SENRA Academic Publishers, British Columbia
Vol. 7, No. 2, pp. 2425-2430, June 2013
Online ISSN: 1920-3853; Print ISSN: 1715-9997

Canadian Journal of pure&applied sciences

TURBULENT MODELING FOR NON-NEWTONIAN FLUID IN AN ECCENTRIC ANNULUS

J Khorshidi[1], S Niazi[2], *H Davari[3], M Mahmeli Abyaneh[4] and A Mahmoodzadeh[5]
[1,2]Department of Mechanical Engineering, Hormozgan University, Bandarabbas
[3,4,5]Department of Mechanical Engineering, Roudan Branch, Islamic Azad University, Roudan, Iran

ABSTRACT

The aim of this study is the numerical simulation of non-Newtonian fluid in an eccentric annulus with rotation of the inner cylinder. To evaluate the modeling accuracy, the computational domain has been defined such that the results of this study can be compared with experimental results of Nouri *et al*. (1997). At first the governing equations were selected according to the follow physics. Then by selecting the appropriate network, the physical equations were solved using CFD technique and finite volume method. The turbulent flow modeling has been performed using K-ε, K-ω and K-εRNG methods. The obtained results of the study showed that the K-ω method is compatible with experimental data of Nouri *et al*. (1997) and thus is a more appropriate method for numerical analysis of this type of flow.

Keywords: CFD, turbulent modeling, non-Newtonian fluid, finite volume.

INTRODUCTION

Expect in areas very close to the wall, in turbulent flows, flow layer shape is not easily detectable due to the severe blending processes, and the fluid molecules follow a clear path. In other words, turbulent flow is a type of fluid flow in which the fluid is under fluctuation flow. In a turbulent flow, velocity in any point is always under fluctuations and variations, both in size and motion direction, so that detecting position of each particle inside the follow field and also at any time is difficult. The same permanent and non-specific fluctuations in velocity size can be observed in the size of pressure, temperature and density of each point. Generally, turbulent flows have the following features:

- Spatial and temporal irregularities
- Spatial and temporal continuous spectrum
- High Reynolds
- Increased dissipation of energy and momentum
- Being three-dimensional
- Being periodic

Given the unpredictable nature of turbulent flows, therefore, experimental relations are of special importance for their modeling. Regarding experimental simulation of turbulent flow, many studies have been conducted that differences between these studies depend on fluid type, flow geometry type, range of velocity: pressure and working temperature. Brighton and Jons (1964) experimentally investigated turbulent flow of Newtonian fluid in a fixed annulus. The examined fluid was air which is a compressible fluid. Escudier *et al*. (1994) and Escudier and Gouldson, (1995) examined laminar and turbulent flows of Newtonian and non-Newtonian fluid in a fixed annulus. In the following Nouri *et al*. (1993) fully investigated turbulent flows in an annulus. The external tube is coanstant and the internal tube is rotating. Both a Newtonian and a non-Newtonian fluid have been considered in the performed experiments (Nouri and Whitelaw, 1997).

However, experimental modeling of turbulent flow in real scale is a very costly and timely process and if by applying a suitable numerical model, the flow can be analyzed, certainly there will be enormous savings both in time and cost.

For example, in this study we seek to perform numerical modeling of fluid flow in an eccentric annulus, and the obvious example is in oil industry and drilling sector.

After that drilling bore which is often eccentric with the well, began drilling, mud of drilling which is considered a non-Newtonian fluid comes out of the bore and the well diameter. However, after drilling and in order to strengthen the well and provide an insulating membrane around the well, a metal shell with a diameter less than the well diameter is installed in the well with a distance from the well and eccentric, and mud of drilling is pumped out using a fluid pump inside the well and then cement slurry is pumped for the mentioned purposes.

This industrial example can be simulated with a numerical model. The aim of this study is to examine various numerical methods for modeling turbulent flow of non-

*Corresponding author email: hasan.davari163@gmail.com

Newtonian fluid in an eccentric annulus with rotation of inner cylinder. Finally, the appropriate model which has the most conformity with experimental data will be introduced as the appropriate model.

Computational domain

Computational domain has been considered as a periodic cylinder. The diameter of outer cylinder is 40.1 mm and the inner cylinder is 20mm (Fig. 1). The surface of cylinders is assumed as the wall, and the two ends as periodic boundary conditions and the space between the two cylinders as non-Newtonian fluid. Hexagonal cells the organization is used for networking. In figure 2 you can see Line coordinates of profile investigation of flow variables:

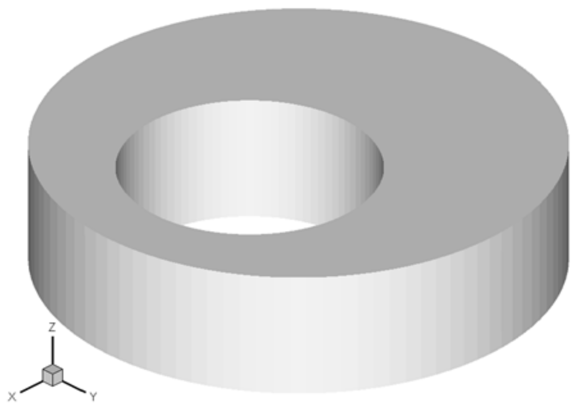

Fig. 1. Geometry of the solution domain.

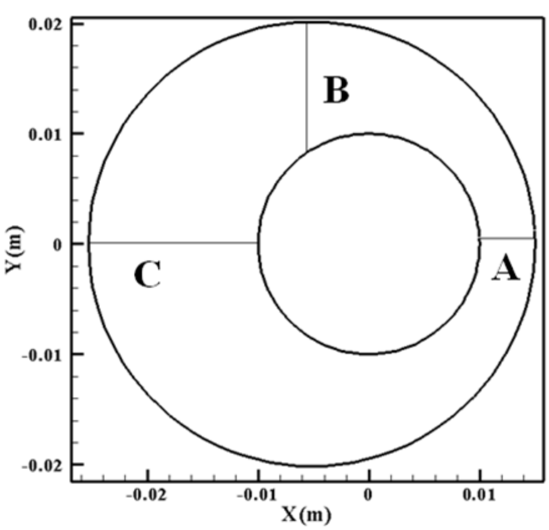

Fig. 2. Line coordinates of profile investigation of flow variables.

Computational domain was chosen such that results of this numerical simulation can be compared with experimental data of Nouri and Whitelaw (1997), in figures 3 and 4 you can see a look of the experimental device used in the study.

The governing equations of non-Newtonian fluid

The equations governing flow field resulting from motion of incompressible non-Newtonian fluid with constant density are as follows based on Navier Stocks equations.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_i}\partial u_i \partial u_j = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \rho g_i \qquad (2)$$

Viscosity is defined based on the following exponential rule.

$$\mu = K \times \gamma^{(n-1)} \qquad (3)$$

That γ shear stress rate and K concentration rate and n is the index of fluid behavior such that when n equals 1, the fluid is Newtonian and when n is smaller than 1, the fluid is non-Newtonian.

As seen in the above equation, besides non-Newtonian viscosity, a new coefficient has been added to the equation which is unknown and it's a turbulent viscosity coefficient that must be determined so that the problem would be closed and could be solved. Models based on turbulent viscosity are classified into various methods based on how the turbulent viscosity coefficient is calculated. Zero equation models, one equation models and two equation models are common models based on turbulent viscosity hypothesis. Unfortunately, there is no turbulent model that can be used for all conditions and different problems; and choosing the turbulent model depends on considerations such as flow physics, the obtained experiences of simulation for particular problems, the required accuracy, the power of computational resources (power of computer) and the available time for performing computations. Of course, capabilities and limitations of all models and conditions must be considered for selecting an appropriate turbulent model. In the following some of the two equation models used in the present study will be studied.

The simplest turbulent models that are relatively perfect are two equation models; because solving two transmission equations separately causes that turbulent velocity and the character length are determined separately. The standard K-ε model is in this group of turbulent models and is considered as one of the most powerful turbulent models for engineering problems. Being powerful, economic calculations and acceptable in a wide range of turbulent flows cause the popularity of this model in industrial and heat transfer issues. K-ε model is a semi-experimental model and its equations have been created based on experimental observations and phenomenological considerations.

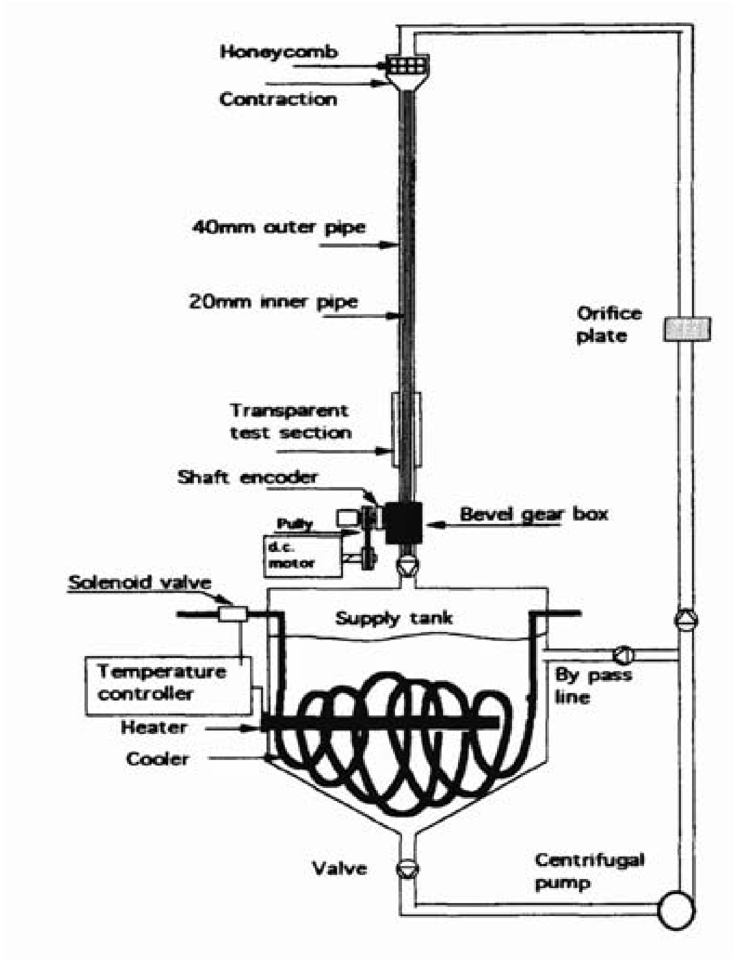

Fig. 3. Experimental setup in Nouri and Whitelaw (1997) .

Relations of turbulent modelK-ε are as follows:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\delta_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k-\varepsilon \tag{4}$$

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\delta_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}G_k-C_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{5}$$

And the vortex viscosity is expressed by the following relations:

$$\mu_t=\rho C_M\frac{k^2}{\varepsilon} \tag{6}$$

Constant coefficients of the model are expressed as follows:

$$C_\mu = 0.09 \tag{7}$$

$$C_{\varepsilon 1} = 1.92 \tag{8}$$

$$\delta_k = 1 \tag{9}$$

$$\delta = 1.3 \tag{10}$$

which K is kinetic energy of turbulent flow and e is dissipation rate of turbulent.

K-ε RNG model

This model has been obtained using statistical methods and is similar to the standard K-ε model. RNG model has higher accuracy in rapidly strained flows due to having additional terms in solving ε equation. Rotation effects on turbulence have been included in the RNG model which causes its higher accuracy in vortex flows. In the RNG model, an analytical formula is used for calculating a turbulent parentel number; while in the standard K-ε model, this number is entered the software by the user, and thus in all stages, it has a constant value. If the standard K-ε model is a model for flows with high Reynolds numbers, in the RNG mode, to determine the effect of viscosity on the flow an analytical differential formula has been used to model flows with low Reynolds number. Of course, the ability of the viscosity effect on the flow depends on the way of performing calculations

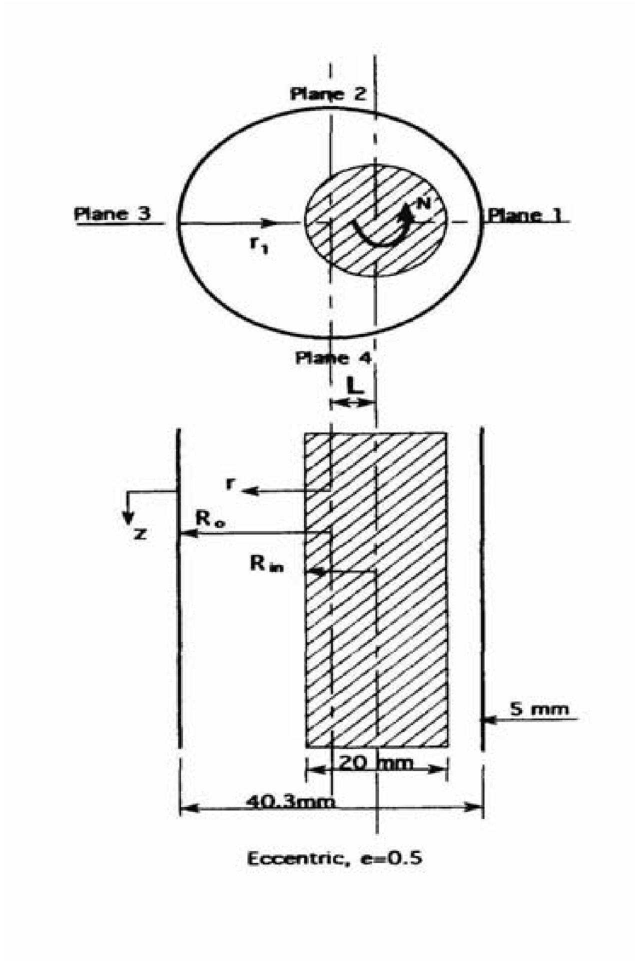

Fig. 4. Test section and co-ordinate system in Nouri and Whitelaw (1997).

and behavior near the wall area. These advantages have caused that RNG model has better accuracy and reliability in a broader range of flows than the standard K-ε model.
K-ω turbulent model

In this model by writing two transmission equations (one for turbulent kinetic energy and the other for dissipation rate of turbulent kinetic energy) and solving them, a longitudinal scale and a velocity scale are obtained that vortex viscosity is created from these values and using dimensional analysis. As K-ε model, k equation is solved for determining velocity scale, and ω equation is solved for determining longitudinal scale. Ω which is usually called specific dissipation has the following relation with K and ε:

$$\omega = \frac{\varepsilon}{\beta_* k} \quad (11)$$

Given the above ω definition, It is proved that applying boundary condition for ω equation is simpler that ε equation. High Re k-ω turbulent model equation is often called the standard k-ω model is as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j}\left[(\mu + \sigma_* \mu_t)\frac{\partial k}{\partial x_j}\right] + P_k - \beta_* \rho \omega k \quad (12)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho u_j \omega) = \frac{\partial}{\partial x_j}\left[(\mu + \sigma \mu_t)\frac{\partial \omega}{\partial x_j}\right] + \gamma \frac{\omega}{k} P_k - \beta \rho \omega^2 \quad (13)$$

In which P_k is:

$$P_k = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{(\partial x_j)} \quad (14)$$

And μ_f in the model is calculated as the following:

$$\mu_t = \gamma_* \frac{\rho k}{\omega} \quad (15)$$

the used constants in the model are as follows:

$$\beta = \frac{3}{40}, \beta_* = \frac{9}{100}, \gamma = \frac{5}{9}, \gamma_* = 1, \sigma = 0.5, \sigma_* = 0.5$$

Assumptions and equation solving method

Finite volume method is used for solving physical equations (energy momentum continuity,…). In the finite volume method, physical equations are used in the integral form. There are two pressure-base and density-base solvers for solving integral equations. Under this condition, the flow is incompressible and the heat transfer is also ignored. Given the incompressibility of the fluid flow, density variations due to the variations in pressure and temperature, do not affect the flow equations; therefore, solution type is selected pressure-base solution. In order to compare the obtained results of this study with the experimental data, characteristics of non-Newtonian fluid is defined based on the study of Nouri and Whitelaw (1997). Concentration index is defined 0.04 and exponential index 0.75. Walls have non-sliding condition. In periodic boundary condition, the constant mass flux for non-Newtonian fluid flow was considered 1.3, 2.6 and 5.2kg/s, and also the velocity of the inner cylinder rotation is considered 0, 300 and 600 rpm based on the prionary guess of periodic boundary. Then repetition is continued until the convergence of residues.

RESULTS AND DISCUSSION

In this study, numerical modeling of non-Newtonian fluid in an annulus with rotation of the inner cylinder has been performed. The computational domain, flow physics and fluid type were selected such that the results of this study can be compared with available experimental data. At first, the appropriate network is generated and independence of the solution of the computational network was also investigated for the three networks of 8000, 20000 and 50000. Eventually, the network with 20000 cells was selected as computational network. The maximum difference of results between the two 20000 and 50000 networks is about 4 percent. In figure 5 you can see a view of the produced network:

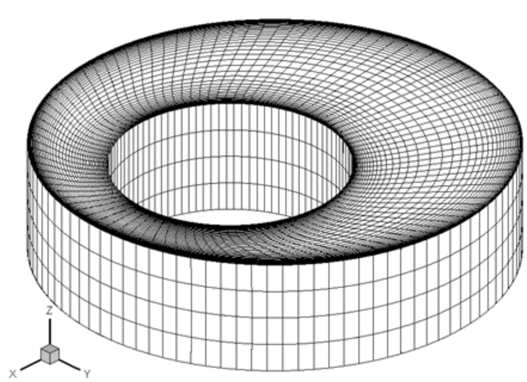

Fig. 5. Networking solution domain.

In figures 6 and 7, sensitivity of normalized horizontal and vertical velocity components to the number of computational network cells can be observed:

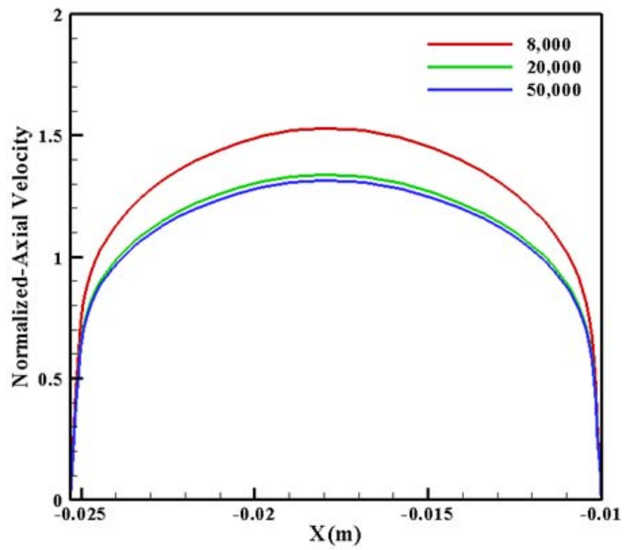

Fig. 6. Examining sensitivity of normalized flow of axial velocity to the number of computational network cells.

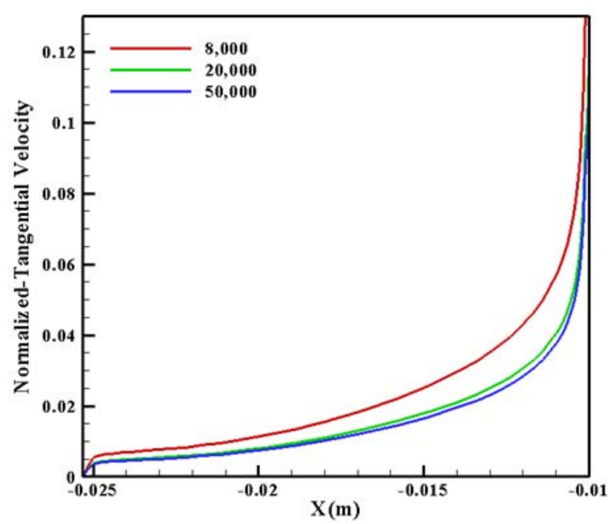

Fig. 7. Examining sensitivity of the normalized tangential velocity to the number of computational network cells.

For verifying the numerical modeling, experimental data of normalized flow of axial velocity and normal tangential velocity have been used.

In figure 8, you can see the investigation of sensitivity of the normalized axial velocity to the turbulent model along with the experimental data.

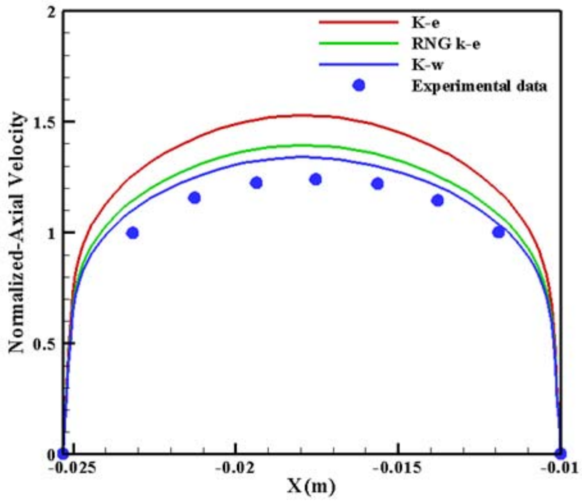

Fig. 8. Examining the sensitivity of normalized axial velocity to the turbulent model.

In figure 9, sensitivity of normalized tangential velocity to the turbulent model along with the experimental data is visible.

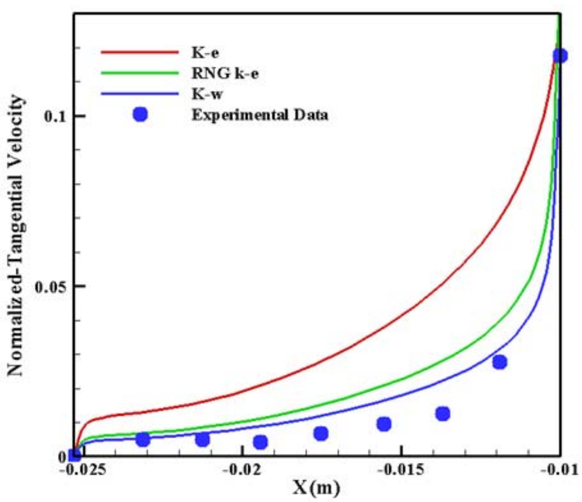

Fig. 9. Examining the sensitivity of normalized tangential velocity to the turbulent model.

As it can be observed in figures 8 and 9, k-w model is more compatible with experimental data of Nouri *et al.* (1993) and thus is a more appropriate model for analyzing this type of flow. The maximum difference of the results of this model and experimental data is about 8 percent.

CONCLUSION

The aim of the study is to select an appropriate numerical model for analyzing non-Newtonian fluid flow in an eccentric annulus. For verifying the turbulent flow model, experimental data of Nouri and Whitelaw (1997) has been used. Thus, computational domain, characteristics of the fluid and flow were selected such that to be compatible with experimental conditions of these researchers.

First, an appropriate model was generated for numerical solution of basic equations on the computational domain. Independence of solution of the computational network was also examined for the three 8000, 20000 and 50000 networks. Eventually, the network with 20000 cells was selected as computational network. The maximum difference of results of the two 20000 and 50000 networks is about 4 percent. In the following, by applying the three models, results of normalized axial and tangential velocities along with the experimental data have been drawn. The results of this study showed that k-w model is more compatible with experimental data and thus is an appropriate model for analyzing this type of flow.

REFERENCES

Brighton, JA. and Jones, JB. 1964. Fully developed turbulent flow in annuli. J. Basic Eng. (86):835-844.

Escudier, MP., Gouldson, IW. and Jones, D M. 1994. Flow of shear-thinning fluids in a concentric annulus. Exp. Fluids. (18):1-14.

Escudier, MP. and Gouldson, IW. 1995. Concentric annular flow of shear thinning liquids with center body rotation. Int. J. Heat Fluid Flow. (16):156-162.

Nouri, JM. and Whitelaw, JH. 1997. Flow of Newtonian and non-Newtonian fluid in an eccentric annulus with rotation of the inner cylinder. Int. J. Heat and Fluid Flow. (2): 236-246.

Nouri, JM., Umur, H. and Whitelaw, JH. 1993. Flow of Newtonian and non-Newtonian fluids in concentric and eccentric annuli. J. Fluid Mech. (253):617-641.

Received: Feb 23, 2013; Accepted: April 1, 2013