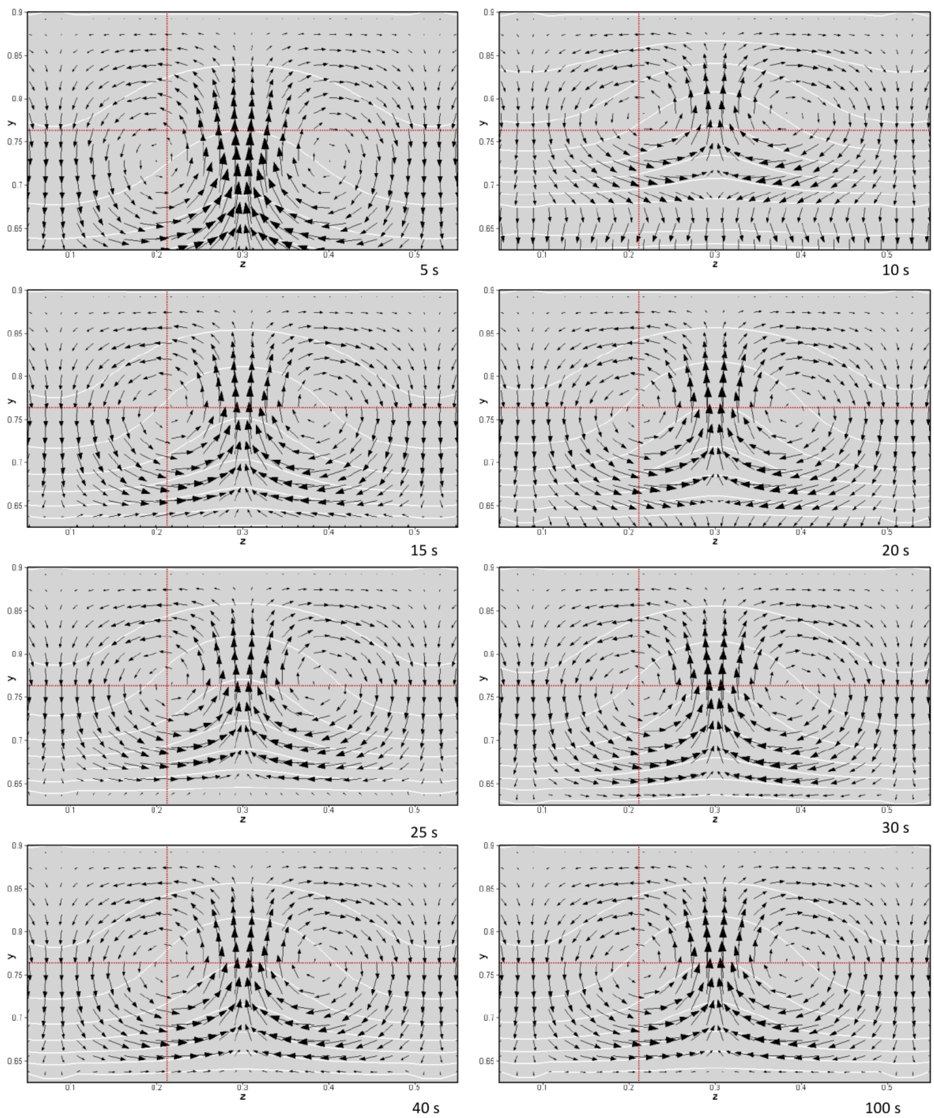

Figure 5: Velocity vectors in the Taylor-Couette without translation in different instants of time.

It can been seen an equilibrium in the system inner cylinder-springs near the 75 seconds. Until this instant the inner cylinder experiments translation and the Taylor vortices evidence variable thickness as the annulus between the cylinders variate. Visualizing the vectors in a cut in the $x = 0.5m$ plane in the times 5, 10, 15, 20, 25, 30, 40 and 100 it can been seen thick vortices in the instant $t = 5s$ and thin ones in the instant $t = 10s$, shown in Fig. 5. This variation is less accentuated in the subsequent time steps, as the system go to the rest position.

The variation of the velocities with the inner cylinder position can be visualized in Fig. 6. In the Fig. 6a we have the variations in u velocity component started at the instant $t = 15s$ and going to steady state. Fig. 6b show the steady state for the v velocity in $x = 0.5$ and $y = 0.4m$ and instant $t = 100s$, in comparison with the classical Taylor-Couette flow.

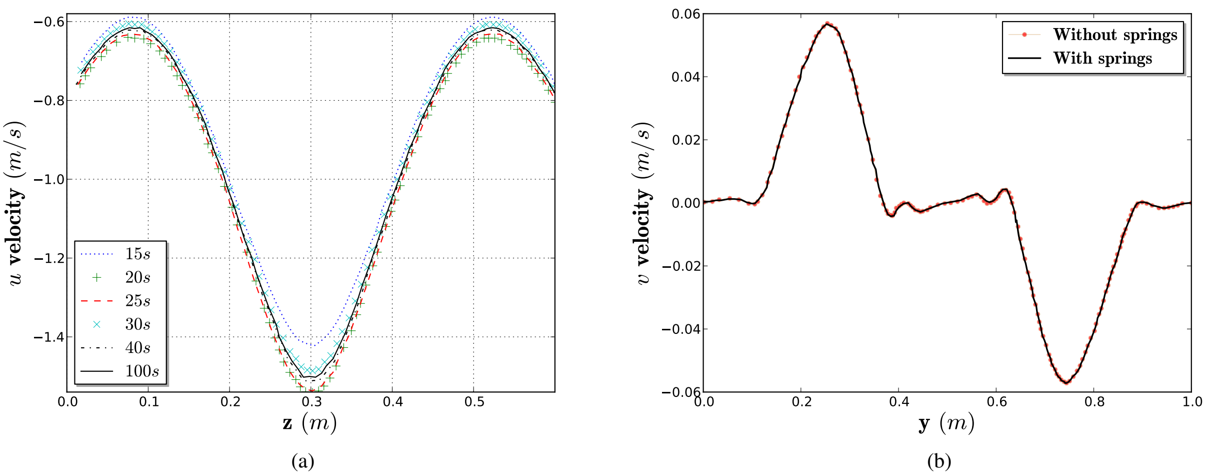

Figure 6: Velocity profiles in the fluid-structure interaction case; (a) u velocity in the center of the Taylor vortices and (b) steady state v velocity comparison with the classical Taylor-Couette flow.

5. CONCLUSIONS

The immersed boundary method has suited very well to represent the cylinders in the simulations performed in the present work and allowing, among other things, the natural development of Taylor vortices. As could been observed in the presented results, the periodic boundary conditions in the z direction allowed the classical Taylor-Couette and the fluid-structure interaction version, even with the short aspect ratio utilized. The developed application has proven to be very capable in the simulation of the proposed tests and the presented results has demonstrated a qualitative agreement with the physics of the analyzed fluid-structure interaction problem.

6. ACKNOWLEDGEMENTS

The authors like to tank the Support and Research of Minas Gerais State (FAPEMIG), the CNPq and the Research Center of PETROBRAS (CENPES) for the financial support.

7. REFERENCES

Davey, A., 1962. "The growth of taylor vortices in flow betwenn rotating cylinders". *J. Fluid Mech.*, Vol. 14, pp. 336–338.

DiPrima, R.C., 1960. "The stability of a viscous fluid between rotating cylinders with an axial flow". *J. Fluid Mech.*, Vol. 9, pp. 621–631.

Eagles, P., 1977. "On the stability of slowly varying flow between concentric cylinders". *Proc. R. Soc. Lond.*, pp. 209–224. A 355.

Escudier, M.P. and Gouldson, I.W., 1995. "Concentric annular flow with centerbody rotation of a newtonian and shear-thinning liquid". *International Journal of Heat and Fluid Flow*, Vol. 16, pp. 156–162.

Escudier, M.P., Oliveira, P.J. and Pinho, F.T., 2002a. "Fully developed laminar flow of purely viscous non-newtonian liquids through annuli, including the effects of eccentricity and inner-cylinder rotation". *International Journal of Heat and Fluid Flow*, Vol. 23, pp. 52–73.

Escudier, M.P., Oliveira, P.J., Pinho, F.T. and Smith, S., 2002b. "Fully developed laminar flow of purely non-newtonian liquids through annuli: Comparison of numerical calculations with experiments". *Experiments in Fluids*, Vol. 32, pp. 101–111.

Hwang, J.Y. and Yang, K.S., 2004. "Numerical study of taylor-couette flow with an axial flow". *Computer & Fluids*, Vol. 33, pp. 97–118.

Kaye, J. and Elgar, E.C., 1957. "Modes of adiabatic and diabatic fluid in an annulus with an inner rotating cylinders". *Transactions of the ASME*, Vol. 80, pp. 753–765.

Kim, J. and Moin, P., 1985. "Application of a fractional-step method to incompressible navier-stokes equations". *Journal of Computational Physics*, Vol. 59, No. 2, pp. 308 – 323. ISSN 0021-9991. doi:DOI: 10.1016/0021-9991(85)90148-2.

Lima e Silva, A.L.F., da Silveira Neto, A. and Damasceno, J.J.R., 2003. "Numerical simulation of two-dimensional flows over a circular circular cylinder using the immersed boundary method". Vol. 189, pp. 351–370.

Lockett, T.J., Richardson, S.M. and Worraker, W.J., 1992. "The stability of inelastic non-newtonian fluids in couette flow between concentric cylinders: a finite-element study". *Journal of non-Newtonian Fluid Mechanics*, Vol. 43, pp. 165–177.

Ludweig, H., 1964. "Experimentelle nachprüfung der stabilitätstheorien für reibungsfreie strömungen mit schraubenlienienformingen strömlinien". *Zeitschrift für Flugwissenchaften*, Vol. 12, pp. 304–309.

Lueptow, R.M., Docter, A. and Kyungyoon, M., 1992. "Stability of axial flow in an annulus with a rotating inner cylinder". *Physics of Fluid A*, Vol. 4, pp. 2446–2455.

Padilla, E.L.M., 2004. *Simulação de grandes escalas da transição à turbulência em sistemas rotativos com transferência de calor*. Ph.D. thesis, UFU, Universidade Federal de Uberlândia, Uberlândia, MG, Brasil.

Patankar, S.V., 1980. *Numerical Heat transfer and Fluid Flow*. Hemisphere Publishing Corporation, 1st edition.

Peskin, C.S., 1977. "Numerical analysis of the blood flow in the heart". Vol. 25, p. 220.

Stone, H.L., 1968. "Iterative solution of implicit approximations of multidimensional partial differential equations". *SIAM Journal on Numerical Analysis*, Vol. 5, pp. 530 – 558.

Taylor, G.I., 1923. "Stability of a viscous liquid contained between two rotary cylinders". *Philosophical Transactions of the Royal Society of London*, pp. 289–343. A 223.

Wang, Z., Fan, J. and Luo, K., 2007. "Combined multi-direct forcing and immersed boundary method for simulating flows with moving particles". *International Journal of Multiphase Flow*, Vol. 34, pp. 283–302.

Weisberg, A.B., Kevrekidis, I.G. and Smits, A.J., 1997. "Delaying transition in taylor-couette flow with axial motion of the inner cylinder". *J. Fluid Mech.*, Vol. 11, pp. 1–12.

Wereley, S.T. and Lueptow, R.M., 1998. "Spatio-temporal character of non-wavy and wavy taylor-couette flow". *J. Fluid Mech*, Vol. 18, pp. 1–9.

Wereley, S.T. and Lueptow, R.M., 1999. "Velocity field for taylor-couette flow with an axial flow". *Phys Fluids*, Vol. 11, pp. 3637–49.

8. Responsibility notice

The author(s) is (are) the only responsible for the printed material included in this paper.