VOL. 11, NO. 16, AUGUST 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

EVALUATION OF PRESSURE LOSSES DUE TO SOLIDS-IN-LIQUID PIPE FLOW BY CFD SIMULATION

Hussain H. Al-Kayiem, Tang M. Huong and Javed A. Khan
Mechanical Engineering Department, Universiti Teknologi Petronas, Bandar Seri Iskandar, Tronoh, Perak, Malaysia
E-Mail: hussain_kayiem@petronas.com.my

ABSTRACT

One of the most important elements of liquids transportation process, mainly the crudes and oil products, is the pipelines flow assurance by maintaining lowest pressure losses. Existence of two phase flow in the pipe would generate different pressure drop than the designed drop under single phase flow assumption. This paper presents CFD simulation results of solid-in-liquid, i.e. slurry horizontal pipe flow using ANSYS-CFX software. The influencing of sand particles' diameter and concentration on the pressure loss of pipelines at various flow rates of sand-in-Diesel 2D was studied. Three cases have been investigated; single liquid flow, homogeneous slurry sand-in-liquid flow, and two layers slurry flow, suspended layer in the upper and dead bed in the bottom. For validation, water flow was also simulated. Water and Diesel 2D were assumed Newtonian, incompressible and the region of simulation was fully developed. Mesh independency study was conducted so that the produced results would not be affected by the number of element. Fluid flow was simulated as single phase flow with velocities of 0.5, 1.0, 1.5 and 2.0 m/s. Then, particles were dispersed in the flow with volumetric concentrations of 10%, 15% and 20%. The investigated particles' sizes were 0.25, 0.50 and 1.00 mm. The simulation procedure was validated through comparison of the pressure drop and friction factor results with the well-established methods in the literature. Analysis of the homogeneous solid-in-liquid flow results demonstrates an increase in the pressure drop. In the case of two-phase two-layer flow, the pressure drop increases dramatically due to the high shear between the upper and lower layers leading to very high resistance at the interface surface between the stationary bed and the upper homogenous layer. The simulation explores interesting phenomena of particle settling due to the high shear at the interface surface resulting in creation of moving bed between the layers. Further investigations will enhance the understanding of the multi layers slurry flow phenomena.

Keywords: pressure drop in slurry flow, two-phase flow, two layer pipe flow, slurry flow.

INTRODUCTION

Nowadays, one of the most important elements in the oil industry is the pipelines. Pipelines are used to transport oil, natural gas, slurry and others. During the process of transportation, the sand which is contained in the slurry may deposit on the bottom of the horizontal pipelines. As time passes, it will form a layer of stationary sand deposit which is the sand bed. The existence of sand bed will bring effects on the rate of production as it will cause a high pressure drop in the pipelines. Hence, this subject must be considered and included during the pipelines design stage.

The presence of sand in oil is unavoidable and has brought serious problems in pipe flow assurance. It can affect the smoothness of the oil flow and also possess risks in damaging the pipelines, as well as non-estimated pumping power. Therefore, the presence of sand through oil pipe is one of main concerns in oil and gas industry. According to Kaushal and Tomita [1], slurry pipeline design parameters include solid concentration profiles, pressure drop and deposition velocity. Faitli [2] mentioned that pressure loss is one of the main parameters in pipeline designing as it will determine the most suitable pump to be used to transport the oil and this is one of the most expensive parts.

However, there are also few factors that will affect the pressure loss and among them are the size (diameter) of the solid particles and its concentration. These two parameters affect the pressure loss in pipes as they will cause friction losses. According to Matousek [3], friction is divided into two types which are the mechanical friction and also viscous friction. The mechanical friction is created through the contact between the solid particles and the pipe wall and it can be permanent or random. The viscous friction in case of particle-in-liquid flow is created due to the formation of solid particles in the near wall layer of carrying liquid as addition to the fluid viscous friction losses. In this case, the properties of the carrying liquid near the wall layer are changed.

Friction loss in solid-in-liquid flows depends highly on the flow conditions and also the pattern of the flow. Matousek [3] categorized the flow pattern into four: fully stratified flow, fully suspended flow, non-stratified flow and partially-stratified flow. In fully suspended flow, the solid particles are uniformly distributed across the pipe and no solid will deposit or act against the pipe wall. As for non-stratified flow, a small concentration gradient will appear across the pipeline but no contact bed is present.

Moreover, El-Nahhas *et al* [4] divided slurry into two main types which are the settling and non-settling. The settling slurry will have a range of flow patterns and these flow patterns are relying on the physical properties of the carrier fluid and also the transported solids, the velocity and the concentration of slurry. When reaching one point, the solid particle will form a gravity bed which is called sliding bed or stationary bed. This kind of flow pattern is commonly known as partially-stratified flow and it may be assumed as a pseudo-homogeneous flow. In addition, there are two conditions which are in between settling flow and partially-stratified flow: saltation and heterogeneous flow regimes. They also commented that non-settling slurries has a homogeneous flow pattern where the solid particles will settle very slowly and will distribute equally throughout the pipe. It was also indicated that the increase in solid particle content will cause the mixture to no longer be regarded as two separate components. The resulting fluid might possess non-Newtonian characteristics which signify a more complicated situation. In this situation, the solid particle concentration is higher in the bottom of the pipe which also indicates a settling pattern. This settling pattern will cause higher frictional losses compared to homogeneous slurries.

Kaushal *et al.* [5] mentioned that most of the previous studies on slurry pipeline systems deals with moderate solid concentrations up to 26%. In their study, the increase in solid particle concentration will cause the pressure drop to increase in any flow velocity. They mentioned that the smaller particle size will have lower pressure drop at lower velocities and higher pressure drop at higher velocities compared to bigger sized particle. They also explained that the increase in pressure drop for bigger particle size at low velocity is because of the increase in particle amount moving in the bed is due to gravity. Furthermore, Nabil *et al* [6] claimed that the bigger sized particle needs more power to compensate the energy loss and the difference between the pressure drops of various particle sizes decreases as the slurry velocities increases.

Generally speaking, the literature lacks a comprehensive coverage of the pressure drop in solids-in-liquid flows. The case is related to huge industrial application and financial investment. More investigations are required to assist in understanding the pressure drop in solid-in-liquid pipe flow, and in proper design and operation of pumping systems. In particular, the size and the concentration of the solid particles have been emphasized to be supplementary investigated, for pressure drop prediction of solid-in-liquid pipe flow.

Ali [7] was the first researcher who studied and analyzed the cutting transport parameters using CFD. He has applied hole cleaning simulation using the Discrete Phase Modeling (DPM) in FLUENT and conducted the analysis for horizontal and vertical wells. He judged the qualities of the hole cleaning through the transport efficiency. The effect of the mud flow rate, mud weight, mud viscosity, drilling rate, cutting size and cutting density were analyzed. Relative deviation between the model prediction and the experimental data was observed at high velocities, and this was related to difference of the cuttings sizes tested in each case. Ali's results indicated the importance of the annular velocity in cuttings removal. He also reported that horizontal well cleaning was better than for vertical wells.

It was eventually noticed that some of the researchers' findings obtained through CFD did not closely correspond to observable reality since the simulation results do not match either physical facts or valid experimental results; e.g. [Waltson [8], Li and Walker [9], and Ramadan [10]].

Another CFD technique used to investigate steady state cuttings transport in horizontal and deviated wells was employed by Mishra [11]. Instead of DPM, Eulerian Mixture Modeling capabilities in FLUENT software were used in the study. This study considered parameters of the fluid flow rate, ROP, angle of inclination, drill-pipe rotation and cutting size. The results indicated that fluid flow rate, angle of inclination, and ROP have a major impact on the cutting concentration. Furthermore, they recorded that larger particles were more efficiently cleaned, and pipe rotation would greatly enhance the hole cleaning, especially for the smaller sizes.

Once again, the CFD results were noted to be in conflict with the experimental and analytical works, where it has been repeatedly demonstrated that smaller particles are indeed easier to clean, as noted in [Walker and Li [12], Kamp and Reviro [13], Martins *et al*. [14]. This is especially in the cases where pipe rotation is involved. Even so, we can compare Mishra's results to the claims of Wilson and Judge on 1978, who have also claimed that smaller particles are harder to clean than larger one. The particle sizes used in the study by Mishra were within the given range of the easiest removable sizes cited in [Walker and Li [12].

The present study is aimed to investigate the effect of the solid particle's diameter and concentration on the pressure loss of horizontal pipeline flow. The diameter and concentration of solid particles and the fluid velocities are among the factors that influencing the pressure losses. CFD technique is adopted using ANSYS-CFX software to simulate three different cases of pipe flow including single phase flow, homogenous solid-in-liquid two phase flow and two layers two phase flow. All cases have been simulated as steady flow. The pressure drop is predicted in each case. Oil and water are considered as Newtonian, incompressible fluids and the flow is steady and fully developed with no chemical reaction between the solid and liquid phases. The values of the research parameters have been adopted from industrial application for liquids transportation, mainly the oil transport in Malaysia.

PROBLEM FORMULATION AND SOLUTION METHODS

Two analysis techniques have been adopted in the present investigation; namely, analytical analysis and numerical analysis by CFD technique. The analytical analysis is carried out through the use of the well-established correlations for fluid flow and pressure drop predictions. The CFD technique is carried out through modeling of horizontal pipe flow with suitable boundary conditions to identify the influence of the parameters of the solid-in-liquid flow. The solid particle, as commonly

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

present in the oil production and transportation, are sand particles.

Three types of flow have been considered in this study.

- Single phase, water and Diesel 2D flow, is used for validation of the simulation and also as bench mark to compare with the two phase flow case.
- Homogenous two phase flow of solids-in-water and solids-in-Diesel 2D with various concentrations of sand particle.
- Two phase two layer flow consisting of homogenous flow at the upper part of the pipe and dead bed of sand particles at the lower part of the pipe.

The flow cases are presented schematically in Figure-1.

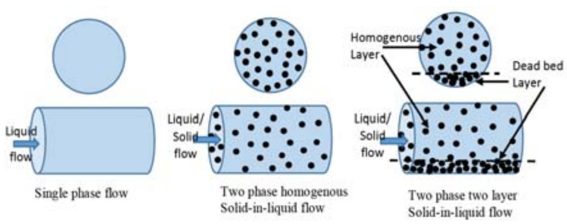


Figure-1. Investigated flow cases; (i) Single phase flow, (ii) Homogenous slurry, (iii) Two layer two phase slurry.

The variables considered in the study and the range of their variations are summarized in Table-1.

Table-1. Ranges of investigation variables, 0.2 m pipe diameter.

variable		units	Range of variation			
Flow rate	As oil production	bbl/d	8538	17075	25607	34145
	As oil velocity	m/s	0.5	1.0	1.5	2.0
Sand concentration (by volume)			10%, 15%, and 20%			
Particle size		mm	0.25, 0.50, and 1.00			
Dead bed height		m	0.25D			

The commonly used pipes for crudes and oil products transportation are 0.2 m (8 Inch) diameter and hence it is adopted in the present work. The simulated pipe segment is 20 m long. It is assumed as part of the fully developed region. The pipe is made of Galvanized Iron which has surface roughness of 1.5x10^{-4} m. The material properties adopted in the study are shown in Table-2.

Table-2. Material properties.

parameter	unit	value
Density of Water	kg/m^3	998.2
Viscosity of Water	kg/m.s	1.002 x 10^{-3}
Density of Oil Diesel 2D	kg/m^3	849.0
Viscosity of Oil	kg/m.s	0.0041
Density of Sand	kg/m^3	2500.0

a) Analytical procedure

In order to calculate the pressure drop in pipelines, the well-established Darcy-Weisbach equation has been used, as:

$$\Delta p = f \cdot \frac{L}{D} \cdot \frac{\rho u^2}{2} \qquad (1)$$

where, Δp (Pa/m) is the pressure drop, f is the friction factor, L and D (m) are the length and diameter of the pipe, respectively, ρ (kg/m^3) is the density of fluid and u (m/s) is the mean velocity of the flow predicted from the flow rate over the cross sectional area of the pipe.

To calculate the friction factor, the Colebrook equation (Rennels and Hudsen [15]) is used since the flow is turbulent:

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{\varepsilon}{3.7D} + \frac{2.51}{Re\sqrt{f}}\right) \qquad (2)$$

(3)

- If the solid in mixture concentration is known by volume, then

$$\rho_m = [C_v\rho_s + (1 - C_v)\rho_L]/100 \qquad (4)$$

where: ρ_m, ρ_s, and ρ_L are the densities of mixture, solid and liquid, respectively, in kg/m^3; C_w and C_v are the solid phase concentration by weight and volume %, respectively.

The dynamic viscosity of the solid-liquid mixture with concentration is calculated using equation 5 as recommended by Menon [16]

$$\mu_m = \mu_L\,[1 + 2.5\alpha + 10.05\alpha^2 + 0.00273\exp(16.6\alpha)] \qquad (5)$$

where, μ_m and μ_L are the viscosity of mixture and liquid, respectively, and α is the volume fraction of solids in mixture, ($\alpha = C_v/100$).

Then, the friction factor and pressure drop are predicted by the same procedure of the single phase using equations 1 and 2.

b) Numerical procedure

The flow geometry is modelled with pipe length of 20 m and diameter of 0.2 m. The pipe has three boundaries to be identified which are inlet, outlet and wall.

Boundary conditions includes pressure at inlet, flow rate at outlet and no slip conditions in the inner wall of the pipe where the fluid is viscous, Newtonian. The mesh is generated first by default. According to Team [17], the accuracy of the mesh and boundary conditions depends on the accuracy of the converged solution. The default mesh is subjected to mesh independency check. Six different settings of the mesh have been conducted and the six setups and the obtained outcomes are shown in table-3. The mesh, coded 03 in Table-3, has been selected since it compromises reasonable computational time and accuracy, with aspect ratio of 5.4 and skewness of 0.133.

Table-3. The resulted parameters of the various mesh size.

Meshing	Mesh 01	Mesh 02	Mesh 03	Mesh 04	Mesh 05	Mesh 06
Nodes	19840	310545	**416639**	520539	571641	706410
Elements	16588	295924	**396400**	495500	543500	676500
Element Quality	0.897	0.180	**0.220**	0.315	0.292	0.248
Aspect Ratio	1.611	5.942	**5.443**	4.360	4.448	4.963
Jacobian Ratio	1.540	1.207	**1.236**	1.239	1.217	1.195
Skewness	0.205	0.121	**0.133**	0.133	0.139	0.120

Based on the quality concept depicted in Table-3, the decision on the mesh type 3 is made where it has provided good compromise of 5.443 aspect ratio, 396400 elements, and skewness of 0.133. In addition, the prediction accuracy of the pressure gradient, as index parameter, is carried out as in figure-2 and the residual RMS error values are reduced to less than 10^{-5} at steady state iteration. The percentage of difference with respect to mesh type 4 is only 0.1% Pa/m.

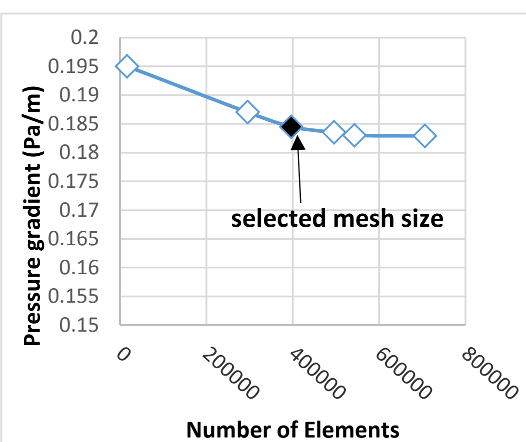


Figure-2. Selection of mesh size.

The resulted meshed configuration of the pipe is shown in Figure-3.

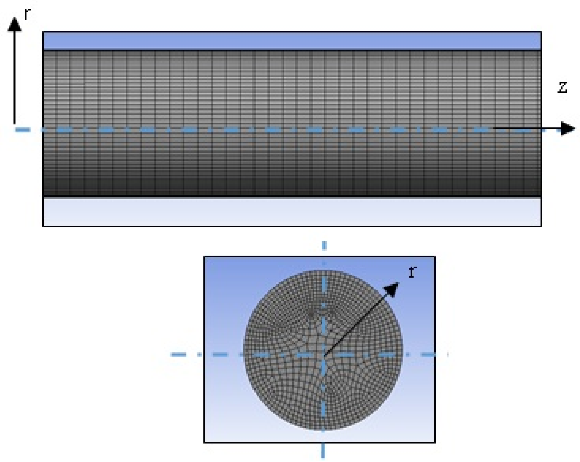


Figure-3. Final mesh configuration of the pipe segment.

RESULTS AND DISCUSSIONS

To ensure good quality results with least error, the simulation procedure should be validated. After that, the results are presented for two-phase homogenous one layer flow, and two phase two layer flow.

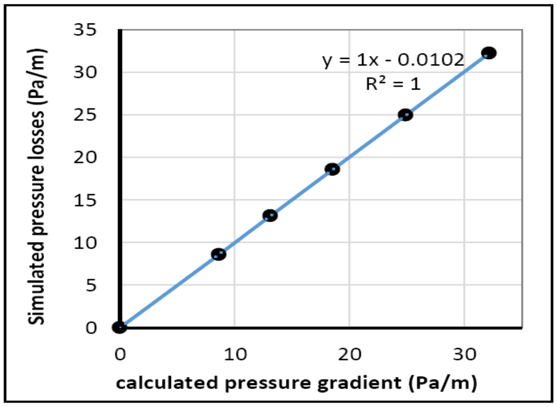
a) Validation of the simulation procedure

The purpose of conducting the verification and validation of simulation model is to ensure that the produced results are accurate with minor percentage error. The pressure drop, for the case of water flow and for Diesel 2D flow, are predicted using the design equations 1 to 5, and then extracted from the simulation. Figure-4 a and b show the results obtained at velocity range from 0.3 m/s to 0.8 m/s for water and Diesel, respectively. The comparison demonstrate good agreement between the simulation and calculation results. The line angle is almost 45^o in both cases of water and Diesel oil and the fitness parameter, R^2 is equal unity. It is found that the margin of error is reducing as the flow rate is increasing. At velocity of 0. 3 m/s, the percentage of error is 0.24 and 3.9 for water and oil, respectively; while at flow velocity of 0.8 m/s, it becomes only 0.03 and 0.1 for water and Diesel flow, respectively.

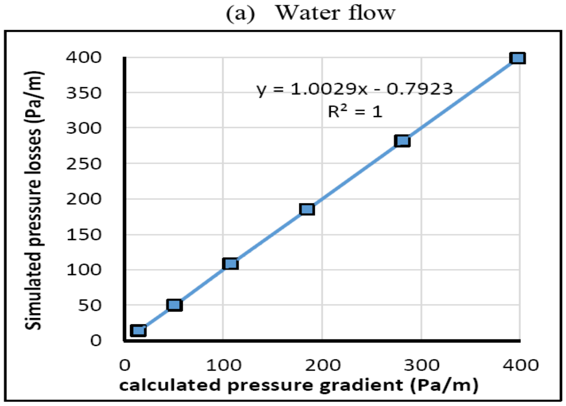
The mean percentage of error, of the pressure gradient, over the entire flow rate range is 0.115% and 1.1% for water and oil, respectively. With this small differences and high R^2 between the simulation and prediction results, the numerical procedure is considered valid and is able to predict the other cases of the research.

VOL. 11, NO. 16, AUGUST 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com



(a) Water flow



(b) Diesel 2D flow

Figure-4. Validation of the simulated pressure drop results by comparison with theoretically predicted pressure drop results for, (a) Water and (b) Diesel flows.

b) Results homogeneous slurry flow

To explore the influence of the particles parameters in terms of the particle size, and particle concentrations, the ranges of parameters investigated in the present work are shown previously in Table-1.

1) Effect of particle size

To analyze the contribution of the sand particle size on the frictional loses, the simulation has been repeated for the case of 10% solid concentration for flow velocity of 0.5, 1.0, 1.5, and 2.0 m/s. The simulation results of the friction factor and the simulated pressure gradient are shown in table-4. The values are all the same for particles sizes of 0.25, 0.5, and 1.0 mm. These findings are similar to those reported by Li *et al* [18], who concluded that in drilling cutting transport, the particle diameter has a very small effect in the transport performance and the build-up of the dead bed. In general, during looking on several studies on the transport of solid particles-in-liquid, it was perceived that it is difficult to underline either small or large particles are easier to be removed. Kelin *et al* [19] advocated large cuttings transport is mainly driven by the flow rate, while fluid rheology is the key factor in transport of small cuttings. Duan *et al* [20] suggested that difficulty to transport small cuttings occurs where low viscous fluids were in use. Some studies have focus on the transport efficiency of small and large size particles, (e.g. Ford *et al* [22] Walker and Li [12] Duan *et al* [23]) but not on how the particle size influence the pressure losses. This is quite interesting to be further investigated, and the equations used to calculate the mixture density and viscosity should be modified to count for the particles size.

However, comparison between the friction factor values of pure Diesel and homogeneous Diesel/solids are showing considerable difference. The mean increase in the friction factor values over the tested range of $9x10^3 < Re < 3.6x10^4$ is about 15%. The increase in the pressure drop about 28.7%.

Table-4. Simulation results of the friction factor and the pressure drop and their increase due to the flow velocity, at homogenous sand-in-Diesel slurry, with 10% concentration.

Velocity (m/s)	Re	Friction factor, *f*		Relative % of increase	ΔP/ΔL (Pa/m)		Relative % of increase
		Diesel 2D	slurry		Diesel 2D	slurry	
0.5	9219	0.02726	0.031	12.06	13.895	19.5	28.7
1.0	18438	0.02402	0.028	14.3	49.814	70.2	29.0
1.5	27657	0.02259	0.027	16.3	107.770	150.2	28.2
2.0	36876	0.02176	0.026	16.3	184.644	260.2	29.0

2) Effect of solid concentration

Solid concentration of 10%, 15% and 20% is considered in the present simulation. The simulation results of the friction factor and the pressure drop, assuming homogenous slurry, are shown in figures-5 and figure-6, respectively. To inspect the escalation in the frictional losses due to various concentrations of solid, the

ARPN Journal of Engineering and Applied Sciences

simulation results of the single phase flow as Diesel 2D are inserted in the figures. The results show that the concentration of sand particles and fluid velocity will cause effect on the pressure drop while the size of sand particles has no significant effect on the flow pressure drop. Moreover, the friction factor in the flow decreases as the Reynolds Number increases.

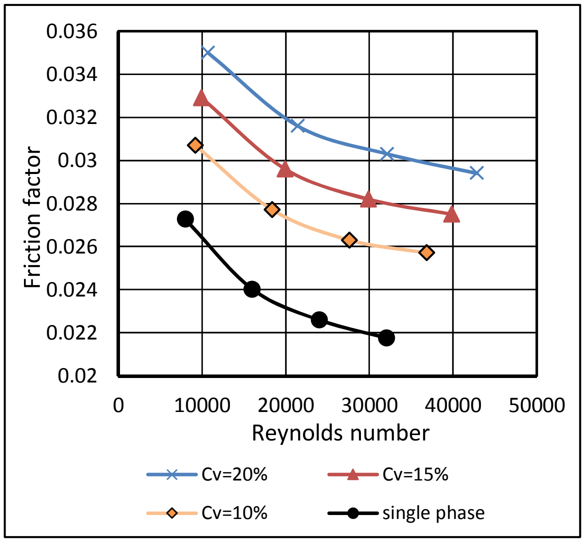


Figure-5. Simulation results of the friction factor of single-phase and various concentration homogenous slurry of Diesel 2D flow, at various velocities.

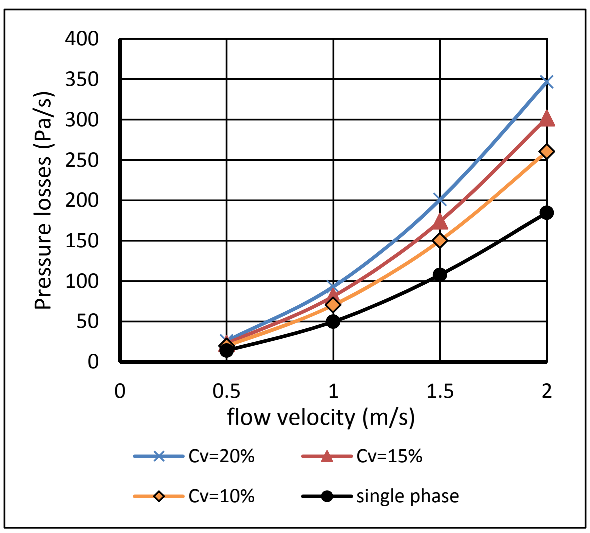


Figure-6. Simulation results of pressure losses of various concentration, homogenous sand-in-Diesel 2D slurry flow, at various velocities.

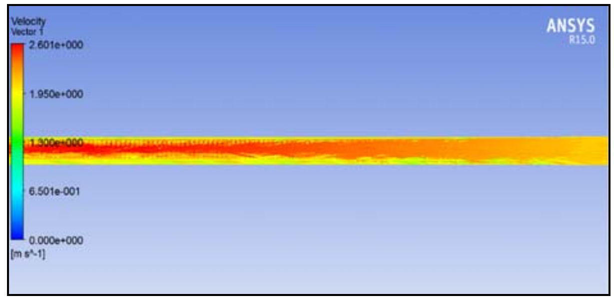


Figure-7. Velocity contour of homogenous sand-in-diesel 2D slurry flow.

c) Two-phase two-layer slurry flow

For the case of two phase two layer flow, the upper layer is treated as in the case of homogenous slurry flow, but the lower contact surface is treated exceptionally. The interface between the suspended upper layer and the dead lower bed is considered as very rough surface and the highest possible roughness is input to identify the interface surface. ANSYS software has the capability to predict the friction factor of such interface. The values obtained from the procedure are in good agreement with the suggestion of [Ramadan *et al* [10] who assumed the friction factor between the suspended layer flow and the dead bed is within 0.25 to 0.7. However, the cross section of the flow field becomes a semicircle where flow is assumed to have zero velocity. The simulation of the two phase two layer slurry is shown in Figure 7.

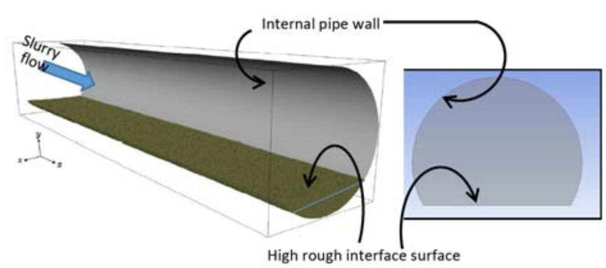


Figure-8. CFD model of two-phase two-layer flow of sand-in-Diesel slurry.

Comparing the pressure drop results of the two-phase two-layer with the single phase and with homogenous cases demonstrates that the presence of the dead bed is highly increasing the pressure drop. The simulation results, as in table-5, revealed that presence of solid-in-liquid slurry influence the pressure drop, which means considerable larger pumping power. The presence of high rough interface surface between the layers contributes considerably in the high pressure losses.

The increase of the pressure drop due to the solid concentration is very consistent at 1.0 and 2.0 m/s velocities. When all the particles are suspended and dispersed homogenously, the increase in the pressure loss is 29.0%, 38.7%, 46.5% for 10%, 15% and 20% concentrations, respectively for both cases of 1.0 and 2.0 m/s slurry velocities. The presence of the dead bed causes

VOL. 11, NO. 16, AUGUST 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

the losses to jump very high compared to the single phase flow, as 82.0%, 85.0% and 87.0% for concentration of 10%, 15% and 20%, respectively for both cases of 1.0 and 2.0 m/s slurry velocities. In spite that the interface area between the layers is small compared to the pipe wall contact area with slurry, but it increases the pressure losses almost double time compared to the homogenous slurry case.

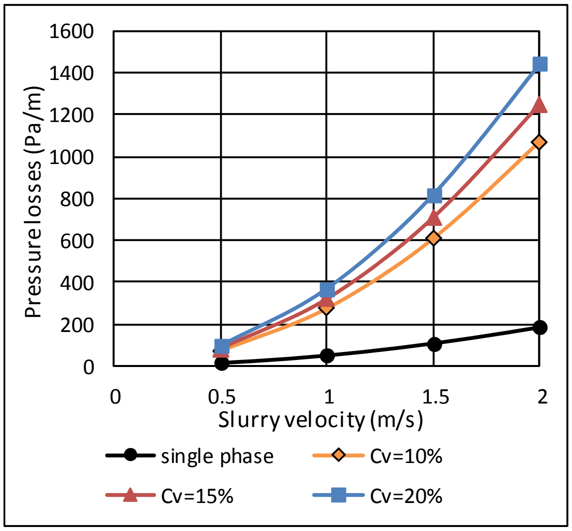


Figure-9. Pressure losses per pipe length for the case of two-phase two-layer flow at various slurry velocities. D_p = 0.25 mm.

The powerful of CFD appears here as provides a support to the justification of this phenomena. In the simulation, it is assumed that the bed layer is completely stilled. Behavior of the suspended flow near to the bed may tend to be slower, that could return alternatively laminar flow patterns. That is evident in the velocity contours in figure-10, in which, the dead bed is excluded and the lower boundary is representing the highly rough interface surface. It could be noted that in the region close to the inlet of the simulated segment, all the cross section is homogenous, but a new layer is created and enlarge in size as proceeding downstream. This is believed to be a moving bed build up between the dead bed and the suspended layer.

Built up of the moving-bed layer is an indication for the particles transfer from the stationary-bed to the suspended layer and vice versa. Some particles tend to settle and transfer from core of the suspended layer to the zero velocity region. They decelerate during their travel from the slurry velocity to the zero velocity. This travel distance is representing the moving bed. As such, there are two interface surfaces, the first is between the suspended layer and moving bed, and second between the moving bed and the dead bed. This is why the pressure losses are increasing to high values. El-Nahhas *et al* [4] realized this settling pattern and he also concluded that it will cause higher frictional losses compared to homogeneous slurries. The velocity contours are clearly demonstrating that the moving bed is enlarging as moving downstream away from the inlet. The suspended layer is reduced in size and the slurry velocity of the suspended layer is increasing, while the very slow region close to the dead bed surface is increasing

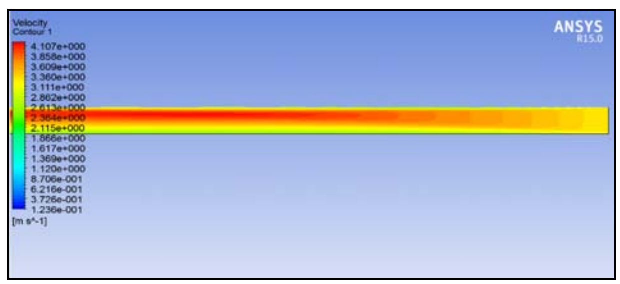


Figure-10. The velocity contour in the case of two-phase two-layer of sand in diesel slurry flow.

Table-5. Comparison between the simulation pressure drop results of single, homogenous and two-phase two-layer slurry flows of sand-in-Diesel 2D.

Mode of slurry	Concentration (C_v %)	Pressure drop at 1 m/s velocity (Pa/m)	Percentage of losses increase (%)	Pressure drop at 2 m/s velocity (Pa/m)	Percentage of losses increase (%)
Single phase of Diesel 2D	zero	49.814	-	184.644	-
Homogenous slurry	10	70.2	28.8	260.2	29.0
	15	81.3	38.7	301.8	38.8
	20	93.1	46.5	346.4	46.7
Two-phase two-layer slurry	10	275.6	81.9	1073.6	82.8
	15	321.8	84.5	1252.4	85.2
	20	370.8	86.6	1444.4	87.2

CONCLUSIONS

The single and two phase, solid-in-liquid of water and oil are simulated and validated via comparison with the available design equation for friction and pressure losses in pipe flow. Three flow cases of sand-in-Diesel 2D have been simulated and analyzed in terms of pressure losses prediction and comparison. The homogenous slurry and two-phase two-layer slurry with 10.0, 15.0 and 20.0 C_v % have been considered, as well as different sand particle sizes have been investigated. It could be concluded that:

The size of the sand particles do not have any significant effect on the pressure drop in all two-phase flow case.

In two phase flow, the increase in sand concentration and the increase in the flow velocity will cause increase in the flow pressure drop. This becomes more effective in the case of two-phase two-layer flow.

Through visualization of the velocity contours, produced from the CFD simulation, it has been noted that a third layer is created between the dead bed and the suspended slurry. This moving layer has low velocity and it increases in size as proceeding with the flow to the end of the pipe.

There are some recommendations and improvements can be made in this project which are:

- Investigating the effect of pipe diameter and length on the pressure loss.
- An experiment can be set up to compare the actual results with the simulated results.

The project can be continued with Non-Newtonian fluids.

ACKNOWLEDGEMENTS

The authors acknowledge Universiti Teknologi PETRONAS for the logistic and financial support to conduct the present research work.

REFERENCES

[1] Kaushal, D. R., & Tomita Y. (2002). Solids concentration profiles and pressure drop in pipeline flow of multi sized particulate slurries. International Journal of Multiphase Flow 28, 1697–1717.

[2] Faitli, J., Pressure loss calculation model for well-graded solid-liquid pipe flows on the basis of systematic pilot plant investigations. Materials Technology 64, pp2-7. 2012.

[3] Matousek, V. (2002). Pressure drops and flow patterns in sand-mixture pipes. Experimental Thermal and Fluid Science 26, 693–702.

[4] El-Nahhas, K., El-Hak, N. G., Rayan, M.A., & El-Sawaf, I. (2009). Effect of particle size distribution on the hydraulic transport of settling slurries. Thirteenth International Water Technology Conference, IWTC13, (pp. 463-474). Hurghada, Egypt.

[5] Kaushal, D. R., Sato, K., Toyota, T., Funatsu, K., & Tomita, Y. (2005). Effect of particle size distribution on pressure drop and concentration profile in pipeline flow of highly concentrated slurry. International Journal of Multiphase Flow 31, 809–823.

[6] Nabil, T., El-Sawaf, I., & El-Nahhas, K. (2013). Computational fluid dynamics simulation of the solid-liquid slurry flow in a pipeline. Seventeenth International Water Technology Conference, IWTC 17. Istanbul.

[7] Ali, M. W., A parametric study of cutting transport in vertical and horizontal well using computational fluid dynamics (CFD), M.S. thesis, Dept. of Petroleum and Natural Gas Eng., WVU, Morgantown, West Virginia, pp. 108, 2002.

[8] Walton, I. C., Computer Simulator of Coiled Tubing Wellbore Cleanouts in Deviated Wells Recommends Optimum Pump Rate and Fluid Viscosity," SPE Paper 29491-MS, presented at SPE Production Operations Symposium, 2-4 April 1995, Oklahoma City, Oklahoma, 1995.

[9] Li, J. and Walker, S., Sensitivity Analysis of Hole Cleaning Parameters in Directional Wells, SPE Journa Paper 74710-PA , vol. 6, pp. 356-363; 356, 2001.

[10] Ramadan, A., Skalle, P. and Johansen, S. T., A mechanistic model to determine the critical flow velocity required to initiate the movement of spherical bed particles in inclined channels, Chemical Engineering Science, vol. 58, pp. 2153-2163, 5. 2003.

[11] Mishra, N., Investigation of hole cleaning parameters using computational fluid dynamics in horizontal and deviated wells, M.S. thesis, Dept. of Petroleum and Natural Gas Eng., WVU, Morgantown, West Virginia, pp. 75, 2007.

[12] Walker, S. and Li, J., The Effects of Particle Size, Fluid Rheology, and Pipe Eccentricity on Cuttings Transport," SPE Paper 60755-MS, presented at SPE/ICoTA Coiled Tubing Roundtable, 5-6 April 2000, Houston, Texas, 2000.

[13] Kamp, A. M. and Reviro, M., Layer modeling for cuttings transport in highly inclined wellbores, SPE Paper 53942-MS, presented at Latin American and Caribbean Petroleum Engineering Conference, Caracas, Venezuela, 1999.

[14] Martine, A. L., Sa, C. H. M., Lourenco, A. M. F. and Campos, W., Optimizing Cuttings Circulation In Horizontal Well Drilling," SPE Paper 35341-MS, presented at International Petroleum Conference and

Exhibition of Mexico, 5-7 March 1996, Villahermosa, Mexico, 1996.

[15] Rennel, D. C. and Hudson, H.M., Pipe flow: A practical and comprehensive guide. AIChe, Wiely, New Jersey, 2012.

[16] Menon, E. S., Piping Calculations Manual, McGraw Hill, New York, 2005.

[17] Team, L. C. (2012, January 17). Tips & Tricks: Convergence and Mesh Independence Study. Retrieved February 12, 2015, from LEAP's Computational Fluid Dynamics (CFD) Blog: http://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/

[18] Li, Y., Bjorndalen, N., and Kuru, E., Numerical modelling of cuttings transport in horizontal wells using conventional drilling fluids. Journal of Canadian Petroleum Technology, 47(7). DOI: 10.2118/07-07-TN. 2007.

[19] Kelin, W., Tie, Y., Xiaofeng, S. and Shizhu, L., review and analysis of cuttings transferrin complex structural wells, Open Fuels and Energy Science Journal, 6, 2013.

[20] Duan, M., Miska, S. Z., Yu, M., Takach, N. E., Ahmed, R. M. and Hallman, J. H., Experimental Study and Modeling of Cuttings Transport Using Foam With Drill pipe Rotation, SPE Paper 116300-MS, presented at SPE Annual Technical Conference and Exhibition, 21-24 September 2008, Denver, Colorado, USA, 2008.

[23] Ford, J. T., Peden, J. M., Oyeneyin, M. B., Gao, E. and Zarrough, R., Experimental investigation of drilled cuttings transport in inclined boreholes, SPE Paper 20421-MS Presented at SPE Annual Technical Conference and Exhibition, New Orleans, Louisiana, 1990.