3D NUMERICAL SOLVER IN CYLINDRICAL COORDINATES FOR INCOMPRESSIBLE FLOWS: VALIDATION PROCESS

Elie Luis Martínez Padilla
Marcos Antonio de Souza Lourenço
Aristeu da Silveira Neto
Federal University of Uberlândia,
School of Mechanical Engineering,
38400-902, Uberlândia, MG, Brazil

Abstract. *The development of an application to solve the unsteady, three-dimensional Navier-Stokes equations in the cylindrical system of coordinates is presented, aimed to analyze isothermal, one-phase, incompressible fluid flows. The solver uses a staggered finite volume method with second order in space and time discretizations (the plan, in the future, is to use the large eddy simulation). The consistence and stability were verified using a code validation in two parts: first, only Poiseuille annular and Couette annular were compared with its analytical solutions. In the second, Taylor-Couette flow is considered (i.e., with the presence of three-dimensional structures, Taylor Vortices), making use of experimental and numerical data in the comparison. The numerical simulation was made for many values of Taylor and Reynolds numbers and the quantitative and qualitative approaches allowed the evaluation of the code performance.*

Keywords*: 3D solver, Incompressible flow, Validation*

1. INTRODUCTION

According to Taylor (1923), who studied experimentally and analytically flows between rotating concentric cylinders, for small gaps between them (compared with the radii of the internal cylinder), the problem simplifies and becomes dependent on the Taylor number. When this parameter increases above the critical value, counter-rotating axisymmetric vortices of toroidal shape, also referred to as Taylor-Couette instabilities, arises in the flow. Later, many other researches had been carried out (Davey, 1962; Eagles, 1971; Wereley and Lueptow, 1994) due to the great number of applications in several areas of engineering. The Taylor-Couette flow with superposed axial flow, also has been object of many investigations, for same reasons previously mentioned. In particular, the Taylor-Couette flow with superposed Poiseuille flow (Kaye and Elgar, 1957; DiPrima, 1960; Lueptow *et al.*, 1992) as well as superposed Couette flow (Ludweig, 1964; Weisberg *et al*., 1992; Hwang and Yang, 2003), are of interest in the well drilling engineering in the oil and gas production.

In real applications, the types of flows found in well drilling processes are much more complex than the flows presented, because there are additional problems, for instance: eccentric movement determined by the interaction of internal and external flows (related to internal channel) and fluids with changeable viscosity due the stress rate (non-Newtonian fluids). Considering, about simplified form of additional problems before mentioned, some works 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*. (2002) and Escudier *et al*. (2002-b) for fixed eccentric configurations and non-Newtonian fluid.

Due this 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 (2003) used the finite volume method with second order in space and third order in time discretizations, in cylindrical coordinates. In a computational application, the determined velocities are not averaged values, as experimentally by particle image velocimetry (PIV), whose problem (of *aliasing*) can be solved using mesh refinement.

In the present work the finite volume method was utilized in the discretization using a cylindrical system of coordinates. Global second order was utilized: spatially, with the central difference scheme and using 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 for pressure correction, many methods are available, as TDMA-SOR (Three Diagonal Matrix Algorithm, combined with Successive Over Relaxation) and ICCG (Conjugate Gradient method, preconditioned using Incomplete Cholesky decomposition), Multigrid, and some combinations of these, that account for stability and convergence rates of the solver. For this work was applied the Strongly Implicit Procedure (SIP), of Stone (1968), that join together simplicity, good stability and convergence rate.

2. MATHEMATICAL MODEL

The fluid confined is isothermal and incompressible, with constant properties. The computational modeling requires solving three-dimensional flows equations, or either solving the Navier-Stokes equations. These equations in dimensional form and cartesian coordinates are presented as follows:

$$\nabla.\vec{u} = 0 \,, \tag{1}$$

$$\frac{\partial \vec{u}}{\partial t} + \nabla.(\vec{u}\vec{u}) = -\frac{1}{\rho}\nabla \vec{p} + \nabla.\left[\nu(\nabla\vec{u} + \nabla\vec{u}^T)\right], \tag{2}$$

where the velocity vector $\vec{u}$ has components u, v, w in r, θ, z (radial, tangential and axial) directions, respectively, p is the pressure field, ρ is the density and ν is the cinematic viscosity. Introducing a scale L for length, ν/L for velocity, ν/L^2 for time and $\rho\nu^2/L^2$ for pressure, the Eqs. (1-2) are transformed in dimensionless form. The refereed equation are solved with the following boundary conditions: in radial direction, no sleeping for v and w and impenetrability for u ; in tangential and axial directions, periodicity for all components of velocity.

3. NUMERICAL PROCEDURE

In order to perform the discretization of the equations, the finite volume method (Patankar, 1977) was employed on staggered grid, having second order schemes in space and time: central differencing an Adams-Brashforth schemes, respectively. The pressure velocity coupling method was done using the fractional step (Kim and Moin, 1985), where the steps named predictor and corrector are used. The pressure correction is evaluated by solving the Poisson equation using a strongly implicit procedure method, as proposed by Stone (1968).

The time step is evaluated following the CFL stability criteria. Moreover, and non-uniform mesh (concentrated near the walls) is used, with 5% of increment/decrement ratio.

4. PRELIMINARY TESTS

The isothermal, annular, Couette and Poiseuille flows were utilized as preliminary test cases, both having analytical solutions. The annular Couette flow is the steady state flow occurring between two concentric rotating cylinders, owning different velocities of rotation. In this case, a zero velocity of rotation is considered in the extern cylinder of radius R_e and the internal cylinder, of radius R_i, have rotate at velocity ω, as depicted in Fig. 1. The values of the Reynolds number, or Ta , characteristics for this flow are lower than the critical Reynolds number Ta_c (after this point the Taylor instabilities arise). The dimensionless parameters defining the geometrical configuration are the radius and aspect ratios, $\eta = R_e/R_i$ and $\Gamma = C_a/E$ respectively, where C_a is the axial length of the cylinders and E is the difference between the internal and external cylinders radius. Lastly, the Taylor number is defined as $Ta = \omega R_i E/\nu$ (Wereley e Lueptow, 1999).

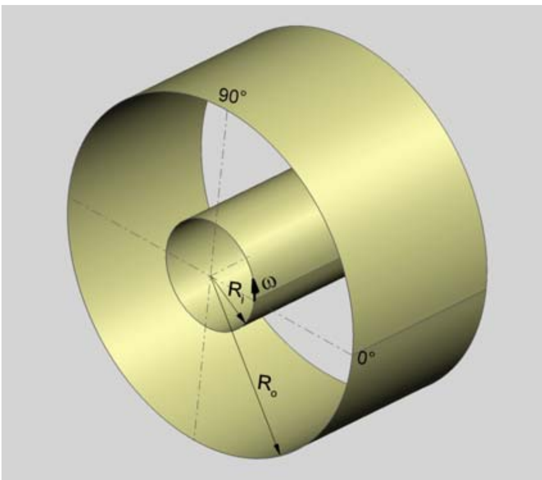


Figure 1. Schematic drawing of the domain between two cylinders. Only the internal cylinder has non-zero velocity of rotation.

The geometrical configuration for the simulation is given by $\eta = 1.1325$ and $\Gamma = 1$, two values of the Taylor number and mesh dimensions of 10x3x60 in the radial, tangential e axial directions, respectively. For the value of η considered, the critical Taylor number is $Ta_c = 114$, in conformity to Anderek et al. (1985) and Kupferman (1998). The comparisons between analytical (White, 1974l; Bird *et al*., 2002) and numerical solutions, for the radial distributions of the tangential velocity are presented in Fig. 2, for $Ta =$ 50 and 100. It's possible to observe an excellent agreement with the analytical solution.

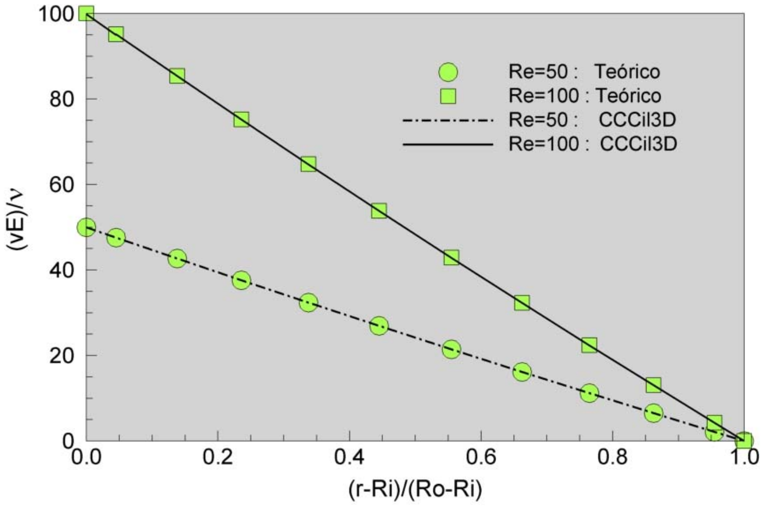


Figure 2. Radial distribution of the tangential velocity for annular Couette flow.

On the other hand, the annular Poiseuille flow is developed due a pressure difference in the axial direction in the annulus between the cylinders. Thus, pressure is imposed on the inlet (P_e) and in the outlet (P_s) of the annulus. The same parameters of the last problem are utilized, beside the one governing the flow, the Reynolds number, defined as $Re = w_m E/\nu$, where w_m is the mean velocity in the axial direction (based on the imposed pressure difference).

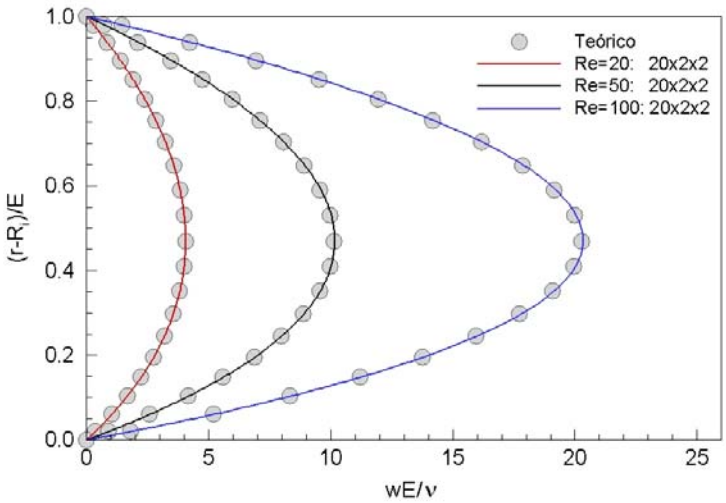


Figure 3. Comparison of the axial velocity component for annular Poiseuille flow.

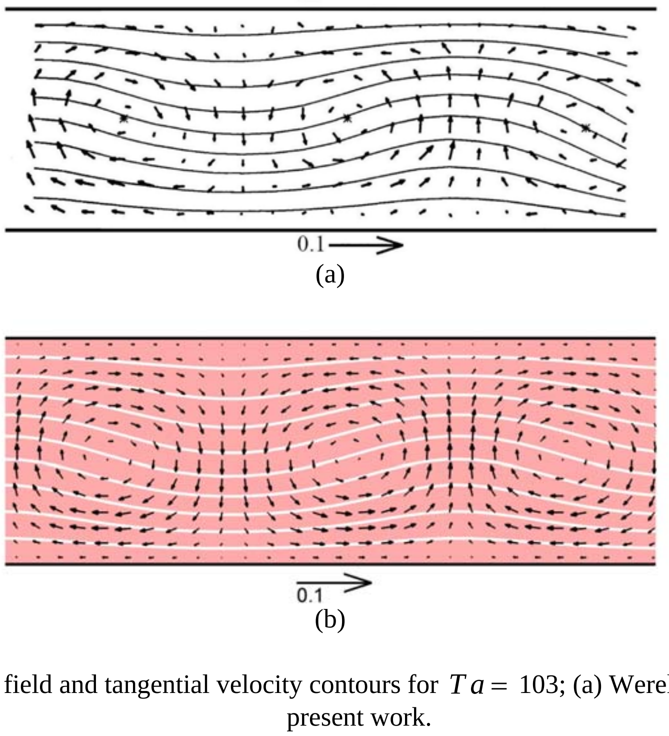
For this simulation, three values of the Reynolds number, $Re = 20$, 50 and 100, and the parameters $\eta = 2$ and $\Gamma = 1$ were utilized with a mesh of dimensions 20x2x2. The determined axial velocity profiles, depicted in Fig. 3, agreed very well in accordance with the analytical results (White,1974, Bird et al., 2002), with differences smaller than 0,4%.

4. RESULTS

Sequentially to the preliminary tests, the Taylor-Couette flow was utilized as a more serious test case to the validation of the computational code. The problem configuration is the same of the annular Couette flow (see Fig. 1), with a higher internal cylinder rotational velocity and the Taylor number above the critical value. The simulations has been realized with the following parameters: $\eta = 1.205$ (Wereley e Lueptow, 1999) e Γ = 6, Taylor numbers over the range $Ta =$ 103 e 124 and different mesh densities of refinement. As mentioned by Wereley e Lueptow (1999), for the defined value of η, the Taylor critical value is about 102, changing to between 124 and 131, when the Taylor vortices becomes unstable.

The determined results are presented as: a) r, z planes in azimutal position $\theta = 3{,}0$ *rad*, showing dimensionless tangential velocity vectors $v/(\omega R_i)$ (Figs. 4 e 6) and b) dimensionless profiles of axial velocity $w/(\omega R_i)$, in radial direction and of radial velocity $u/(\omega R_i)$ in axial direction z/E, passing through the core of a positive vortice, using the right hand rule (Figs. 5, 7 e 8). The references for comparisons are: the experimental work of Wereley and Lueptow (1999), based on particle image velocimetry (PIV), and the numerical work of Hwang and Yang (2004), using the finite volume method and Γ = 32.

The velocity vector field and non-dimensional isocontours of tangential velocity (white contours incrementing in 0.1) for $Ta =$ 103, are presented in Fig. 4, where the bottom and upper straight lines represent the intern and extern cylinders surfaces respectively. In Fig. 4(a) the experimental results of Wereley e Lueptow (1999) are shown and compared with the ones determined in this work depicted in Fig. 4(b), for a mesh of 14x20x60 volumes, in radial tangential and axial direction respectively. In the whole domain, the flow is composed by three Taylor vortices pairs, with a characteristic wave length of $2E$, and only one pair and half of the first pair (left) are shown in Fig. 4. Some performed experimental characteristics are accurately reproduced: the radial flow (positive) between the vortices pairs is more intense than the radial flow (negative) between the adjacent vortices pairs, the radial flow direction alters the tangential velocity field, producing peaks and valleys and the distance between the cores of a vortices pair is smaller than the distance to the adjacent vortices core.

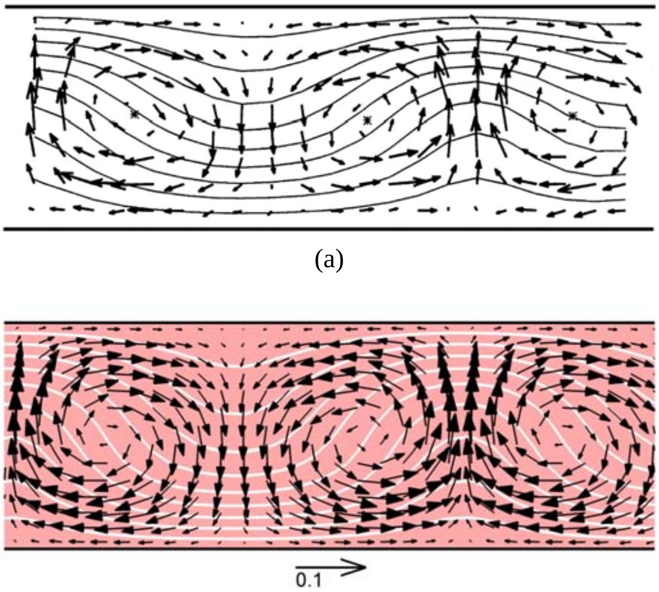


(a)

(b)

Figure 4. Velocity vector field and tangential velocity contours for $Ta =$ 103; (a) Wereley e Lueptow (1998), (b) present work.

Incrementing the Taylor number to Ta = 124 (see Fig. 5), near the second critical value, the counter-rotating vortices become more strong, according Davey (1962), Wereley and Lueptow (1994), and others. Consequently, the velocity vector field is greater in modulus. On the other hand, the radial flow between the vortices pairs and the adjacent vortices is more intense, although deformed contours are derived from the tangential velocity field, when compared with the fields in Fig. 3. The Fig. 5(a) correspond to the experimental result of Wereley and Lueptow (1998). The result obtained using a 14x40x60 mesh (Fig. 5b) is similar to the reference data and can also be observed a little pronunciated tangential velocity in the contours of Wereley e Lueptow (1998) when compared with the numerically determined.



(a)

(b)

Figure 5. Velocity vector field and tangential velocity contours for $Ta =$ 124; (a) Wereley e Lueptow (1998) for, (b) present work.

The radial velocity profiles through axial direction (Fig. 6) show suitable contours to the characteristics velocity fields, formed by positive-negative cell pairs (Padilla *et al.*, 2007; Padilla e Silveira Neto, 2008; Kupferman, 1998). In the above mentioned figure is presented a comparison of the profiles using three different meshes, refined much more on the radial direction, as well as in the comparison to the experimental data of Wereley e Lueptow (1998). The profiles using 14x20x60 and 16x20x60 meshes are similar, with small difference in the peaks. Nevertheless, the profile using a 12x20x60 mesh present a little higher value compared to profiles of the other meshes. Although the averaged tendency of the experimental data of Wereley e Lueptow (1998) can be good represented, is hard to accurately measure the differences.

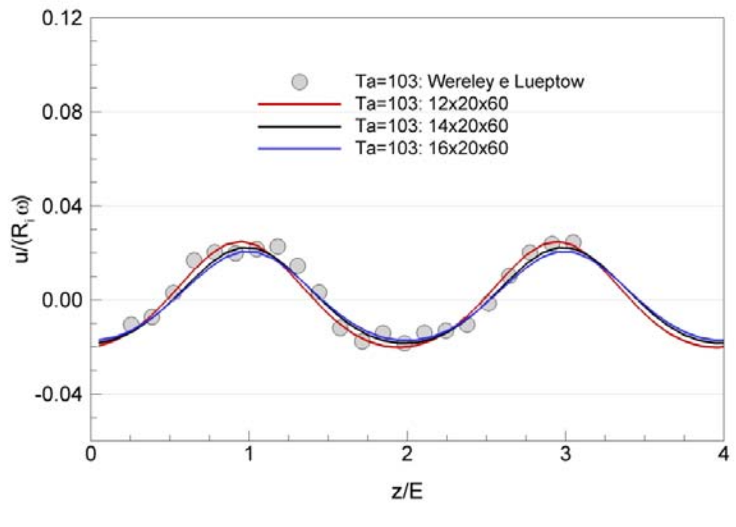


Figure 6. Radial velocity distribution in axial direction in the position $r = (R_o + R_i)/2E$. Comparison with experimental data, $Ta =$ 103.

For the Taylor number of 124, the meshes used had a refinement between 10 and 14 volumes in the radial direction, and 20 and 120 volumes in the tangential direction. The results with few volumes, on the other hand, have minor accuracy and due to the nature of the structures present in the flow is necessary to use a minimal number of volumes in each direction to assert a good solution. For the present configuration, this can be obtained using at least 14 volumes in radial direction and 40 volumes in tangential direction.

In Figs. 7 and 8, the components of radial e axial velocities respectively, results using 12x120x60 and 14x40x60 meshes are compared with the numeric and experimental references. Both profiles, as the results of Hwang and Yang (2004), present values of high magnitude than the ones of Wereley e Lueptow (1998). The difference evidenced to the data of Hwang e Yang (2004), in the radial velocity component (Fig. 7), can be due to the Taylor value used of $Ta =$ 123 and, possibly because to the use of a coarse mesh in the axial direction (approximately 8 volumes for each axial sub-domain of length E). At last, according the experimental data of Wereley and Lueptow (1998), Hwang and Yang (2004) have obtained higher magnitudes for the peaks of each velocity component, due to the fact of the maximum velocity values are lightly reduced as the values on the points used to calculate the averaged values are smaller than the peak value (the cross correlation that PIV methodology is based). It must be considered that the associated errors with other experiments, and the authors reported an error of 4% in determination of the Taylor number.

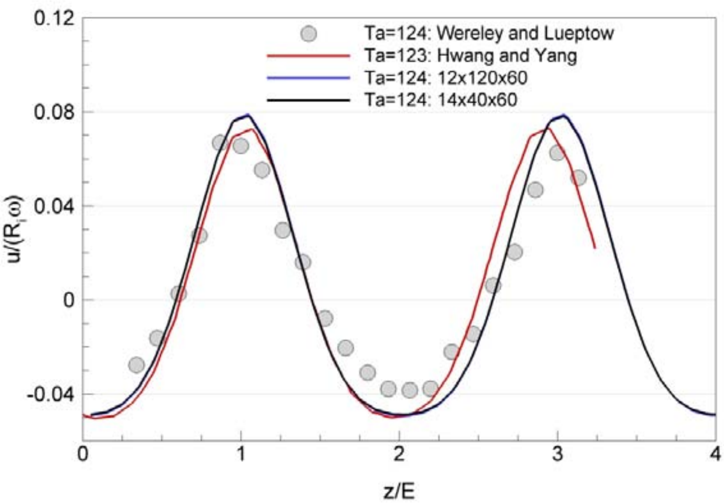


Figure 7. Radial velocity distribution in axial direction in the position $r = (R_o + R_i)/2E$. Comparison with experimental data, $Ta =$ 124.

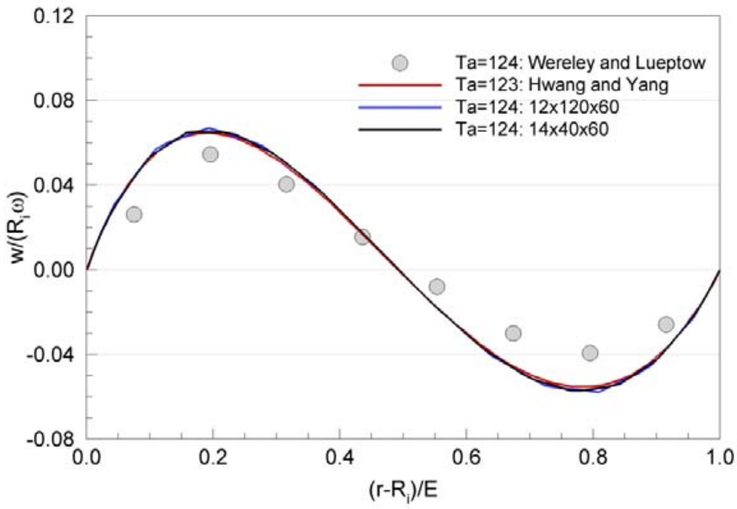


Figure 8. Comparison with experimental data of the axial velocity distribution in radial direction, passing through the core of the central vortice (positive), $Ta =$ 124.

5. CONCLUSIONS

The validation process of a computational solver for the newtonian and isothermal fluid flow was realized in the present paper. The numerical code uses the cylindrical system of coordinates and is possible to solve one, two and three-dimensional problems. A few minor errors were obtained reproducing the one-dimensional annular Poiseuille and Couette flows. The three-dimensional Taylor-Couette flow was be characterized using many mesh refinements configurations, analyzing the influence over the characteristic structure, the Taylor vortices, and the velocity field. The satisfactory qualitative and quantitative comparisons show, evidently, the reliability of the obtained results and the validation of the developed numerical code.

6. ACKNOWLEDGEMENTS

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

7. REFERENCES

Andereck, C.D., Lui, S.S. and Swinney, H.L., 1986, "Flow Regimes in an Circular Couette System with Indepently Rotating Cylinders", J. Fluid Mech., vol. 164, pp. 155-183.

Escudier, M.P. Oliveira, P.J. and Pinho, F.T., 2002-a, "Fully Developed Laminar Flow of Purely Non-Newtonian Liquids Through Annuli, Including the Effects of Eccentricity and Inner-Cylinder Rotation", Int. J. Heat and Fluid Flow, vol. 23, pp. 52-73.

Escudier, M.P. Oliveira, P.J. Pinho, F.T. and Smith, S., 2002-b, "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.

Eagles, P.M.., 1971, "On the Stability of Taylor Vortices by Fifth-Order Ampliitude Expansion", J. Fluid Mech., vol. 49, pp. 529-550.

DiPrima, R.C., 1960, "The Stability of a Viscous Fluid Betwenn Rotating Cylinders with an Axial Flow", J. Fluid Mech., vol. 9, pp. 621-631.

Davey,A., 1962, "The Growth of Taylor Vortices in Flow Betwenn Rotating Cylinders", J. Fluid Mech., vol. 14, pp. 336-338.

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", J. Comp. Phys., 59, pp. 308-323.

Kupferman, R., 1988, "A Numerical Study of the Axisymmetric Couette-Taylor Problem Using a Fast High-Resolution Second-Order Central Scheme", SIAM J. Sci. Comp., vol. 20(3), pp. 858-877.

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(11), pp. 2446-2455.

Ludweig, H., 1964, "Experimentelle Nachprüfung der Stabilitätstheorien für Reibungsfreie Strömungen mit Schraubenlienienformingen Strömlinien", Zeitschrift f6ur Flugwissenchaften, vol. 12, pp. 304-309.

Patankar, S.V., 1980, "Numerical Heat transfer and Fluid Flow", Hemisphere Publishing Corporation, New York.

Stone, H.L., 1968, "Iterative Solution of Implicit Approximations of Multidimensional Partial Differential Equations", SIAMJ Numer. Anal., vol. 5, pp. 530-558.

Taylor, G.I, 1923, "Stability of a Viscous Liquid Countained Betwenn Two Rotatin Cylinders", Phil. Trans R Soc., vol. A223, pp. 289-343.

Wereley, S.T. and Lueptow, R.M., 1994, "Azimutal Velocity in Supercritical Circular Couette Flow", Exp. Fluids, 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(12), pp. 3637-3649.

8. RESPONSIBILITY NOTICE

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