

Research Article

CFD Analysis of Pressure Losses and Deposition Velocities in Horizontal Annuli

Rasel A. Sultan ⁽¹⁾, ¹ Mohammad Azizur Rahman, ² Sayeed Rushd, ³ Sohrab Zendehboudi, ⁴ and Vassilios C. Kelessidis⁵

¹BID Group Technologies LTD., Deltech Manufacturing Division, Prince George, BC, Canada
 ²Texas A&M University at Qatar, Doha, Qatar
 ³King Faisal University, Al Ahsa, Saudi Arabia
 ⁴Memorial University of Newfoundland, St. John's, NL, Canada
 ⁵Petroleum Institute, Abu Dhabi, UAE

Correspondence should be addressed to Rasel A. Sultan; ras380@mun.ca

Received 31 October 2018; Accepted 5 January 2019; Published 3 February 2019

Academic Editor: Evangelos Tsotsas

Copyright © 2019 Rasel A. Sultan et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Estimation of pressure losses and deposition velocities is vital in the hydraulic design of annular drill holes in the petroleum industry. The present study investigates the effects of fluid velocity, fluid type, particle size, particle concentration, drill string rotational speed, and eccentricity on pressure losses and settling conditions using computational fluid dynamics (CFD). Eccentricity of the drill pipe is varied in the range of 0–75%, and it rotates about its own axis at 0–150 rpm. The diameter ratio of the simulated drill hole is 0.56. Experimental data confirmed the validity of current CFD model developed using ANSYS 16.2 platform.

1. Introduction

In the petroleum industry, predicting frictional pressure losses and settling conditions for the transportation of drilling fluids in the annuli are important for drilling operations. Inaccurate predictions can lead to a number of costly drilling problems. A few examples of such problems are loss of circulation, kicks, blockage, wear, abrasion, and improper rig power selection. The existing empirical models become less accurate as those involve many simplified assumptions. CFD simulations help to minimize such assumptions by using the physics-based Navier-Stokes equations to model the hydrodynamics of the flow system. Current work is focused on developing a comprehensive CFD model which is capable of considering the effects of all important drilling parameters, such as fluid velocity, fluid type, particle size, particle concentration, drill pipe rotation speed, and drill pipe eccentricity.

2. Literature Review

The estimation of pressure loss in an annulus is more difficult compared with pipe flow due to the complexities in hydraulics resulting from the complex geometry [1, 2]. From an empirical perspective, the issue is usually addressed by replacing pipe diameter in the pipe flow models with an "effective diameter" of annulus. A number of definitions of "effective diameter" have been proposed till date. However, it is difficult to select a definition for a field application as those were developed and/or applied empirically. A comparison of multiple definitions in predicting pressure losses is presented by Anifowoshe and Osisanya [3]. Other issues that make the estimation of pressure losses in drilling holes difficult are the eccentricity and the rotational speed of inner drill pipe. Many studies have been done on the flow of non-Newtonian fluids in annuli to introduce empirical/ analytical models which allow to take these effects into

account [1, 3–10]. The results of the previous studies show that the annular pressure losses for non-Newtonian (power law) fluids flowing in a drill hole depend on drill pipe rotation speed, fluid properties, flow regimes (laminar/transitional/turbulent), diameter ratio, eccentricity, and equivalent hydrodynamic roughness.

Using commercially available CFD packages like ANSYS FLUENT to predict pressure losses for the annular transportation of drilling fluids is comparatively a new approach. Sorgun and Ozbayoglu [9] demonstrated the better performance of CFD model compared with the existing empirical models in predicting frictional pressure losses. Sorgun [8] investigated the effect of pipe eccentricity on pressure loss, tangential velocity, axial velocity, and effective viscosity by using CFD. Erge et al. [6] presented a CFD modeling approach which is applicable to estimate frictional pressure losses in an eccentric annulus with inner pipe rotation while circulating yield power law fluids. However, most of these CFD studies were limited to the laminar flow of a single phase in hydro dynamically smooth annuli.

Annular flow of drilling fluids containing cuttings, i.e., slurry, has not been studied in sufficient detail. Examples of works in the field of annular slurry flow are available in references [11-14]. The focus of these studies was to understand the hydrodynamics of the slurry flow in annulus from real-time experiments and to produce empirical models based on data analysis. Recently, different researchers [15-17] have used CFD in studying the transportation of slurry in annuli. Ofei [15] examined the effect of the rheological parameters of the carrying fluids on the velocity of solid. Sorgun and Ulker [16] compared the predictions of pressure losses obtained using artificial neural network (ANN) and CFD. Both methods produced comparable results. Sun et al. [17] studied the effects of inclination, rotational speed, and flow rate on the distribution of solid concentration and the frictional pressure loss. Similar to the single-phase annular flow works, most of these CFD studies were limited to the laminar slurry flow conditions.

3. Methods

3.1. CFD Simulation. In the current work, the CFD simulation model of the annular slurry flow is developed using ANSYS Fluent 16.2 platform. Following previous works [18–20], a multi-fluid granular model is used to describe the flow behavior of a fluid-solid mixture. The granular version of Eulerian model is selected as the multiphase model (Appendix C). This is because high solid volume fraction is expected to be used for this study and the granular version captures the hydrodynamics of high concentration slurries consisting of varying grain sizes. It allows modeling of multiple separate but interacting phases. The phases can be liquids, gases, or solids in nearly any combination. The Eulerian–Lagrangian treatment that is used for the discrete phase model.

The description of multiphase flow as interpenetrating continua incorporates the concept of phasic volume fractions,

which represent the space occupied by each phase. Each phase satisfies the laws of conservation of mass and momentum individually. The conservation equations are modified by averaging the local instantaneous balance for each of the phases [21] or by using the mixture theory approach [22]. A detailed description is available in Appendix A.

For Eulerian multiphase calculations, the phase-coupled SIMPLE (PC-SIMPLE) algorithm is used for the pressurevelocity coupling. PC-SIMPLE is an extension of the SIMPLE algorithm to multiphase flows [23, 24]. The velocities coupled by phases are solved in a segregated fashion. The block algebraic multigrid scheme used by the densitybased solver is used to solve a vector equation formed by the velocity components of all phases simultaneously [25]. Then, a pressure correction equation is built based on the total volume continuity. Pressure and velocities are then corrected so as to satisfy the continuity constraints.

To ensure stability and convergence of iterative process, a second-order upwind discretization was used for momentum equation, and first upwind discretization was employed for volume fraction, turbulent kinetic energy, and its dissipation. Upwinding refers to the face value derived from quantities in the cell upstream, or "upwind," relative to the direction of the normal velocity. When first-order accuracy is desired, quantities at cell faces are determined by assuming that the cell-center values of any field variable represent a cell-average value and hold throughout the entire cell; the face quantities are identical to the cell quantities. Thus, the face value is set equal to the cell-center value of the upstream cell when first-order upwinding is selected. In contrast, when second-order accuracy or second-order upwinding is desired, quantities at cell faces are computed using a multidimensional linear reconstruction approach [26]. In this approach, higher order accuracy is achieved at cell faces through a Taylor series expansion of the cellcentered solution about the cell centroid. The total simulation process is shown in Figure 1.

3.2. Turbulence Model Selection. Turbulent quantities for fluid flow are computed using Reynolds stress model [27-29] (Appendix D). Abandoning the isotropic eddy-viscosity hypothesis, the RSM closes the Reynolds-averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for the dissipation rate [30]. Here, five additional transport equations are required in 2D flows and seven additional transport equations must be solved in 3D (please see Appendix B for further details). The turbulence model was selected for the current work, analyzing the relative performance of different models. A typical example of the analysis is presented in Figure 2. In this figure, % difference refers to the percentile difference between the experimental measurements of pressure loss and the corresponding CFD predictions. In most cases, the difference was less than 10% when RSM was used. A list of the important results is presented in Table 1.

3.3. Length Independence Study. The length of the flow domain is considered long enough to achieve the fully



FIGURE 1: Simulation process diagram.



FIGURE 2: Performance of turbulence models in predicting pressure loss (data source: Kaushal et al., 2005 [31]).

developed flow. Minimum entrance length considered for the flow development is $50D_h$, where hydraulic diameter $D_h = OD-ID$ [36, 37]. An example of the length independent test is shown in Figure 3. The simulation results were not dependent on length after 3 m from the inlet.

3.4. Mesh Analysis. The computational grids for an annular section are generated using ANSYS Fluent, and the meshing is finalized on the basis of proper mesh

independency check. Multiple layers of inflation near wall are added from both inner and outer walls to compute the characteristics of different parameters near wall more precisely. Shear stress between wall surface and fluids is much higher, and this inflation helps to create denser meshing near wall. An example of computational grid distribution and mesh independence test is shown in Figures 4 and 5. Mesh independent results could be produced for more than 800000 number of nodes. All the results presented in the current work were obtained using around 900000 nodes.

The values of dimensionless wall distance (y^+) were checked during near wall cells generation in consideration of the convergence requirement of y^+ for cells adjacent to the wall. The value of y^+ depends on the wall shear stress, fluid density, hydraulic diameter, and molecular viscosity:

$$y^{+} = \frac{y}{\mu} \sqrt{\rho \tau_{w}},\tag{1}$$

where y is the distance from the wall to the cell center; μ , the molecular viscosity; ρ , the fluid density; and τ_w , the wall shear stress. Eventually, y^+ depends on the mesh resolution and the flow Reynolds number. Default standard wall

Reference	Experimental conditions		% Difference [100(experiment–simulation)/experiment]			
	*	RSM	SST k - ω	Standard k - ω	Standard k - ε	
Kaushal et al. (2005) [31]	Pipe orientation: horizontal Pipe dia.: 0.0549 m Pipe material of construction: stainless steel (density 8030 kg/m ³ , smooth wall) Fluid: single-phase water (density: 998.2 Kg/m ³ , viscosity: 0.001003 kg/m-s) Fluid velocity: 2 m/s	8	15	17	17.48	
Camçi (2003) [32]	Annulus orientation: horizontal and concentric Annulus dimensions: 0.0432 m ID, 0.123 m OD Material of construction: aluminum (smooth wall) Fluid: single-phase water Fluid velocity: 0.2 m/s	9	17	20	23	
Kelessidis et al. (2011) [33]	Annulus orientation: horizontal and concentric Annulus dimensions: 0.04 m ID, 0.07 m OD Material of construction: Plexiglas (smooth wall) Fluid: single-phase water Fluid velocity: 1.12 m/s	8	13	16	22.2	
Skudarnov et al. (2004) [34]	Pipe orientation: horizontal Pipe diameter: 0.023 m Pipe material of construction: stainless steel (wall roughness: $32 \mu m$) Fluid: two-phase solid-liquid slurry Liquid: water (density: 998.2 Kg/m ³ , viscosity: 0.001003 kg/m-s) Solid: glass spheres (double species with densities of 2490 kg/m3 and 4200 kg/m3, 50% by 50% volume mixture, particle diameter (d_m): 140 μ m, volumetric concentration (C_v): 15%) Fluid velocity: 1.724 m/s	-2	-2.8	-3	-3.2	
Ozbelge and Beyaz (2001) [35]	Annulus orientation: vertical and concentric Annulus dimensions: 0.125 m OD, 0.025 m ID Material of construction: stainless steel (density 8030 kg/m3, smooth wall) Fluid: two-phase solid-liquid slurry Liquid: water (density 998.2 Kg/m3, viscosity 0.001003 kg/m-s) Solid: feldspar (mean particle diameter 0.23 mm, mean density 2500 kg/m3, volumetric concentration, Cv = 1.80%) Fluid velocity: 0.135 m/s	-6	14	15	-15.1	
Ozbelge and Beyaz (2001) [35]	Annulus orientation: vertical and concentric Annulus dimensions: 0.125 m OD, 0.025 m ID Material of construction: stainless steel (density 8030 kg/m3, smooth wall) Fluid: two-phase solid-liquid slurry Liquid: water (density 998.2 Kg/m3, viscosity 0.001003 kg/m-s) Solid: feldspar (mean particle diameter 0.23 mm, mean density 2500 kg/m3, volumetric concentration, Cv = 1.80%) Fluid velocity: 0.197 m/s	-4	8	13	16	

TABLE 1: Analysis of turbulence model performance.



FIGURE 3: Length independence analysis (A) 75% eccentric, 150 RPM inner pipe rotation speed, and 400 kg/min flow rate; (B) concentric, stationary inner pipe, and 200 kg/min flow rate.

functions are generally applicable if the first cell center adjacent to the wall has a y^+ value larger than 30 [38]. In view of the minimum requirement ($y^+ > 30$), the value of y^+ was maintained above 45 in our study.

3.4.1. Convergence Rate Analysis. An optimum convergence rate of 10^{-5} was selected for the termination of iteration. Figure 6 shows an example of the analysis used to find out the optimum convergence rate within $10^{-6}-10^{-4}$. The simulation results varied when the convergence value was



FIGURE 4: Mesh distribution. (a) Concentric inner pipe (number of nodes 901595). (b) 25% eccentric inner pipe (number of nodes 897540). (c) 50% eccentric inner pipe (number of nodes 909730). (d) 75% eccentric inner pipe (number of nodes 919432).

greater than 10^{-5} . However, the results did not change at all for the values less than 10^{-5} .

3.5. *CFD Model Validation.* As the preliminary step, the CFD modeling approach is validated with respect to the data available in open literature. Few examples of the validation are presented as follows.

3.6. Single-Phase Flow through Annuli. Sets of experimental data from Kelessidis et al. [33] and Camçi [32] are compared in Figure 7 with the proposed CFD model. The geometry is taken from Kelessidis et al. (2011), where the inner diameter (ID) is 0.04 m, outer diameter (OD) is 0.07 m, and length is 5 m (horizontal concentric annuli). Fluid is taken as water and wall material is Plexiglas (hydro dynamically smooth wall, $\varepsilon_a = 0$). In Camçi (2003), the inner diameter is 0.0432 m,



FIGURE 5: Mesh independence analysis.



FIGURE 6: An example of optimum convergence rate analysis.



FIGURE 7: Comparison of simulated pressure gradient with experimental data from Kelessidis et al. (2011) and Camçi (2003) at different inlet velocities.

outer diameter is 0.123 m, and length is 5 m. Wall material is aluminum (smooth wall).

In Figure 7, the graph represents log-log scale. The pressure gradient increasing rate in log-log scale is almost linear for both cases. The average percentage errors of simulation results from Kelessidis et al. (2011) and Camçi (2003) are 9.88% and 8.46%, respectively, which indicates a very good agreement (it was estimated that the error of the experimental data is approximately $\pm 10\%$).

3.7. Two-Phase (Solid-Liquid) Flow through Annuli. Pressure gradient (Pa/m) profile of water-sand slurry flow through vertical concentric annuli is compared with Ozbelge and Beyaz [35] experimental data in Figure 8. For the CFD simulation, the liquid phase is considered as water (density 9982 kg/m³ and viscosity 0.001003 kg/m-s) and the solid phase is taken as feldspar (mean particle diameter 0.23 mm and mean density 2500 kg/m^3). Length 5 m, outer diameter 0.125 m, inner diameter 0.025 m, inlet velocity range of 0.0738-0.197 m/s, and overall slurry volumetric concentration range of 1.0%–1.8% with 0.23 mm grain size (d_p) are considered as boundary conditions. A smooth pipe of stainless steel (density 8030 kg/m^3) is used for the simulation. The pipe is assumed to be vertical, i.e., gravity effect is included, and gravity acceleration is directed opposite to the outlet. No slip condition for liquid and solid phases is used at the walls. Figure 9 shows the comparison of simulated and experimental two-phase frictional pressure drop through vertical annuli at different mixture velocity and at different volume concentration of slurry for 0.23 mm mean sand particle diameter (d_p) . Simulated results are in good agreement with experimental values with 2.62% average error.

4. Results and Discussion

After achieving good validation of proposed model with experimental data, a parametric analysis is done to observe the effect of the changing flow rate, drill pipe rotation, eccentricity, and particle size (detailed data tables are presented in Appendix (A and B)). The parameters used for the analysis are presented in Table 2.

4.1. Effect of Flow Rate. The effect of fluid flow on maximum bed concentration is analyzed in Figures 10–13. Water with sand particle mixture (slurry) is used as operating fluid. Four different conditions are taken into consideration with fix sand inlet concentration (20%) and sand particle size (0.1 mm). The conditions are as follows:

- (i) Concentric annuli with stationary inner pipe
- (ii) Concentric annuli with 150 rpm rotating inner pipe
- (iii) 50% eccentric annuli with stationary inner pipe
- (iv) 50% eccentric annuli with 150 rpm inner pipe

From each case of the analysis, it is observed that bed concentration near the bottom wall decreases when the flow rate increases. Because of gravitational force and horizontal orientation, sand particles have the tendency to gather near



FIGURE 8: Comparison of simulated two-phase frictional pressure gradient at different mixture velocities and volume concentration of slurry with Ozbelge and Beyaz, 2001 (0.125 m outer diameter, 0.025 m inner diameter, $d_m = 0.23$ mm) [$d_m =$ sand particle diameter and $C_v =$ sand volumetric concentrations].



FIGURE 9: Effect of inner pipe eccentricity on pressure loss at different rotational speeds.

TABLE 2: Variable and fixed parameters.

Parameters	Value/ranges	Unit
Outer diameter	0.1143	m
Inner diameter	0.0635	m
Length	5	m
Wall roughness	0	mm
Sand concentration	5-20	%
Sand particle diameter	0.1 - 0.005	mm
Inner pipe rotation	0-150	rpm
Inner pipe eccentricity	0-75	%
Water equivalent Reynolds number	$4 * 10^4 - 3 * 10^5$	Unit less

bottom wall and create a flow blockage. Decreasing flow enhances the process. From the analysis, it is found that below $9 * 10^4$ Reynolds number bed concentration near bottom wall is more than 25% in all cases and when it goes



FIGURE 10: Effect of fluid flow on particle concentration profile (concentric annuli with stationary inner pipe).



FIGURE 11: Effect of fluid flow on particle concentration profile (concentric annuli with 150 rpm rotating inner pipe).

down $4 * 10^4$, the percentage is above 50%. In Figure 14, sample contour distribution of sand particles is shown at different flow rates. Operating conditions are taken from Figure 10. Contour distribution of annuli cross section at 3 m distance from inlet displays the effect of fluid flow and gravitational force on concentration distribution clearly.

4.2. Effect of Drill Pipe Rotation and Eccentricity. Inner pipe of annuli can affect pressure loss by changing its eccentricity and rotation. The effect of inner pipe rotation and eccentricity is presented in Figures 9 and 15. Single-phase water is used as fluid in this analysis. From Figure 15, it can be visualized that pressure loss increases with the increase of rotational speed, and this trend is same at different flow rates. Stationary, 50 rpm, 100 rpm, and 150 rpm rotation of inner pipe are taken into consideration. Increasing trend is



FIGURE 12: Effect of fluid flow on particle concentration profile (50% eccentric annuli with stationary inner pipe).



FIGURE 13: Effect of fluid flow on particle concentration profile (50% eccentric annuli with 150 rpm inner pipe).

higher at high flow rates. Because of rotational speed particle-particle and particle-wall collision and rebound increase, which results in higher pressure loss.

Figure 9 shows the effect of inner pipe eccentricity on pressure loss. Concentric, 25% eccentricity, 50% eccentricity, and 75% eccentricity are analyzed in this figure. At a fixed flow rate (Re = 50000), with the increase of inner pipe eccentricity, pressure loss decreases. This trend is applicable during inner pipe stationary condition. With inner pipe rotation, the trend is opposite. Because of extra collision added by pipe rotation, this change occurs.

4.3. Effect of Particle Size. In slurry flow, sand particle size has an effective role on particle blockage near bottom wall of horizontal annular pipe. Figure 16 analyzes the effect of







FIGURE 14: Contour distribution of sand concentration profile: (a) $Re = 4 * 10^4$, (b) $Re = 7 * 10^4$, (c) $Re = 10^5$, and (d) $Re = 2 * 10^5$.

sand particle size at concentric and stationary inner pipe with 5% inlet sand concentration. With the increase of sand particle size, particle deposition near bottom wall increases because individual particle weight increases, which eventually results in this blockage. From previous analysis, we have found maximum bed concentration decreases with the increase in the flow rate (Figures 8, 10–13), but for particle size 0.005 mm, maximum bed concentration near



FIGURE 15: Effect of inner pipe rotation on pressure loss.



FIGURE 16: Effect of sand particle size on bed concentration near wall.

bottom wall is almost constant at different flow rates. That means, for smaller particle size (<0.01 mm), the effect of the flow rate on particle deposition in negligible.

To be noted, these mean particle sizes are selected from particle size distribution (PSD) charts considered during CFD analyzes. One of the PSD charts is shown in Figure 17, with mean particle size 0.1 mm, where 0.1 mm particle size is selected from eight different sizes based on cumulative weight (%) of each particles. To simplify the parametric analysis process, only mean particle sizes are shown.

5. Conclusions

In an effort to develop a widely accepted CFD model of multiphase flow through drilling annuli, the current work relates the overall plan and the initial progress of the project. In summary, this study can be recounted as follows:



FIGURE 17: Particle size distribution (PSD) of concentric and stationary inner pipe with 5% inlet sand concentration and 0.1 mm average particle size.

- (i) A CFD modeling methodology to predict frictional pressure loss and settling conditions is validated. The validation is presented with examples which demonstrate its applicability in complex drilling conditions.
- (ii) The effects of following important drilling parameters on pressure loss and settling conditions are tested: fluid flow rate, rotational speed, and eccentricity of drill pipe and solid particle size.
- (iii) Fluid flow rate has the maximum impact on particle deposition. In all analyzed conditions, with the decrease of fluid flow, particle deposition near the bottom wall increases. Specific deposition velocity depends on specific requirement of maximum deposition near wall which can be calculated using our approach.
- (iv) With the increase of rotational speed and eccentricity of inner pipe, the energy loss (pressure loss) of fluid flow increases. However, at stationary condition, pressure loss decreases with the increase of eccentricity.
- (v) Preliminary results of the current research program are presented with Newtonian fluid flow. The project is expected to produce a comprehensive CFD model capable of considering all important drilling parameters with Newtonian and non-Newtonian fluid flow. Analysis with non-Newtonian fluid flow is ongoing.

Appendix

A. Parametric Study Chart with Single-Phase Fluid

The results of the parametric study chart with the singlephase fluid are given in Table 3.

B. Parametric Study Chart with Two-Phase Fluid

The results of the parametric study chart with the two-phase fluid are given in Table 4.

TABLE 3

TABLE 3: Continued.

Cont	rolled variables	s	CFD results	Cont	rolled variables	8	CFD results
Reynolds number	Rotational	Eccentricity	Pressure	Reynolds number	Rotational	Eccentricity	Pressure
$(\text{Re} = d_{\rm H} V \rho / \mu)$	speed (rpm)	(%)	gradient (Pa/m)	$(\text{Re} = d_{\text{H}}V\rho/\mu)$	speed (rpm)	(%)	gradient (Pa/m)
20000	0	0	82	80000	50	50	644.75
40000	0	0	150	100000	50	50	965.534
50000	0	0	230	20000	100	50	203
60000	0	0	320	40000	100	50	251
80000	0	0	557.575	50000	100	50	323
100000	0	0	920.161	60000	100	50	416
20000	50	0	103	80000	100	50	689.054
40000	50	0	158	100000	100	50	1003.29
50000	50	0	234	20000	150	50	355
60000	50	0	324	40000	150	50	373
80000	50	0	562.112	50000	150	50	428
100000	50	0	923.5	60000	150	50	505
20000	100	0	165	80000	150	50	750.208
40000	100	0	209	100000	150	50	1056.07
50000	100	0	269	20000	0	75	71
60000	100	0	345	40000	0	75	135
80000	100	0	595.795	50000	0	75	215
100000	100	0	933.455	60000	0	75	310
20000	150	0	275	80000	0	75	574.076
40000	150	0	293	100000	0	75	907.722
50000	150	0	343	20000	50	75	123
60000	150	0	407	40000	50	75	197
80000	150	0	645.529	50000	50	75	288
100000	150	0	966.1	60000	50	75	395
20000	0	25	77	80000	50	75	680.247
40000	0	25	140	100000	50	75	1013.64
50000	0	25	214	20000	100	75	231
60000	0	25	299	40000	100	75	296
80000	0	25	555.578	50000	100	75	388
100000	0	25	902.474	60000	100	75	500
20000	50	25	97	80000	100	75	810 132
40000	50	25	150	100000	100	75	1166 54
50000	50	25	221	20000	150	75	401
60000	50	25	304	40000	150	75	449
80000	50	25	559 412	50000	150	75	528
100000	50	25	906 422	60000	150	75	631
20000	100	25	169	80000	150	75	948.062
40000	100	25	206	100000	150	75	1313.03
50000	100	25	259	100000	150	75	1010.00
60000	100	25	327	~ ~			-
80000	100	25	542 354	C. Description	n of Multip	phase Mod	el
100000	100	25	916 803				
20000	150	25	291	The volume of p	hase q, V_q , is	defined by	
40000	150	25	208		17	- 177	(1 1)
40000 50000	150	25	340		$V_q =$	$a_q a v$,	(AI)
60000	150	25	395	1	5		
80000	150	25	582 623	where			
10000	150	25	951 93		$\sum_{n=1}^{n}$		
20000	0	50	901.95		$\sum a_q$	= 1.	(A2)
20000	0	50	150		q=1		
40000 50000	0	50	238	The effective	density of ph	ase a is	
60000	0	50	230		actioncy of pli	use y 15	<i></i>
80000	0	50	542 620 221		$\overline{\rho}_q = a$	$a_q \rho_q$,	(A3)
10000	0	50	057.201	where a is the	nhysical dens	ity of phase a	1
20000	50	50	112	The equation	e for fluid fl	uid and gran	ular multiphasa
20000	50	50	112		is for fluid-life	uiu aiiu grafi	iulai multipilase
40000	50	50	1/8	nows are present	ea nere for th	e general cas	se of an n -phase
50000	50	50	203	flow. The volume	e traction of ea	ach phase is c	alculated from a
00000	50	50	364	continuity equation	ion as below.		

Table	4
-------	---

	Coi	ntrolled variabl	es		CED results	
Revnolds number	Rotational	Eccentricity	Particle input	Particle	Pressure	Maximum bed concentration
$(\text{Re} = d_{\rm H} V \rho / \mu)$	speed (rpm)	(%)	concentration (%)	size (mm)	gradient (Pa/m)	near bottom wall (%)
40000	Stationary	Concentric	5	0.05	766.966	58.12
70000	Stationary	Concentric	5	0.05	848.724	22.09
100000	Stationary	Concentric	5	0.05	1120.29	12.38
200000	Stationary	Concentric	5	0.05	3126.42	7.49
40000	Stationary	Concentric	20	0.05	767.134	58.14
70000	Stationary	Concentric	20	0.05	848.557	22.04
100000	Stationary	Concentric	20	0.05	1120.29	12.36
200000	Stationary	Concentric	20	0.05	3118.21	7.47
40000	Stationary	Concentric	5	0.1	767.674	58
70000	Stationary	Concentric	5	0.1	837.6	42.1
100000	Stationary	Concentric	5	0.1	1112.26	25.91
200000	Stationary	Concentric	5	0.1	3120.31	10.15
40000	Stationary	Concentric	20	0.1	761.902	52.73
70000	Stationary	Concentric	20	0.1	837.166	42.15
100000	Stationary	Concentric	20	0.1	1111.41	26.02
200000	Stationary	Concentric	20 E	0.1	5129.55 750.069	10.16
40000	Stationary	Concentric	5	0.005	212 262	5.27
10000	Stationary	Concentric	5	0.005	040.002	5.14
200000	Stationary	Concentric	5	0.005	3143.08	5.06
40000	Stationary	Concentric	20	0.005	759 924	5.00
70000	Stationary	Concentric	20	0.005	848 871	5.14
100000	Stationary	Concentric	20	0.005	1121.6	5.1
200000	Stationary	Concentric	20	0.005	3119.51	5.06
40000	150	Concentric	5	0.05	812.232	57.79
70000	150	Concentric	5	0.05	888.334	21.84
100000	150	Concentric	5	0.05	1162.98	12.33
200000	150	Concentric	5	0.05	3151.97	7.48
40000	150	Concentric	20	0.05	812.265	57.74
70000	150	Concentric	20	0.05	888.189	21.65
100000	150	Concentric	20	0.05	1162.52	12.32
200000	150	Concentric	20	0.05	3151.46	7.47
40000	150	Concentric	5	0.1	808.946	53.66
70000	150	Concentric	5	0.1	886.414	41.42
100000	150	Concentric	5	0.1	1155.53	25.81
200000	150	Concentric	5	0.1	3160.58	10.17
40000	150	Concentric	20	0.1	810.85	51.43
/0000	150	Concentric	20	0.1	886.051	41.47
200000	150	Concentric	20	0.1	1156.06	25.85
200000	150	Concentric	20	0.1	700 078	5.27
70000	150	Concentric	5	0.005	882 585	5.14
10000	150	Concentric	5	0.005	1162.34	51
200000	150	Concentric	5	0.005	3166.04	5.06
40000	150	Concentric	20	0.005	799.909	5.27
70000	150	Concentric	20	0.005	882.953	5.14
100000	150	Concentric	20	0.005	1162.38	5.1
200000	150	Concentric	20	0.005	3146.69	5.06
85000	Stationary	50	5	0.05	1042.12	33.71
100000	Stationary	50	5	0.05	1198.23	26.03
200000	Stationary	50	5	0.05	3255.59	9.96
300000	Stationary	50	5	0.05	6848.14	7.76
85000	Stationary	50	20	0.05	1055.6	33.46
100000	Stationary	50	20	0.05	1206.28	24.9
200000	Stationary	50	20	0.05	3278.31	9.9
300000	Stationary	50	20	0.05	6693.65	7.78
85000	Stationary	50	5	0.1	1011.03	26.39
200000	Stationary	50	5	0.1	1203.43	25.50
300000	Stationary	50	5	0.1	6479 87	7.71 6 83
500000	Stational y	50	5	0.1	01/2.0/	0.05

TABLE 4: Continued.

Controlled variables					CFD results	
Reynolds number	Rotational	Eccentricity	Particle input	Particle	Pressure	Maximum bed concentration
$(\text{Re} = d_{\rm H} V \rho / \mu)$	speed (rpm)	(%)	concentration (%)	size (mm)	gradient (Pa/m)	near bottom wall (%)
85000	Stationary	50	20	0.1	1016.64	30.3
100000	Stationary	50	20	0.1	1179.16	23.6
200000	Stationary	50	20	0.1	3180.33	10.62
300000	Stationary	50	20	0.1	6636.22	7.2
85000	Stationary	50	5	0.005	1036.05	5.35
100000	Stationary	50	5	0.005	1192.55	5.3
200000	Stationary	50	5	0.005	3184.09	5.15
300000	Stationary	50	5	0.005	6544.09	5.1
85000	Stationary	50	20	0.005	1035.73	5.36
100000	Stationary	50	20	0.005	1195.77	5.3
200000	Stationary	50	20	0.005	3186.63	5.15
300000	Stationary	50	20	0.005	6519.47	5.1
85000	150	50	5	0.05	1095.77	32.5
100000	150	50	5	0.05	1259.55	24.06
200000	150	50	5	0.05	3342.23	10.21
300000	150	50	5	0.05	6794.4	7.72
85000	150	50	20	0.05	1096.93	29.37
100000	150	50	20	0.05	1277.49	22.62
200000	150	50	20	0.05	3352.92	9.85
300000	150	50	20	0.05	6849.01	7.55
85000	150	50	5	0.1	1075.24	25.28
100000	150	50	5	0.1	1226.04	24.57
200000	150	50	5	0.1	3196.13	9.8
300000	150	50	5	0.1	6533.21	6.94
85000	150	50	20	0.1	1071.57	25.25
100000	150	50	20	0.1	1244.69	22.86
200000	150	50	20	0.1	3216.8	10.19
300000	150	50	20	0.1	6656.04	7.32
85000	150	50	5	0.005	1092.88	5.35
100000	150	50	5	0.005	1257.26	5.29
200000	150	50	5	0.005	3246.63	5.15
300000	150	50	5	0.005	6567.77	5.1
85000	150	50	20	0.005	1091.97	5.35
100000	150	50	20	0.005	1261.21	5.29
200000	150	50	20	0.005	3243.02	5.15
300000	150	50	20	0.005	6561.63	5.1

$$\frac{1}{\rho_{rq}} \left\{ \frac{\partial}{\partial t} \left(a_q \rho_q \right) + \nabla \left(a_q \rho_q \overrightarrow{\vartheta_q} \right) = \sum_{p=1}^n \left(\dot{m}_{pq} - \dot{m}_{qp} \right) \right\},$$
(A4)

where ρ_{rq} is the phase reference density or the volume averaged density of the q^{th} phase in the solution domain, \dot{m}_{pq} characterizes the mass transfer from the p^{th} to q^{th} phase, and \dot{m}_{qp} characterizes the mass transfer from the q^{th} to p^{th} phase.

The conservation of momentum for a fluid phase q is $\frac{\partial}{\partial t} \left(a_q \rho_q \overrightarrow{\vartheta_q} \right) + \nabla \left(a_q \rho_q \overrightarrow{\vartheta_q} \overrightarrow{\vartheta_q} \right)$

$$= -a_{q}\nabla p + \nabla \cdot \overline{\tilde{\tau}}_{q} + a_{q}\rho_{q}g + \sum_{p=1}^{\infty} \left\{ K_{pq} \left(\overrightarrow{\vartheta}_{p} - \overrightarrow{\vartheta}_{q} \right) + \dot{m}_{pq}\overrightarrow{\vartheta}_{pq} - \dot{m}_{qp}\overrightarrow{\vartheta}_{qp} \right\} + \left(\overrightarrow{F}_{q} + \overrightarrow{F}_{\text{lift},q} + \overrightarrow{F}_{vm,q} \right),$$
(A5)

where \overrightarrow{g} is the acceleration due to gravity, $\overline{\overline{\tau}}_q$ is the q^{th} phase stress-strain tensor, \overrightarrow{F}_q is an external body force, $\overrightarrow{F}_{lift,q}$ is a lift force, and $\overrightarrow{F}_{vm,q}$ is a virtual mass force. The conservation of momentum for the solid phase is

$$\frac{\partial}{\partial t} \left(a_{s} \rho_{s} \overrightarrow{\vartheta_{s}} \right) + \nabla \left(a_{s} \rho_{s} \overrightarrow{\vartheta_{s}} \overrightarrow{\vartheta_{s}} \right)$$

$$= -a_{s} \nabla p - \nabla p_{s} + \nabla \left[\overrightarrow{\tau_{s}} + \sum_{l=1}^{N} \left\{ K_{ls} \left(\overrightarrow{\vartheta_{l}} - \overrightarrow{\vartheta_{s}} \right) + \overrightarrow{m_{ls}} \overrightarrow{\vartheta_{ls}} - \overrightarrow{m_{sl}} \overrightarrow{\vartheta_{sl}} \right\} + \left(\overrightarrow{F_{s}} + \overrightarrow{F}_{\text{lift},s} + \overrightarrow{F}_{vm,s} \right),$$
(A6)

where p_s is the s^{th} solids pressure, $K_{ls} = K_{sl}$ is the momentum exchange coefficient between fluid or solid phase l and solid phase s, and N is the total number of phases.

D. Description of Turbulence Model

The exact transport equation of Reynolds stress $(R_{ij} \text{ or } u'_i u'_j)$ is as follows:

$$\frac{DR_{ij}}{Dt} = P_{ij} + D_{ij} - \varepsilon_{ij} + \emptyset_{ij} + \theta_{ij}, \qquad (A7)$$

where DR_{ii}/Dt is the summation of the changing rate of R_{ii} and transport of R_{ij} by convection. P_{ij} is the production rate of R_{ij} . D_{ij} is the diffusion transport of R_{ij} . ε_{ij} is the rate of dissipation. \mathcal{O}_{ij} is the pressure strain correlation term. θ_{ij} is the rotation term.

The diffusion term used in simulation is as follows:

$$D_{T,ij} = \frac{\partial}{\partial x_k} \left(\frac{\mu_t}{\sigma_k} \frac{\partial R_{ij}}{\partial x_k} \right), \tag{A8}$$

where $\sigma_k = 0.82$, $\mu_t = C_{\mu}(k^2/\epsilon) \text{ and } C_{\mu} = 0.09$. Production rate P_{ij} of R_{ij} or $u'_i u'_j$ can be expressed as

$$P_{ij} = -\left(\overline{u'_i u'_m} \frac{\partial U_j}{\partial x_m} + \overline{u'_j u'_m} \frac{\partial U_i}{\partial x_m}\right).$$
(A9)

The pressure-strain term \emptyset_{ij} has $\emptyset_{ij,1}$ or slow pressurestrain term also known as the return-to-isotropy term, $\emptyset_{ij,2}$ or rapid pressure-strain term, and $\emptyset_{ij,w}$ as wall-reflection term. It can be presented as

$$\emptyset_{ij} = \emptyset_{ij,1} + \emptyset_{ij,2} + \emptyset_{ij,w}, \tag{A10}$$

where

$$\emptyset_{ij,1} = -C_1 \frac{\epsilon}{k} \left[\overline{u'_i u'_j} - \frac{2}{3} \delta_{ij} k \right], \tag{A11}$$

$$\emptyset_{ij,2} = -C_2 \left[P_{ij} - \frac{2}{3} \delta_{ij} P \right], \tag{A12}$$

where $C_1 = 1.8$ and $C_2 = 0.6$.

The wall-reflection term $\mathcal{Q}_{ij,w}$ is responsible for the normal stresses distribution near the wall. This term is modeled as

$$\begin{split} \mathscr{Q}_{ij,w} &\equiv C_{1}^{\prime} \frac{\epsilon}{k} \left(\overline{u_{k}^{\prime} u_{m}^{\prime}} n_{k} n_{m} \delta_{ij} - \frac{3}{2} \overline{u_{i}^{\prime} u_{k}^{\prime}} n_{j} n_{k} - \frac{3}{2} \overline{u_{j}^{\prime} u_{k}^{\prime}} n_{i} n_{k} \right) \frac{C_{l} k^{3/2}}{\epsilon d} \\ &+ C_{2}^{\prime} \left(\mathscr{Q}_{km,2} n_{k} n_{m} \delta_{ij} - \frac{3}{2} \mathscr{Q}_{ik,2} n_{j} n_{k} - \frac{3}{2} \mathscr{Q}_{jk,2} n_{i} n_{k} \right) \frac{C_{l} k^{3/2}}{\epsilon d}, \end{split}$$

$$(A13)$$

where $C'_1 = 0.5$, $C'_2 = 0.3$, n_k is the horizontal component of the component normal to the wall, d is the shortest distance to the wall, and $C_l = C_{\mu}^{3/4}/k$, where $C_{\mu} = 0.09$ and k is the von Kármán constant (=0.4187).

The dissipation rate ϵ_{ii} or the destruction rate of R_{ii} is modeled as

$$\epsilon_{ij} = \frac{2}{3}\delta_{ij}\epsilon,$$
 (A14)

where

$$\epsilon = 2\nu \overline{s'_{ij} \cdot s'_{ij}},\tag{A15}$$

and s'_{ij} = fluctuating deformation rate.

Rotation term is expressed as

$$\theta_{ij} = -2\omega_k \Big(R_{jm} e_{ikm} + R_{im} e_{jkm} \Big), \tag{A16}$$

where ω_k = rotation vector. e_{ikm} = alternating symbol, +1, -1, or 0 depending on i, j, and k.

Nomenclature

CFD:	Computationa	l fluid d	ynamics
------	--------------	-----------	---------

- RSM: Reynolds stress model
- SST: Shear stress transport
- RANS: Reynolds averaged Navier-Stokes

SIMPLE: Semi-implicit method for pressure linked equations

- Sand particle diameter $d_{\rm m}$:
- C_{v} : Sand volumetric concentration
- ID: Pipe inner diameter
- OD: Pipe outer diameter
- Hydraulic diameter $D_{\rm H}$:
- v: Fluid velocity
- Water velocity $v_{\rm w}$:
- Sand velocity v_s :
- ε: Wall roughness
- Re: Reynolds number
- f: Friction factor.

Data Availability

The "ANSYS Simulation" data used to support the findings of this study are available from the corresponding author upon request.

Disclosure

A conference paper [39] by the same authors was presented with the primary datasets which are updated and modified in this article.

Conflicts of Interest

The authors also declare that there are no conflicts of interest regarding the publication of this paper.

Acknowledgments

The authors are thankful to the Faculty of Engineering and Applied Science of Memorial University of Newfoundland, Qatar National Research Fund (the grant NPRP10-0101-170091), Texas A&M University at Qatar, and ANSYS Inc., for helping and funding this research work.

References

- [1] A. Pilehvari and R. Serth, "Generalized hydraulic calculation method for axial flow of non-Newtonian fluids in eccentric annuli," SPE Drilling & Completion, vol. 24, no. 4, pp. 553-563, 2013.
- [2] T. Reed and A. Pilehvari, "A new model for laminar, transitional, and turbulent flow of drilling muds," in Proceedings of

SPE Production Operations Symposium, Society of Petroleum Engineers, Oklahoma, OK, USA, 1993.

- [3] O. L. Anifowoshe and S. O. Osisanya, "The effect of equivalent diameter definitions on frictional pressure loss estimation in an annulus with pipe rotation," in *Proceedings of SPE Deepwater Drilling and Completions Conference*, Society of Petroleum Engineers, Galveston, TX, USA, 2012.
- [4] R. M. Ahmed, M. S. Enfis, H. M. El Kheir, M. Laget, and A. Saasen, "The effect of drillstring rotation on equivalent circulation density: modeling and analysis of field measurements," in *Proceedings of SPE Annual Technical Conference and Exhibition*, Society of Petroleum Engineers, Florence, Italy, 2010.
- [5] I. T. Dosunmu and S. N. Shah, "Evaluation of friction factor correlations and equivalent diameter definitions for pipe and annular flow of non-Newtonian fluids," *Journal of Petroleum Science and Engineering*, vol. 109, pp. 80–86, 2013.
- [6] O. Erge, E. M. Ozbayoglu, S. Z. Miska et al., "Equivalent circulating density modeling of Yield Power Law fluids validated with CFD approach," *Journal of Petroleum Science and Engineering*, vol. 140, pp. 16–27, 2016.
- [7] S. A. Hansen and N. Sterri, "Drill pipe rotation effects on frictional pressure losses in slim annuli," in *Proceedings of SPE Annual Technical Conference and Exhibition*, Society of Petroleum Engineers, Dallas, TX, USA, October 1995.
- [8] M. Sorgun, "Computational fluid dynamics modeling of pipe eccentricity effect on flow characteristics of Newtonian and non-Newtonian fluids," *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, vol. 33, no. 12, pp. 1196–1208, 2011.
- [9] M. Sorgun and M. Ozbayoglu, "Predicting frictional pressure loss during horizontal drilling for non-Newtonian fluids," *Energy Sources Part A-Recovery Utilization and Environmental Effects*, vol. 33, no. 11, pp. 1104–1106, 2011.
- [10] R. Subramanian and J. Azar, "Experimental study on friction pressure drop for nonnewtonian drilling fluids in pipe and annular flow," in *Proceedings of International Oil and Gas Conference and Exhibition in China*, Society of Petroleum Engineers, Beijing, China, 2000.
- [11] G. Camçi and T. A. Özbelge, "Determination of dilute slurry densities in a vertical annulus using isokinetic sampling," *Chemical Engineering Communications*, vol. 193, no. 11, pp. 1482–1501, 2007.
- [12] S. K. Dewangan and S. L. Sinha, "Exploring the hole cleaning parameters of horizontal wellbore using two-phase Eulerian CFD approach," *Journal of Computational Multiphase Flows*, vol. 8, no. 1, pp. 15–39, 2016.
- [13] T. A. Özbelge and A. N. Eraslan, "A computational hydrodynamic and heat transfer study in turbulent up-flows of dilute slurries through a concentric annulus," *Turkish Journal* of Engineering and Environmental Sciences, vol. 30, no. 1, pp. 1–13, 2006.
- [14] T. A. Özbelge and G. C. Ünal, "A new correlation for twophase pressure drops in fully developed dilute slurry up-flows through an annulus," *Chemical Engineering Communications*, vol. 196, no. 4, pp. 491–498, 2008.
- [15] T. N. Ofei, "Effect of yield power law fluid rheological properties on cuttings transport in eccentric horizontal narrow annulus," *Journal of Fluids*, pp. 1–10, 2016.
- [16] M. Sorgun and E. Ulker, "Modeling and experimental study of solid-liquid two-phase pressure drop in horizontal wellbores with pipe rotation," *Journal of Energy Resources Technology-Transactions of the ASME*, vol. 138, no. 2, 2016.

- [17] X. Sun, K. Wang, T. Yan, S. Shao, and J. Jiao, "Effect of drillpipe rotation on cuttings transport using computational fluid dynamics (CFD) in complex structure wells," *Journal of Petroleum Exploration and Production Technology*, vol. 4, no. 3, pp. 255–261, 2014.
- [18] B. J. Alder and T. E. Wainwright, "Studies in molecular dynamics. II. Behavior of a small number of elastic spheres," *Journal of Chemical Physics*, vol. 33, no. 5, pp. 1439–1451, 1960.
- [19] S. Chapman and T. G. Cowling, The Mathematical Theory of Non-Uniform Gases: An Account of the Kinetic Theory of Viscosity, Thermal Conduction and Diffusion in Gases, Cambridge University Press, Cambridge, UK, 1970.
- [20] M. Syamlal, W. Rogers, and T. J. O'Brien, *MFIX documentation: Theory guide*, National Energy Technology Laboratory, Department of Energy, Technical Note DOE/METC-95/1013 and NTIS/DE95000031, 1993.
- [21] T. B. Anderson and R. Jackson, "Fluid mechanical description of fluidized beds. Equations of motion," *Industrial & Engineering Chemistry Fundamentals*, vol. 6, no. 4, pp. 527–539, 1967.
- [22] R. M. Bowen, "Theory of mixtures," *Continuum Physics*, vol. 3, part I, 1976.
- [23] S. Patankar, Numerical Heat Transfer and Fluid Flow, CRC Press, Boca Raton, FL, USA, 1980.
- [24] S. Vasquez and V. Ivanov, "A phase coupled method for solving multiphase problems on unstructured meshes," in *Proceedings of ASME 2000 Fluids Engineering Division Summer Meeting (ASME FEDSM'00), paper FEDSM2000-*11281, pp. 1–6, Boston, MA, USA, 2000.
- [25] J. M. Weiss, J. P. Maruszewski, and W. A. Smith, "Implicit solution of preconditioned Navier-Stokes equations using algebraic multigrid," *AIAA Journal*, vol. 37, no. 1, pp. 29–36, 1999.
- [26] T. Barth and D. Jespersen, "The design and application of upwind schemes on unstructured meshes," in *Proceedings of* 27th Aerospace Sciences Meeting American Institute of Aeronautics and Astronautics, Reno, NV, USA, 1989.
- [27] M. M. Gibson and B. E. Launder, "Ground effects on pressure fluctuations in the atmospheric boundary layer," *Journal of Fluid Mechanics*, vol. 86, no. 3, pp. 491–511, 2006.
- [28] B. E. Launder, G. J. Reece, and W. Rodi, "Progress in the development of a Reynolds-stress turbulence closure," *Journal* of Fluid Mechanics, vol. 68, no. 03, pp. 537–566, 2006.
- [29] B. E. Launder, "Second-moment closure and its use in modelling turbulent industrial flows," *International Journal for Numerical Methods in Fluids*, vol. 9, no. 8, pp. 963–985, 1989.
- [30] A. J. Chorin, "Numerical solution of the Navier-Stokes equations," *Mathematics of Computation*, vol. 22, no. 104, 745 pages, 1968.
- [31] D. R. Kaushal, K. Sato, T. Toyota, K. Funatsu, and Y. Tomita, "Effect of particle size distribution on pressure drop and concentration profile in pipeline flow of highly concentrated slurry," *International Journal of Multiphase Flow*, vol. 31, no. 7, pp. 809–823, 2005.
- [32] G. Camçı, Application of isokinetic sampling technique for local solid densities in upward liquid-solid flows through an annulus, METU, PhD dissertation, 2003.
- [33] V. C. Kelessidis, P. Dalamarinis, and R. Maglione, "Experimental study and predictions of pressure losses of fluids modeled as Herschel-Bulkley in concentric and eccentric annuli in laminar, transitional and turbulent flows," *Journal of Petroleum Science and Engineering*, vol. 77, no. 3, pp. 305–312, 2011.

- [34] P. V. Skudarnov, C. X. Lin, and M. A. Ebadian, "Doublespecies slurry flow in a horizontal pipeline," *Journal of Fluids Engineering*, vol. 126, no. 1, pp. 125–132, 2004.
- [35] T. A. Özbelge and A. Beyaz, "Dilute solid-liquid upward flows through a vertical annulus in a closed loop system," *International Journal of Multiphase Flow*, vol. 27, no. 4, pp. 737–752, 2001.
- [36] N. P. Brown and N. I. Heywood, Slurry Handling: Design of Solid-Liquid Systems, Springer Science & Business Media, Berlin, Germany, 1991.
- [37] E. J. Wasp, J. P. Kenny, and R. L. Gandhi, "Solid--liquid flow: slurry pipeline transportation. [Pumps, valves, mechanical equipment, economics]," *Bulk Material Handling*, vol. 1, no. 4, 1977.
- [38] ANSYS, 12.0/12.1 Documentation. Users Guide Manual, Ansys Inc., Canonsburg, PA, USA, 2009.
- [39] S. Rushd, R. A. Sultan, A. Rahman, and V. Kelessidis, "Investigation of pressure losses in eccentric inclined annuli," in *Proceedings of ASME 2017 36th International Conference on Ocean, Offshore and Arctic Engineering*, American Society of Mechanical Engineers, Trondheim, Norway, 2017.

