International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2019): 7.583

Fluid Flow Analysis in a Concentric Annulus with a Rotating Inner Cylinder using ANSYS

M.R. Sahin

[1]Department of Mechanical Engineering, NERIST, Nirjuli, Arunachal Pradesh, India

Abstract: ***In the present study, CFD (Computational Fluid Dynamics) analysis of velocity and pressure fields that occur in annular gap between two concentric cylinders was done. It encompassed the effects of inner cylinder rotation and fluid viscosity on an axial flow structure. Fluid flow in concentric annuli with and without rotating inner cylinder was investigated by numerical simulation as similar flows occur in drilling operations of oil wells. So, prediction of flow in drilling pipe in an annular space between wellbore and drill pipe is essential to determine the variation in fluid pressure within the wellbore. Numerical results obtained for pressure drop and velocity profiles from the present study were compared with experimental data as well as simulated data from the literature using two non-Newtonian fluids (0.2 % Xanthan Gum and 0.2 % Carboxymethyl cellulose). The simulated results of pressure drop in present study showed a good agreement with the experimental data and also yield data that were found rather more inclined towards the simulated data. The velocity and tangential profiles also showed a good agreement with simulated velocity profiles reported in literature. The additional work also contained, the study of Newtonian fluid (water) and its effects on hydrodynamics of these annular systems and compared.***

Keywords*:* Newtonian, Non-Newtonian, Concentric, Annulus, Rotation

1. Introduction

The flow in enclosed space have received much attention because of many practical technology-driven applications such as in production of oil & gas, centrifugally-driven separation processes, electrochemical cells, fluid viscometers and chemical reactors.

The petroleum industry in particular has shown considerable interest in studies of fluid dynamics in annular space of oil drilling operations. A typical example is the case of annular flow of mud between drill shaft and drilling well casing to remove cuttings and friction generated heat after drilling operation. The cut region is cleaned to avoid unnecessary rise in torque due to accumulation of particles **[1]**. The flow of drilling fluid is highly affected by these drag particles which is determined by velocity profiles in annular region **[2]**. Therefore, for efficient drilling operations, knowledge based on drilling fluid hydraulics is necessary. Moreover, the rotating inner drill pipe promotes swirling fluid motion that is superimposed on pressure driven axial flow in bore well which is maintained axially downstream. In practical field, pipe rotation drastically decreases frictional pressure loss inside the wellbores **[3]**. So, these inner pipe rotations too have a significant influence on drilling fluid hydraulics and performance. Moreover drilling muds shows Non-Newtonian behavior which is complicated to describe in a simple model. So, proper selection of rheological model is necessary for calculations to describe drilling fluid rheology.

An experimental study was done on combined axial and rotational flow in annulus section of rotating inner wall (Fig. 1) **[4]**.They mainly identified the following basic flow regimes in the annular gap: laminar flow, laminar flow with vortices, turbulent flow, and turbulent flow with vortices.

The works on flow velocity characteristics of rotating and non-rotating Newtonian fluids in concentric and eccentric annuli stands out among experimental studies on turbulent flow **[5]**. These authors used Newtonian and Non-Newtonian fluids to analyze turbulent flow in vertical annular sections and to determine mean velocity profiles, their fluctuations and cross-correlation by means of LDV technique (Laser Doppler Velocimetry).

An extended study on their earlier experimental studies [5] was done on vertical turbulent flows including the rotational effects of inner tube **[6]**. Further, studies on an eccentric annulus was done and was found that the influence of inner cylinder rotation is more significant in the range of Reynolds number (Re < 3000). However, this influence is found to be lower in turbulent flows.

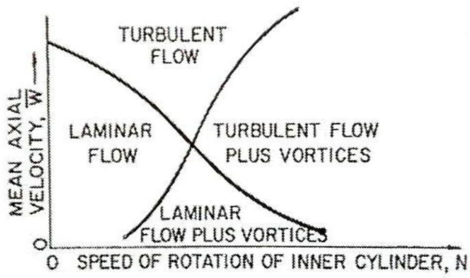


Figure 1: Schematic representation of different regions of flow in annulus [4]

Experimental study of Newtonian and pseudoplastic fluid flow was performed under the influence of central body rotation by using LDA (Laser Doppler Anemometry) as the measuring technique **[7]**. They used glucose syrup solutions as Newtonian fluid and Carboxymethyl-cellulose (CMC) solutions as non-Newtonian fluid to plot velocity profiles for different flow situations and also highlighted the behavior of the friction factor as a function of fluid flow rate.

Evaluation was done on the flow of pseudoplastic fluids in eccentric annulus through numerical simulations using finite difference technique **[8]**. The effects of inner cylinder rotation on laminar flow of both Newtonian and Non-

Newtonian fluids were studied in an eccentric annular section to compare experimental and numerical results and also to highlight velocity profiles with those reported by other authors (**[5], [1]) [9].**

With rapid improvement of computational resources, numerical simulation using CFD technique has become very popular in several fields of Engineering, **[10-16]** helping to shed light on the fluid dynamic behavior of annular flows. The effects of porosity were incorporated at the boundary of the system into their simulations **[17]**. They analyzed the flow to predict pressure loss behavior of drilling fluids which they later reported with information available in literature, but did not compare their numerical results with experimental data. Further, estimation on concentric annular flows in vertical and horizontal arrangements based on CFD simulations was done but without the effects of internal shaft rotation **[18]**. The above mentioned research are mostly been focused on experimental and numerical investigations without comparisons. There is need for studies that combine both experimental work and CFD simulations to investigate pressure gradients of different Non- Newtonian fluids in concentric annuli, evaluating their effects on the hydrodynamics of these systems. Therefore, this piece of work was taken by the present author during the Master degree program (2015-17) at the Institute NERIST (North Eastern Regional Institute of Engineering and Technology), Arunachal Pradesh for evaluation and validations.

In the present work, CFD simulations were performed to evaluate the pressure gradients and flow characteristics in concentric annuli, with or without inner shaft rotation. The velocity profiles and pressure drop is analyzed and validated with standard results reported **[19]** in annuli with radii ratio 0.45 using two Non-Newtonian fluids involving 0.2% Xanthan gum and 0.2% Carboxymethyl-cellulose aqueous solution. In oil and gas drilling operations, the radii ratios of conventional drilling are supposed to be in range of 0.3-0.5. Hence radii ratio of 0.45 is selected as they may show the best flow characteristics of the conventional wellbore drillings. In addition to the above, the work is extended to test the Newtonian fluid (i.e., water) to study the pressure drop and flow characteristics and compared to the results obtained for aforesaid two Non-Newtonian fluids.

2. Governing equations and Boundary Conditions

A schematic view of the geometrical configuration is shown in Fig. 2. The annular region is composed of two cylindrical bodies with inner cylinder radius (R_1=16mm) and outer cylinder radius (R_2=33.5mm). The axial length of cylinder is z= 1.5m having radii ratio 0.45, to evaluate the effect of rotating and non-rotating inner cylinders of two Non-Newtonian fluids on their pressure drop, CFD simulations were done. Velocity and constant pressure boundary conditions of the fluid were imposed on the inlet and outlet of the annulus. No-slip boundary conditions were used at the inner and outer cylinders. The rotational speed of the inner cylinder was 0 and 300rpm. An axial velocity of 0.69 m/s was applied at inlet of the cylinder. The analysis was carried out in laminar regime. The fluid velocity and pressure drop for an annulus was calculated for Non-Newtonian fluids to evaluate their effects on hydrodynamic of annular systems.

Drilling mud shows Non-Newtonian behavior which is complicated to describe in a simple model. So, proper selection of rheological model is necessary for calculations to describe drilling fluid rheology. The rheological data of Non-Newtonian fluids were best fitted by power law model which can be expressed as

$$\tau = k\,(\gamma)^{n} \qquad \text{Eq. (1)}$$

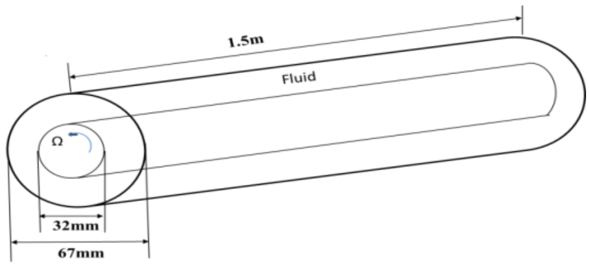


Figure 2: Concentric cylinders with inner shaft rotation

The rheological data of aqueous suspensions of 0.2% Xanthan Gum (XG) and 0.2% Carboxymethyl-cellulose (CMC) were measured using a Brookfield rheometer at a temperature of 25°C **[20]**, where K is consistency factor, n is the power law index and τ is shear rate. The power law parameters have been shown in Table 1 which was obtained by regression for both fluids using a range of shear rate (γ) varying from 0 to 80 s^{-1} to determine the parameters.

Table 1: Parameters of the power law model for the two fluids

Non-Newtonian fluids	Parameters of the power law fluid		
	K[Pa.s^n]	n[-]	R^2
0.2% XG solution	0.678	0.27	0.95
0.2% CMC solution	0.096	0.75	0.94

For an incompressible isothermal laminar fluid flow whose effective viscosity depends only on strain rate tensor, the modeling of flow can be defined by continuity equation (Eq. (2)), using axial, radial and tangential components of the momentum equation (Eq. (3), (4) and (5)) in cylindrical coordinates **[21]**.

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial(\rho r v_r)}{\partial r} + \frac{1}{r}\frac{\partial(\rho v_\theta)}{\partial \theta} + \frac{\partial(\rho v_z)}{\partial z} = 0 \qquad \text{Eq.(2)}$$

$$\rho\left(\frac{\partial v_r}{\partial t} + v_r\left(\frac{\partial v_r}{\partial r}\right) + \frac{v_\theta}{r}\left(\frac{\partial v_r}{\partial \theta}\right) + v_z\left(\frac{\partial v_r}{\partial z}\right) - \frac{v_\theta^2}{r}\right) = -\left(\frac{\partial p(r,z)}{\partial r}\right) + \rho g_r + \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial(r v_r)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2} + \frac{\partial^2 v_r}{\partial z^2} - \frac{2}{r^2}\left(\frac{\partial v_\theta}{\partial \theta}\right)\right) \qquad \text{Eq. (3)}$$

$$\rho\left(\frac{\partial v_\theta}{\partial t} + v_r\left(\frac{\partial v_\theta}{\partial r}\right) + \frac{v_\theta}{r}\left(\frac{\partial v_\theta}{\partial \theta}\right) + v_z\left(\frac{\partial v_\theta}{\partial z}\right) + \frac{v_r v_\theta}{r}\right) = -\frac{1}{r}\left(\frac{\partial p(r,z)}{\partial \theta}\right) + \rho g_\theta + \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial(r v_\theta)}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2} + \frac{\partial^2 v_\theta}{\partial z^2} + \frac{2}{r^2}\left(\frac{\partial v_r}{\partial \theta}\right)\right) \qquad \text{Eq. (4)}$$

$$\rho\left(\frac{\partial v_z}{\partial t}+v_r\left(\frac{\partial v_z}{\partial r}\right)+\frac{v_\theta}{r}\left(\frac{\partial v_z}{\partial \theta}\right)+v_z\left(\frac{\partial v_z}{\partial z}\right)\right)=-\left(\frac{\partial p(r,z)}{\partial z}\right)+\rho g_z+\mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right) \quad \text{Eq. (5)}$$

Considering the *power law* rheological model (Eq. 1)), we used the effective viscosity *(μ_E)* concept to replace the dynamic viscosity *(μ)* in these equations. The parameters of this model (Table 1) were inserted in FLUENT 15 software.

3. Numerical Simulation

The procedure for the numerical simulation was implemented using ANSYS FLUENT 15. Finite volume numerical scheme in FLUENT were employed for solving mathematical model. In FLUENT, the standard SIMPLE and SIMPLEC (SIMPLE Consistent) algorithm are available in solution methods. Simple is the default but SimpleC is more beneficial for relatively less complicated problems. Here, the simulations were performed in steady state regime, with convergence criteria of $1e^{-4}$ and SIMPLE algorithm was used for coupling pressure-velocity. PRESTO scheme was used for pressure discretization and the scheme QUICK for the discretization of equations of motion. The boundary conditions adopted were an axial velocity of 0.69 m/s at entrance and rotational speeds of 0 and 300 rpm in concentric channel of inner cylinder.

3.1 Grid Independence Study

Grid construction was also done in ANSYS Fluent software. In construction of grid, the cells adjacent to the walls of the inner and outer tubes, as well as in the regions of entry and exit were refined by mapped face meshing with a cells growth factor of 1.1. A grid independence test was performed in which the initial grid elements of 15520 were considered and subsequently the values were raised to 154400 and 192000 respectively. The final mesh of 192000 cells was chosen as it gave a grid-independent solution. Figures 3 and 4 showed the refinement of grid along the annular section and in the entry region of concentric annuli respectively.

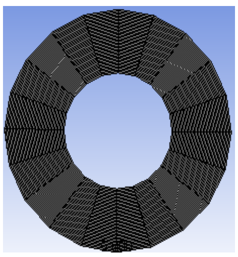


Figure 3: Grid refinement along concentric annular

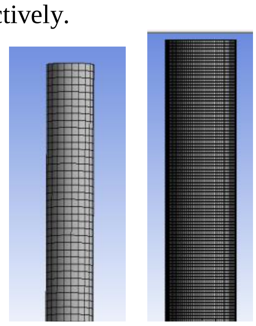


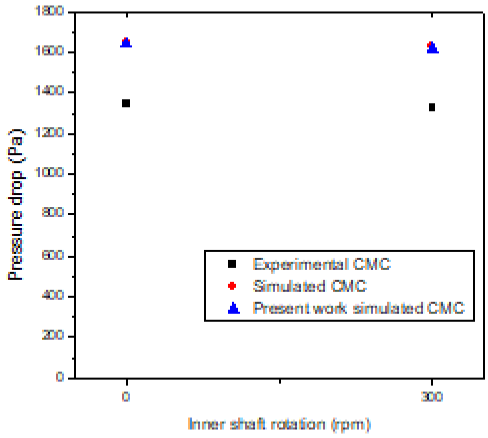
Figure 4: Grid refinement in intake region section

4. Results and Discussions

4.1 Effects of inner cylinder rotation on pressure drop of Non- Newtonian fluids

In order to validate the present work, numerical studies were simulated and the results obtained were compared with the existing results of those studies. The results obtained from the simulation for pressure drop in non-Newtonian fluids XG and CMC are shown in Table 2 and Fig.5. The results were also compared to both simulated and experimental data reported in the literature **[19]**. The difference in data might be due to following of grid construction with maximum refinement in ANSYS FLUENT. The results obtained in the Fluent were taken and plotted in software OriginPro 8.

In the present study, % reduction in pressure drop due to change in rotational speed of simulated values were found 4.0 & 1.16 against the Non-Newtonian fluids XG and CMC respectively. But the simulated values for the same Non-Newtonian fluids were calculated as 4.1 & 1.2 respectively which were already recorded in literature **[19]**. However, the former values found from present study were more nearer to the experimental and simulated values and seemed to be more precise and acceptable than reported earlier by previous workers. So the results from present study showed that on increase in pipe rotation for same inlet velocity led to decrease in pressure drop as listed in Table 2.



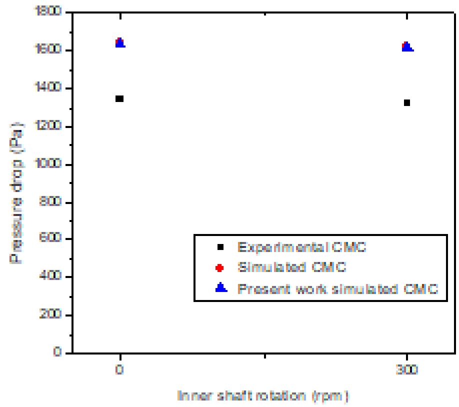


Figure 5: Comparison of Experimental pressure drop, simulated and present work simulated values in concentric annulus: (a) 0.2% XG (b) 0.2% CMC

Table 2: Pressure drop for XG and CMC obtained from present study compared with those reported in literature

Particulars	Rotational Speed(rpm)	Published				Present work Simulated	% reduction in pressure drop due to change in rotational speed of present simulated values
		Experimental	% reduction in pressure drop due to change in rotational speed of experimental values	Simulated	% reduction in pressure drop due to change in rotational speed of simulated values		
0.2% XG solution	0	865	3.87	933	4.1	923	4.0
	300	831.5		894		886	
0.2% CMC solution	0	1337	1.12	1648	1.2	1639	1.16
	300	1322		1628		1620	

4.2 Effect of inner cylinder rotation on flow characteristics of Non- Newtonian fluids

In this section, the numerical results of fluid flow characteristics for two Non-Newtonian fluids (0.2% XG and 0.2% CMC) with and without inner cylinder rotation cases were reported. Velocity vectors and contour along with pressure contour at different rotational speed (0rpm and 300rpm) were presented. Practically, the flow of drilling fluid is highly affected by drag particles accumulated after drilling which is determined by velocity profiles in annular region [2]. So, it becomes necessary to determine velocity profiles in annular region for efficient drilling operations.

In Fig.6 (a) the maximum velocity of XG was located at the centerline in the annulus of concentric cylinders without inner shaft rotation and it showed a non-parabolic profile. This happened due to pseudoplastic nature of this fluid having power index n=0.27. Fig. 6 (b) indicated velocity contours that showed maximum velocity was located at centerline in annulus of concentric cylinder without inner shaft rotation. Again in Fig. 7 (a) the velocity increased towards the inner part of the gap as the inner cylinder rotated at 300 rpm and Fig. 7 (b) showed the corresponding velocity contour that indicated that maximum velocity increased near inner shaft due to rotational speed.

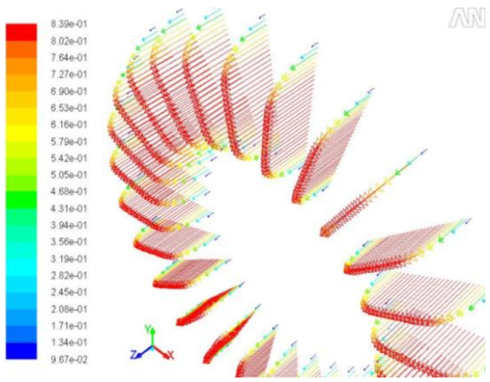


Figure 6: (a) Velocity vector of XG without inner shaft **rotation**

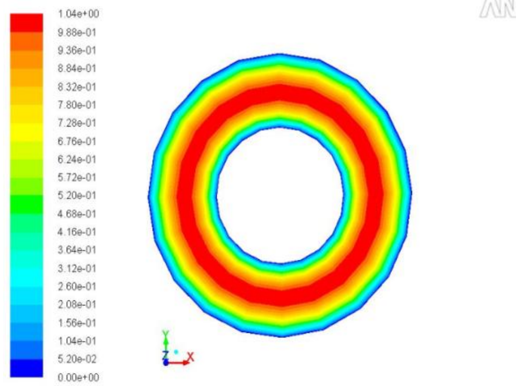


Figure 6: (b) Velocity contour for XG (0rpm)

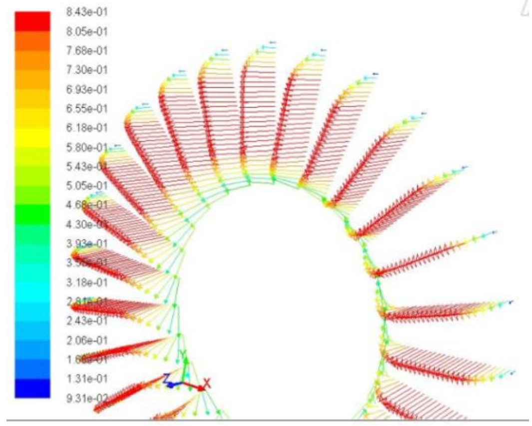


Figure 7: (a) Velocity vector of XG with inner shaft rotation at 300 rpm

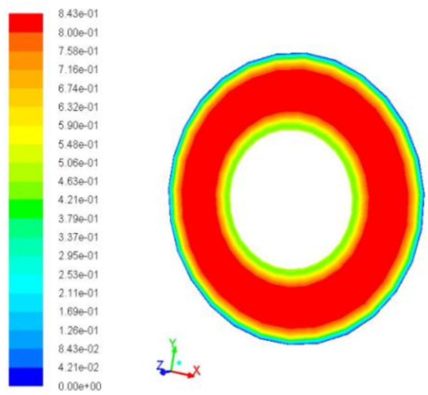


Figure 7: (b) Velocity contour for XG (300rpm)

The pressure decreased at outlet of the cylinders in Fig. 8 (a) whereas in Fig. 8 (b) with shaft rotation, the pressure decreased more rapidly at the outlet.

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2019): 7.583

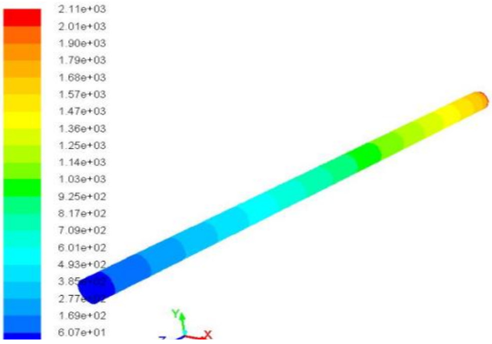


Figure 8: (a) Pressure contour of XG without inner shaft rotation

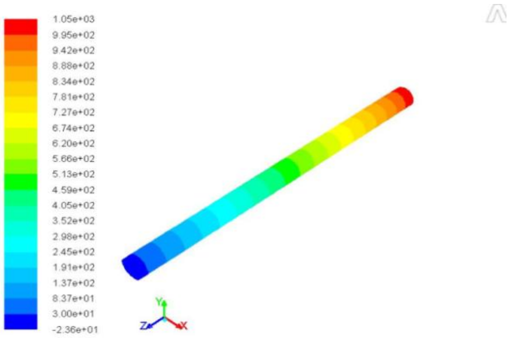


Figure 8: (b) Pressure contour of XG with inner shaft rotation at 300 rpm

In Fig 9 (a) the maximum velocity was located at centreline of annulus. It showed almost parabolic profile as CMC has power law index, n=0.75, which was closer to Newtonian fluid (n=1). Velocity contour for CMC in concentric cylinder without inner shaft rotation was shown in Fig. 9 (b).

Again in Fig. 10 (a) and 10 (b), the velocity vector and contour indicated that the maximum velocity increased towards the inner part as the cylinder rotated at 300 rpm.

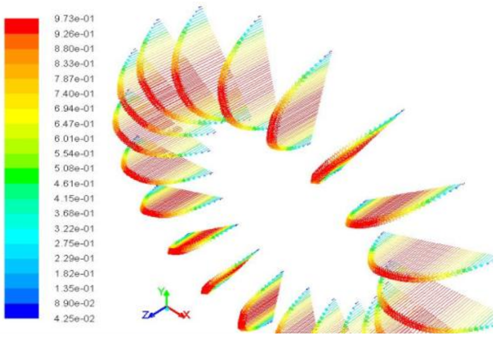


Figure 9: (a) Velocity vector of CMC (0rpm)

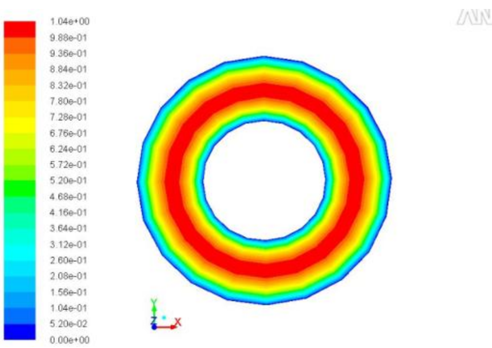


Figure 9: (b) Velocity contour of CMC (0rpm)

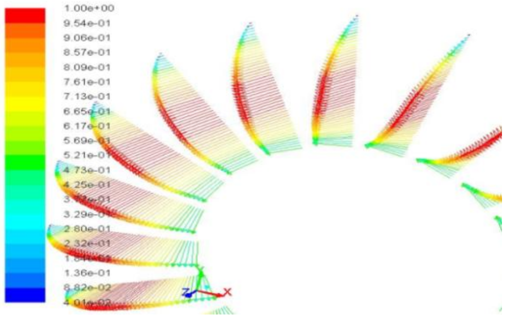


Figure 10: (a) Velocity vector of CMC with shaft rotation at 300 rpm

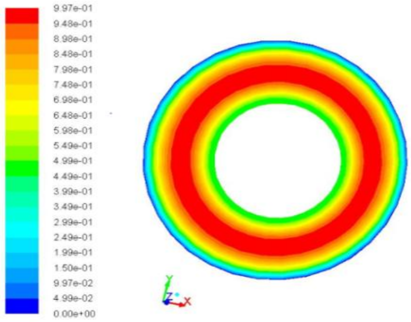


Figure 10: (b) Velocity contour of CMC (300 rpm)

In Fig.11 (a) the pressure decreased at outlet of the cylinders and whereas Fig 11 (b) indicated that with shaft rotation, the pressure decreased more rapidly at the outlet.

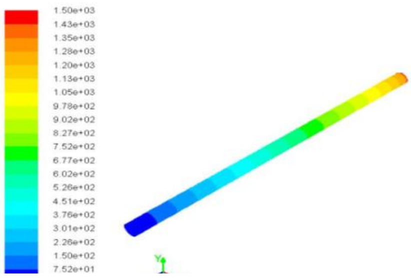


Figure 11: (a) Pressure contour of CMC without inner shaft rotation

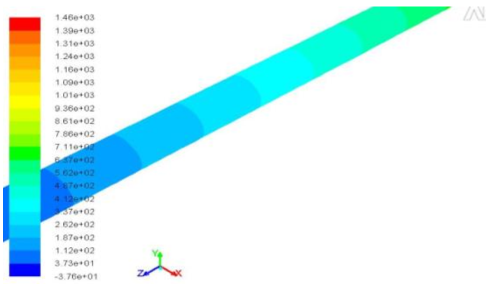


Figure 11: (b) Pressure contour of CMC with inner shaft rotation at 300 rpm

4.3 Effect of inner cylinder rotation on Axial and Tangential Velocities of Non- Newtonian fluids

In this section, the numerical results of axial and tangential velocities for two Non-Newtonian fluids (0.2% XG and 0.2% CMC) with and without inner cylinder rotation cases were reported and compared. The simulated profiles of axial and tangential velocities were presented and compared with the profiles of [19].

In Figure 12 (b) profiles of axial velocities were almost similar to simulated profiles reported in the literature for both the fluids in Figure 12 (a). Distances were normalized by radial distance r1 to S from outer to inner cylinder and velocity was normalized by bulk velocity U_b (0.69m/s). Significant differences could be seen in profiles normalized axial velocities and it was noted due to the rheological characters of the fluids.

The 0.2% CMC solution presented a parabolic profile as it has a behavior index (n=0.75) very close to Newtonian fluid (n=1).But, 0.2% XG solution having low behavior index (n=0.27) displays a flattened profile. It was also seen that the effect of rotation of inner shaft (300 rpm) did not influence the axial velocity profiles as shown in the Figure 12.

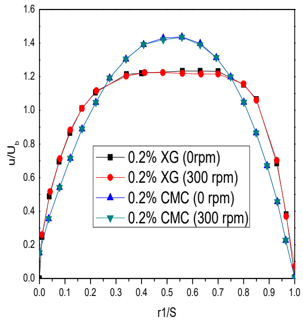


Figure 12: (a) Simulated profiles of axial velocity reported in literature.

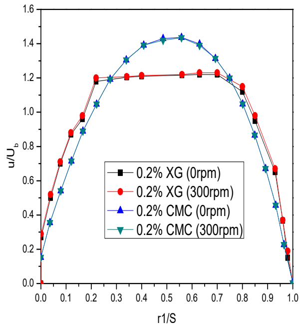


Figure 12: (b) Simulated profiles of axial of present work

Fig.12. Validation of axial velocity profiles normalised by bulk velocity (U_b) for fluids (0.2% XG and 0.2 % CMC) in concentric annulus

In Figure 13, present study showed almost same tangential velocity profiles as simulated values reported in literature [19]. In 0.2% XG solution, the tangential velocity decreased sharply as fluid moved away from the inner cylinder as its non-Newtonian behavior was more evident. In contrast against 0.2% CMC, tangential velocity decreased gradually.

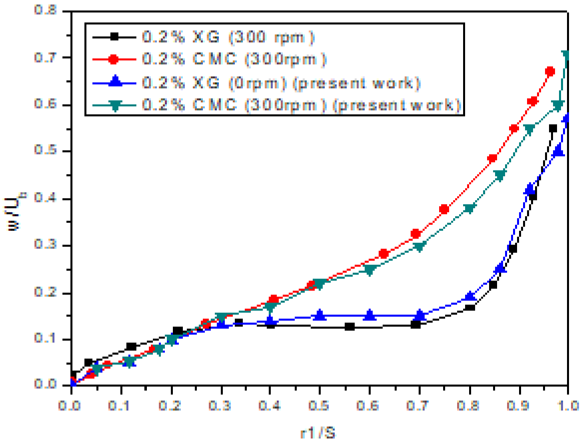


Figure 13: Validation of simulated and present work tangential velocity profiles normalised by bulk velocity (U_b) for fluids (0.2% XG and 0.2 % CMC) in concentric annulus.

4.4 Effects of Inner cylinder rotation on pressure drop of Newtonian fluid

In this section, for further validation, the work was extended for studying flow characteristics in concentric annuli using Newtonian fluid. Water was taken here as Newtonian fluid for analysis. Considering same parameters and boundary conditions, the results obtained after simulation were analyzed below and were compared to the results obtained for the two non-Newtonian fluids (XG and CMC) as shown in Table 3.

It was noted that the pressure drop for water reduced slightly to the tune of 2.9% (Fig.14), which might be due to introduction of inner cylinder rotation (300rpm). By comparison, it was found that pressure drop varied with different fluids flowing through the annulus of rotating cylinders. This was due to differences in behavior of fluids having power index varying from n=0.27 to n=1 (Table 1).

Table 3: Comparisons of pressure drop for 0.2% XG, 0.2% CMC and water due to change in rotational speed

Rotational speed (RPM)	Particulars		
	0.2% XG	0.2% CMC	Water
0	923	1639	1710
300	886	1620	1660
% reduction in pressure drop	4.0	1.16	2.9

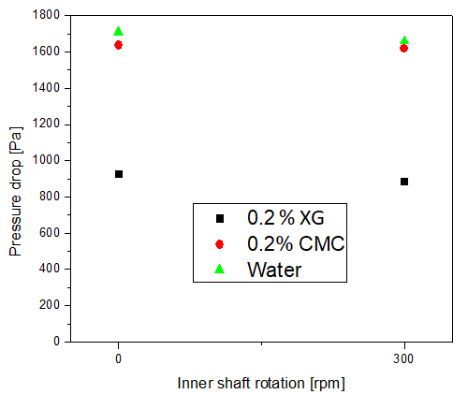


Figure 14: Comparison of pressure drop for both Newtonian and Non-Newtonian fluids

In this section, the numerical results of fluid flow characteristics for Newtonian fluid (Water) with and without inner cylinder rotation cases were reported. The velocity vectors, velocity contour and pressure contour for water were presented here.

In Fig. 15 the maximum velocity without shaft rotation was located at centerline of annulus in concentric cylinders. The profile is parabolic in nature as water has power law index n=1.

The velocity contour as well was shown in Fig. 16. But with shaft rotation at 300 rpm, the maximum velocity occurred near the inner rotating cylinder as shown in Fig. 17 and Fig. 18 respectively.

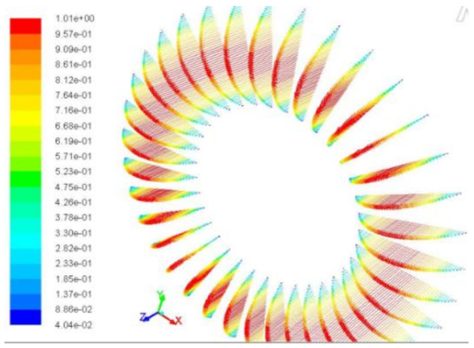


Figure 15: Velocity vectors without inner shaft rotation (Water)

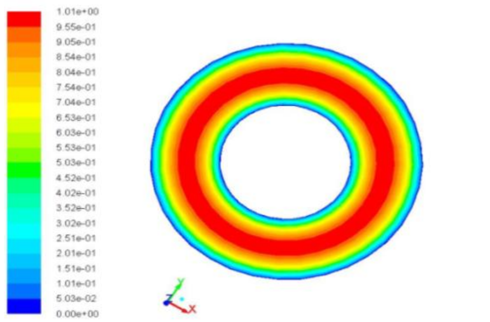


Figure 16: Velocity contour of Water (0rpm)

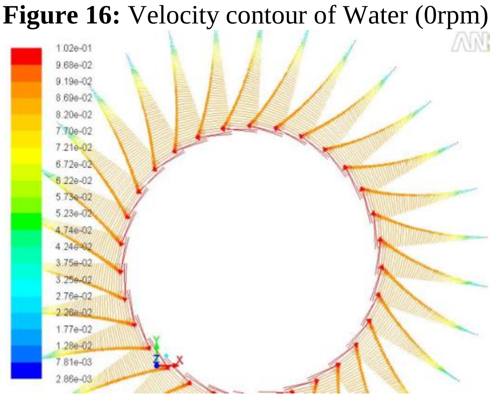


Figure 17: Velocity vectors with inner shaft rotation at 300 rpm (Water)

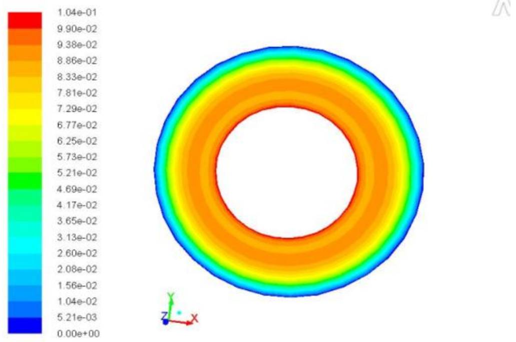


Figure 18: Velocity contour of Water (300 rpm)

In, Fig. 19 (a) the pressure decreased at outlet of the cylinders but Fig 19 (b) indicated that with shaft rotation, the pressure decreased more rapidly at the outlet.

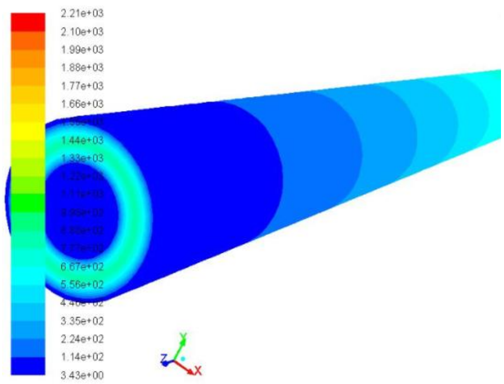


Figure 19: (a) Pressure contour of Water (0rpm)

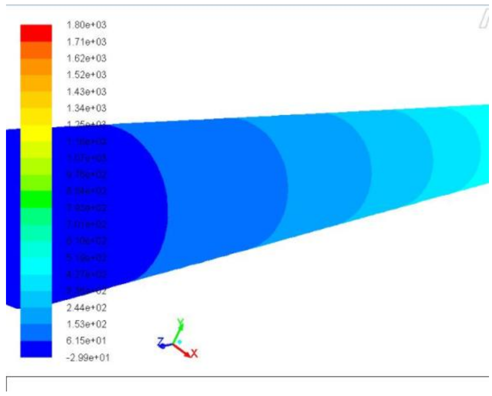


Figure 19: (b) Pressure contour of Water (300rpm)

4.6 Effects of inner cylinder rotation on axial and tangential velocities of Newtonian fluid

In this section, the numerical results of axial and tangential velocities for Newtonian fluid (Water) with and without inner cylinder rotation cases were reported. The simulated profiles of axial and tangential velocities were presented and compared with the two Non-Newtonian fluids (0.2% XG and 0.2% CMC).

In Figure 20, it was seen that water gives a parabolic velocity profile as its power law index is n=1 as compared to XG having a flat profile (n=0.27) and CMC with almost parabolic profile (n=0.75).

Moreover, increment in shaft rotation (300rpm) practically did not interfere in the axial velocity profiles. The differences in profiles of three fluids were due to its rheological characteristics.

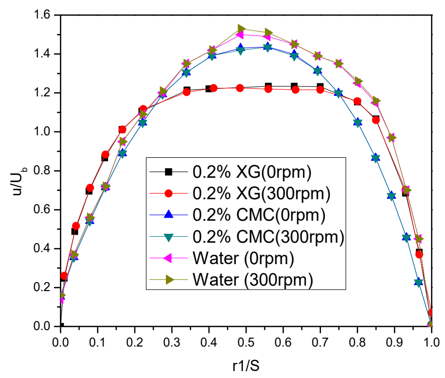


Figure 20: Comparison of axial velocities for both Newtonian and non-Newtonian fluids

In Figure 21, the tangential velocity of water decreased gradually but there was sharp decrease in tangential velocity against Non-Newtonian fluids particularly XG. From comparison with the non-Newtonian fluids, it could be noted that the velocity gradient of the Newtonian fluid was greater than that of Non-Newtonian fluid.

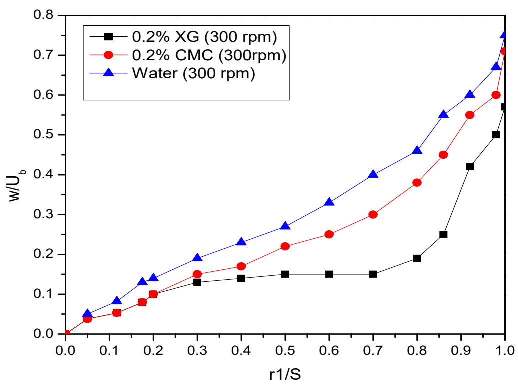


Figure 21: Comparison of tangential velocity both Newtonian and Non-Newtonian fluids

5. Conclusion

The present study revealed the following outcomes of the investigation:

1) The simulated results of present study of the two fluids showed a decrease in pressure drop with inner cylinder rotation in the concentric annulus. The pressure drop after comparison showed nearest values to experimental values as well better results than simulated values reported in literature **[19]**. Hence, it was inferred that the computation for pressure drop was almost accurate and validated using ANSYS Fluent 15.
2) The present simulated results showed that the axial velocities in the concentric annulus were flat for XG solution and parabolic for CMC solution. But, the influences of inner shaft rotation on the profiles were negligible. Again, the tangential velocity profiles showed a sudden decrease in tangential velocity of XG solution as it moved away from the inner cylinder, while this decrease was more gradual with the CMC solution. The velocities after comparison showed similar profiles to simulated velocities collected from the literature. Hence, it was in good conformity with the previous work cited in literature.
3) To accumulate more information and precise data, the current study was extended to Newtonian fluid and compared with the two non-Newtonian fluids. The study revealed that the pressure drop for water reduced slightly (reduction of 2.9%) due to introduction of inner cylinder rotation (300rpm). After comparison, it was computed that pressure drop varied with different fluids flowing through the annulus of rotating cylinders. This was due to differences in behavior of fluids all having power index varying from n=0.27 to n=1. The property of Newtonian Fluid (n=1) was substantiated in the present investigation.
4) For axial velocities water gave a parabolic velocity profile as its power law index is n=1 as compared to XG having a flat profile (n=0.27) and CMC with almost parabolic profile (n=0.75). Moreover, increment in shaft rotation (300rpm) practically did not interfere in the axial velocity profiles. The differences in profiles of three fluids were found due to their variable rheological characteristics.
5) The tangential velocity for water decreased gradually but there was sharp decrease in tangential velocity against Non-Newtonian fluids particularly XG. From comparison with the Non-Newtonian fluids, it was found that the velocity gradient of the Newtonian fluid was greater than that of Non-Newtonian fluid.

Finally, it was inferred that the results obtained from present study were more nearer to the values obtained by the previous workers and hence more reliable. The extended work also helped in differencing the rheological characteristics of various fluids which has effects on hydrodynamics of annular systems.

References

[1] Nouri, J. M. and J.H. Whitelaw (1997). Flow of Newtonian and non-Newtonian fluids in an eccentric annulus with the rotation of the inner cylinder. *Int. J. Heat and Fluid Flow*, 18, 236-246.

[2] Escudier, M.P., I.W., Gouldson, P.J. Oliveira and F.T. Pinho (2000). "Effects of inner cylinder rotation on laminar flow of a Newtonian fluid through an eccentric annulus". *Int. J. Heat Fluid Flow*,21, 92-103.

[3] Duan, M., S. Mishka, M. Yu, N.E. Takach, R.M. Ahmed and J.H. Hallman (2010). Experimental study and modeling of cutting transport using foam with drill pipe rotation. *SPE Drilling & completion*, 25(3), 352-362.

[4] Kaye, J. and E.O.,Elgar (1958). "Modes of Adiabatic and Diabatic Fluid Flow in an Annulus with an Inner Rotating Cylinder, *Trans. Asme*, 80, 753-765.

[5] Nouri, J. M., H. Umur and J. H. Whitelaw (1993). "Flow of Newtonian and non-Newtonian fluids in concentric and eccentric annuli". *J. Fluid. Mech.*, 253,617- 641.

[6] Nouri, J.M. and J. H. Whitelaw, (1994). "Flow of Newtonian and non-Newtonian fluids in a concentric annulus with rotation of the inner cylinder", *J. Fluid. Eng.*, 116, 821-827.
[7] Escudier, M. P. and I. W. Gouldson, (1995). Concentric annular flow with center body rotation of a Newtonian and a shear-thinning liquid. *Int. J. Heat Fluid Flow*, 16, 156-162.
[8] Fang, P., R. M. Manglik and M. A. Jog (1999). Characteristics of laminar viscous shear-thinning fluid flows in eccentric annular channels. *J. Non-Newtonian Fluid Mech.*, 84, 1-17.
[9] Escudier, M.P., P.J. Oliveira, F.T. Pinho (2002). Fully developed laminar flow of purely viscous non-Newtonian liquids thrpugh annuli, including effects of eccentricity and inner cylinder rotation. *Int. J. Heat Fluid Flow*, 23, 52-73.
[10] Vieira Neto, J.L., A. L. Martins, C.H. Ataide and M.A.S. Barrozo (2014). The effects of inner cylinder rotation on the fluid dynamics of Non-rotating Newtonian fluids in concentric and eccentric annuli, *Braz. J. Chem. Eng.*,31(4), 829 – 883.
[11] Vieira Neto, J.L., C.R. Duarte, V.V. Murata, M.A.S. Barrozo (2008). Effect of a draft tube on the fluid dynamics of a spouted bed: Experimental and CFD studies. *Drying Technol.*, 26, 299-307.
[12] Cunha, F. G., K. G. Santos, C. H. Ataíde, N. Epstein and M. A. S. Barrozo (2009). Annatto powder production in a spouted bed: An experimental and CFD study. *Ind. Eng. Chem. Res.*, 48, 976-982.
[13] Barrozo, M. A. S., C. R. Duarte, N. Epstein, J. R. Grace and C. J. Lim (2010).Experimental and computational fluid dynamics study of dense-phase, transition region and dilute-phase spouting. *Ind. Eng. Chem.*, 49, 5102-5109.
[14] Santos, K. G., V. V. Murata and M. A. S. Barrozo (2009). Three-dimensional computational fluid dynamics modeling of spouted bed. *Can. J. Chem. Eng.*, 87, 211-219.
[15] Oliveira, D. C., C. A. K. Almeida, L. G. M. Vieira, J. J. R. Damasceno and M. A. S. Barrozo (2009). Influence of geometric dimensions on the performance of a filtering hydrocyclone: An experimental and CFD study. *Braz. J. Chem. Eng.*, 26(3), 575-582.
[16] Pereira, F. A. R., M. A. S. Barrozo and C. H. Ataíde, (2007).CFD predictions of drilling fluid velocity and pressure profiles in laminar helical flow. *Braz. J. Chem. Eng.*, 24, 587-595.
[17] Fisher, K. A., R. J. Wakeman, T. W. Chiu and O. F. J. Meuric (2000). Numerical modeling of cake formation and fluid loss from non-Newtonian muds during drilling using eccentric/concentric drill strings with/ without rotation. *Chem. Eng. Res. Des.*, 78, 707714.
[18] Ali, W. A. (2002).Parametric study of cutting transport in vertical and horizontal well using computational fluid dynamics (CFD). *M. Sc. Thesis, Department of Petroleum and Natural Gas Engineering, West Virginia University, United States*.
[19] Vieira Neto, J.L., A. L. Martins, C.H. Ataide and M.A.S. Barrozo (2014). The effects of inner cylinder rotation on the fluid dynamics of Non-rotating Newtonian fluids in concentric and eccentric annuli, *Braz. J. Chem. Eng.*,31(4), 829 – 883.
[20] Ataíde, C. H., F. A. R. Pereira and M. A. S. Barrozo (1999). Wall effects on the terminal velocity of spherical particles in Newtonian and non-Newtonian fluids. *Braz. J. Chem. Eng.*, 16, 387-394.
[21] Bird, R. B., W. E. Stewart and E. N. Lightfoot (2002).Transport Phenomena. *Second Edition, John Wiley & Sons Inc., New York.*

Author Profile

Mehtaj Ridwana Sahin is currently working in Jorhat Polytechnic as Lecturer in Mechanical (Automobile Deptt.), Jorhat. Assam. She did her Master's degree in Mechanical Engineering with area of specialization in Thermo-Fluid from NERIST, Nirjuli, Arunachal Pradesh in 2017, graduated from Dibrugarh University, Dibrugarh, Assam in 2014.