UDC 532
DOI: 10.15587/1729-4061.2022.254214

IDENTIFYING THE MECHANISM OF THE MIXED-PHASE FLOW IN THE HORIZONTAL PIPELINE USING COMPUTATIONAL FLUID DYNAMIC APPROACH

Ashham Mohammed Aned
Corresponding author
Lecturer*
E-mail: ashham.m@mtu.edu.iq
Saddam Hussein Raheemah
Senior Lecturer*
Kareem Idan Fadheel
Senior Lecturer*
*Department of Mechanical Technical/Production
Al kut Technical Institution
Middle Technical University
Al-Za'franiya, Baghdad, Iraq, 10074

In this study, computational analysis has been carried out using computational fluid dynamics (CFD). These calculations have been made to investigate the rheological behavior of the mixed-phase flow in horizontal pipelines. In order to study the shear stress in a vertical pipe, a new numerical model for oil-water dispersion in three dimensions has been developed. CFD software has been used to study the wall shear stress function and water droplet pressure. Using Reynolds numbers and the Navier-Stokes equations with k–turbulence factor to save energy, the flow range for the continuous process was explained. The results from a recent study on experimental methodology were simulated. In this study, the diameter of the tube is 40 mm and the length is 3.5 m and modeled and analyzed using Ansys software. Thus, the geometry has been imported and modeled using the CFD tool. The meshed model has been tested and converged accordingly. The primary data of the simulation have been verified with experimental results successfully. Oil droplet widths have previously been thought to be dependent on the flow Reynolds number, which was confirmed in this case study. Droplet diameter Dd was measured at 6 mm while the mixture moved at a speed of 1.9 m/s. It was found that the largest shear stress value was found at the top of the pipe, where the oil fraction (cut-off) was 0.3, in the simulation results for varied velocities (1.6, 2.5, 2.9 m/s) and oil fraction (cut-off) values. The results of the simulation analysis of the two-phase flow of crude oil for the horizontal pipe are wall shear stresses with different velocities for crude oil in the two-phase flow. As well as pressure drop at different velocities for the same fluids

Keywords: Iraqi crude oil, CFD, FEM, wall shear stress, pressure drop

Received date 15.02.2022
Accepted date 28.03.2022
Published date 28.04.2022

How to Cite: *Aned, A. M., Raheemah, S. H., Fadheel, K. I. (2022). Identifying the mechanism of the mixed-phase flow in the horizontal pipeline using computational fluid dynamic approach. Eastern-European Journal of Enterprise Technologies, 2 (7 (116)), 44–50. doi: https://doi.org/10.15587/1729-4061.2022.254214*

1. Introduction

Two phases of the pipe flow dispersion are usually employed in the petroleum sector including petroleum droplets [1]. The dispersion during oil extraction is also called harmful since it improves the efficiency of the fluid viscosity in the device resulting in higher pressure decreases. Emulsions, though, create issues with the oil-water isolation. However, the strain might be lowered as oil is dispersed in water in some conditions, such as with highly viscous oils, which leads to a product similar to water than raw content [2]. During treatment as good corrosion measures by steel pipe oil, oil continuous dispersions may be chosen.

Various types of raw oils have been the subject of several rheological research [3]. Sulfur content, density, and average specific gravity (API) were the most important factors in assessing petroleum (S).

Petroleum's organic system explains the physical and chemical processes that lead to the creation of crude oil. Carboxylic and phenolic acids are two examples of equatorial additives. Chemical bases, and metal complexes. The chemical composition must be understood in advance of the operation of every natural substance. The physicochemical characteristics that are important for the function and management of crude oil include oxidation, strength, adhesive quality, adsorption pressure, and solubility. Indeed one of the several significant factors in Arctic materials is their physicochemical behavior. Polar compounds are considered by way of fluorine and silicone coating acids, chemical bases, and metal clusters. For any organic acid, naphthenic acid is popular [4].

The viscosity of the emulsion is greatly impacted by the physical stage. The effect on the conductance of the shear level is due to the strength of the dispersed point. The above activity is attributed to particle swarming or systemic viscosity. In reality, part of the water volume in the spread stage affects the viscosity of the emulsion greatly. An expanding percentage of water volume raises the viscosity of the emulsion. Hydrogen interactions and hydrostatic forces influence the water volume component mainly. In order to determine methods of oil processing and transport, the analysis is considered important for the fluid influence of petroleum emulsions in rheology [5].

With a thickness of 10 cm in the 18 m long Plexiglass tunnel. [6] studied the impact on the stream pattern of the oil-water flow orientation, oil viscosity, and mixing rate, as well as on the working volume of natural ASTM marine water. Three different flow behaviors have been identified, almost separated and semi-mixed.

In a depressed state, crude oil also has low water content. Dynamic existence means it has to deal with many obstacles in various processes, such as extraction and distribution. Because of substantial hydrogen and acid emissions in powerful fuel products, their transit in pipes is adversely impacted. Two-step oil and water source segregation is also common in the oil business, as seen by the final segregation stage in the wells [7]. Water from the pipeline may be combined with gas in a tank to boost the amount of oil produced. When oil and water have densities that are too close to be separated, it might be difficult to tell the two apart. Carbon dioxide (CO_2) and sulfur dioxide (SO_2), which are highly corrosive to carbohydrate pipes, can be dissolved during the water bath (H_2S). Due to the rise in capital and operational expenditures, these two issues contribute to a decrease in production efficiency. The activities and features of the two-phase oil-water flow must therefore be taken into account in the design and operation of wells, storage facilities, and distribution pipes in the oil and gas industry. In the last few decades, a number of studies have sought to detect and evaluate in two stages the complex oil-water flow dynamics. Exam objectives will include any of its major findings in the subject of two-stage oil-water flow [8].

Streaming behavior is significantly influenced by the presence of water [9]. As a result, a thorough examination of its properties (such as schedules and drop sizes), as well as hydrodynamic characteristics (such as phase stoppage and pressure change), can be helpful [10]. A wide range of factors influencing flow dispersion are examined in experiments.

The usage of 3D computational fluid dynamics (3D-CFD) codes to secure nuclear reactors has gained publicity in the last thirty years. Consequently, the usage of codes such as downsizing, counter-current stream decreases and pressure-reduced thermal shock (PTS) for the prediction of horizontal two-phase flow processes is being researched gradually. A novel parameter in the reactor protection domain is the accessibility of 3D circumstantial data in such a process [11]. Experimental and geometric rules that are a precondition in single-dimensional number codes could be replaced by considerably more central closing rules in 3D-CFD. The Euler-Lagrange method included in the ANSYS (C FDS) tool assumes that the description of the discrete process by global particles that possess mean diameters during a constant fluid phase is expected to be more stable than the one-dimensional codes [12]. CFD therefore might not actually be geometry-dependent. Owing to oil droplets scattered in water, their globularity can also be assumed to be retained in the computations and no break-up or merger is observed [13].

Several numerical studies have been undertaken in the investigation of shear stress and pressure drop for crude oils [14]. These investigations cooperate in solving issues regarding two- or more-phase flow.

Therefore, the study of the numerical analysis was carried out in accordance with two primary factors. Its goal is to predict a decrease in pressure for a two-phase crude oil mixture, as well as shear stress in pipelines, in advance of the event.

2. Literature review and problem statement

As gas and liquid flow rates increase, the topologies of two-phase flows comprising gases and liquids can change substantially. A bubble column chemical reactor, refrigeration, drying, and electronic cooling systems can all benefit from the use of heat exchangers (such as BWRs, boilers, and steam generators in traditional plants). When designing these systems, a greater grasp of two-phase flow is necessary. Understanding these fluxes is a difficult task because of the numerous length scales and complex physics involved. To ensure the safe operation and growth of nuclear reactors, two-phase flow parameters, such as void distribution, must be accurately predicted in safety studies. CFD simulations utilizing a two-phase flow model can provide precise, three-dimensional flow behavior for improved system design and performance [15].

Because of technology improvements, engineers and scientists who utilize computational fluid dynamics (CFD) to investigate and predict different types of flows now have more tools at their disposal [16]. CFD models were used in a large number of studies because they are capable of properly responding to changes in crucial flow parameters throughout time and space [17]. Due to recent developments in CFD algorithms, single-phase flow problems can still be dealt with certainty. Due to technical restrictions, Direct Numerical Simulation (DNS) cannot be used to solve large-scale industrial two-phase flow issues. The computationally intensive nature of DNS makes this particularly true for gas-liquid flows characterized by the high number of bubbles of various sizes. When it comes to two-phase flow dynamics, CFD-based two-fluid modeling should be used in industrial manufacturing in the same manner that single-phase modeling is used when it comes to one-phase flow dynamics.

Although the two-fluid model provides a reasonable compromise between accuracy and computation time, considerable constitutive modeling is required to account for the time averaging that is required for the model [18]. Interfacial move requirements and interfacial transferring terms define the transfer of mass, momentum, and energy through the interface when the transmission is averaged over time. Predictions based on two-fluid models are highly dependent on the correctness and robustness of the constitutive models that were used. The multiphase models utilized in CFD methods are currently in development. For CFD to be utilized for fuel gas with the same degree of assurance as single-phase flows, advanced analytical modeling, as well as experimental study, is required.

Depending on the concentration of interfacial area, the equations for contact transfer are dependent on the total mixture surface area (IAC). The two-fluid model requires accurate modeling of the IAC in order to adequately analyze the two-phase flow. A two-phase empirical or quasi correlation and an algebraic set of methods for the flow regime have traditionally been used to close IAC. However, there are a number of drawbacks in this approach [19]. As an alternative to static flow regimes, the Inter-facial Area Transport Equation (IATE) was put out as an analytical tool that might better account for the dynamics of interfacial structure development [20].

A computational fluid dynamics model of an orifice plate opening is used to simulate the flow of thermal two-phase fluid (CFD). To predict pipeline pressure before and after a sharp-edged orifice plate, CFD modeling was employed. This claim was substantiated by data gathered from the field. The model was used to predict the flow rate (for steam and liquid), as well as the pressure drop and velocity distribution, in a pipeline with just an eccentric aperture. When it comes to geothermal wells, a two-phase mixture of steam, which is characterized in terms of mass flow rate and enthalpy, is normal [21]. In addition to daily field management, developers must adhere to legally agreed-upon fluid take requirements, which must be followed. Continuous metrics monitoring is required for the early diagnosis of technical defects that are closely related to one another. Because of this, appropriate treatments can be designed and implemented as early as possible [22]. The use of a centralized separator for steam field design facilities in a modern geothermal power station is less expensive, easier to operate, and requires less maintenance than using a distributed separator. Due to the fact that the mass flow rates and enthalpy of all of the production wells are averaged together in the central separator, it is necessary to monitor the mass flow rates and enthalpy of each production well. The use of a sharp-edged orifice plate as an alternate method of measuring individual well productivity is common among geothermal producers. In geothermal field testing conducted in New South Wales, Indonesia, and the Philippines, a concentric sharp-edge orifice plate was used to analyze existing two-phase flowmeter correlations, which were previously published [23].

Always check to see if a well will self-discharge before taking any measurements. Different methods can be used to anticipate a geothermal well's discharge capability, and stimulation techniques can be used to discharge (jump-start) a well if necessary. A down-hole pump can be used to create pressurized liquid phase thermal brine if a well cannot be emptied after stimulation. Pump profiles, single-phase opening or Venturi meter, and the recorded liquid temperature can be used to compute the single-phase flow velocity [24].

Gas-liquid two-phase flows are capable of producing a wide range of bubble shapes and sizes, which can be regulated by flow parameters such as gas and liquid flow rates, system pressure and channel geometries. Depending on the bubble's size and shape, as well as the bubbles interaction processes, interfacial forces can have a significant impact on the bubble transport properties (drag, lift, etc.). For this reason, IATE should account for these variances in bubble diameters while formulating. One of the most common types of gas-liquid two-phase flow bubbles is the churn-turbulent bubble. There are two basic groups of transport features for small, spherical and nearly spherical (distorted) bubbles, which include the cap, the slug (Taylor), and the churn-turbulent bubbles. Consequently, a two-group was created to compensate for low void percentages in bubbly streams with bubbles that are almost spherical in shape. Two sets of equations are used to cover the fundamental gas-liquid different transportation phenomena under a wide variety of flow circumstances. This necessitated the development of a two-group formulation that could handle both large and small bubbles in the two-phase flow. Detailed information on the two different flows will be provided in the next section.

Current research is focusing on studying and analyzing the rheological behavior of crude oil in terms of pressure drop and wall shear stress.

3. The aim and objectives of the study

The primary aim of the study is to investigate the rheological characteristics of two-phase crude oil flow using computational fluid dynamics.

To achieve this aim, the following objectives are accomplished:

– to calculate Drop Ratio in Oil-to-Water Mixtures;

– to investigate the influence of the Wall Shear Stress for Mixture Phases.

4. Materials and methods of research

4. 1. Validation Process

It is totally dependent on the previous study, which was carried out with the help of the ANSYS FLUENT 15 simulation software and was confirmed by experimental data [25], for the current analysis relating to Canadian crude oil. Because of this, constraints will be employed as major limits in the ongoing study, and new restriction requirements will be incorporated into the process. It has been decided to use ANSYS FLUENT 15 to model new boundary environments.

4. 2. Primary Boundary Conditions

The numerical analysis in this study was based on experiments on the flow of oil and water in an erect tube with an inner diameter of p=40 mm, which were carried out on the same tube. A rough blender was given on its entrance to the premise oil and water in order to minimize the time it took for the flow to become fully dispersed. For this experiment, the researchers employed municipal water as a solvent and the chemical D145 as a reagent. Using these parameters, the temperature was set to 25 degrees Celsius, the density and viscosity of water and oil were set to 998 kg/m^3 (water) and 827 kg/m^3 (oil), and the oil-water interfacial tension was set to 0.94310 Pa·s (water) and 5.03 Pa·s (oil), respectively (oil). This research has utilized the finite elements method (FEM) in order to examine the hexagonal element type. The pipeline's gravitational pull and substance have been overlooked.

4. 3. Rheological Model

Crude oil is the focus of the current study (two-phase water flow). In this case, the water is flowing in two directions simultaneously through a pipe with a diameter of 40 mm and a length of 3.5 m. There have always been the same presumptions made. The region was discredited with around 20,000 computer cells. The Euler-Lagrange method requires that the cells be larger than droplets. The simulation phase was carried out for homographic oil droplets with sizes Dd=1, 2, 3.5, 3.6, 3.9, 4.2 mm. The average fuel/water mixture is Um=2.5, 3.0, 3.5 m/s; the input fuel is 0.25, 0.6, 0.8, 1.0 m/s.

4. 4. New Boundary Condition for Current Study

The current project makes use of crude oil, as well as water. This mixture consists primarily of a liquid water phase with varying fraction (cut) ratios. Liquid-liquid two-phase flow is formed by the two of them. It is found that crude oil has the following characteristics, according to [26]: viscosity – 2.99 mPa·s, density – 0.801 g/ml, also a tension of 31 mN/m exists between oil and water at the interfacial level.

4. 5. Mesh and Geometry

AutoCAD was used to design the pipeline and grid. Because of the hexahedral mesh's ability to deliver better solutions with fewer cells, geometry has been adopted [27]. Meshed models were used in this study, as shown in Fig. 1.

The geometry has been imported and modeled successfully. In this research, the mesh has been generated using the Ansys CFD tool. The meshed model has been tested and converged for the further process accordingly.

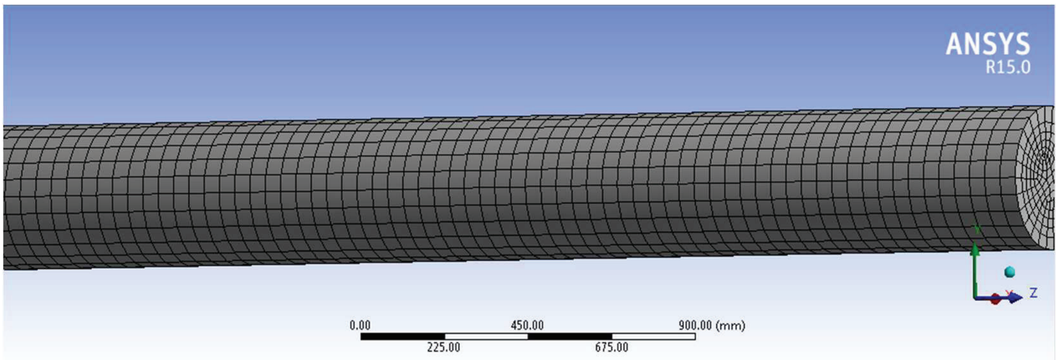

Fig. 1. Geometry and mesh

4. 6. Mesh Independent Test for Current Study

In order to solve a punctual flow, a grid independent research was undertaken in order to acquire an appropriate mesh density. According to the results, the velocity scales linearly with the number of components. There were 15,194 components in the oil-water blend, with an average speed of Um=1.1 m/s. Grid-independent solutions are unaffected by grid refinement, resulting in a stable solution. There is no difference in rpm when the number of elements reaches 17,284 and 18,185 (Fig. 2).

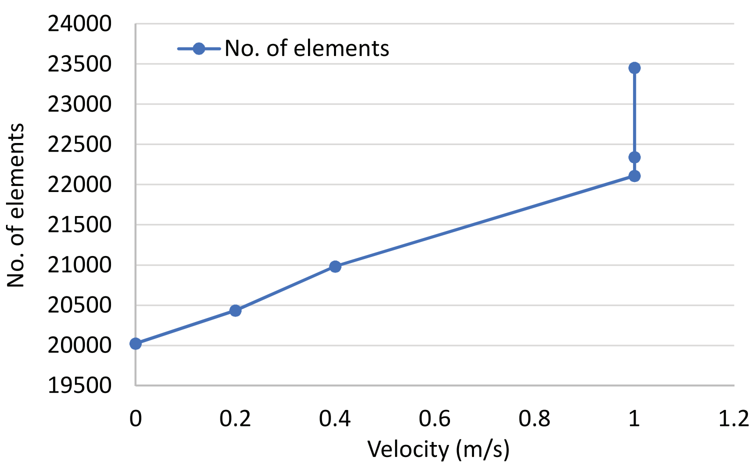

Fig. 2. Grid evaluation test

In this study, the mesh has been converged and set up accordingly. At 1 m/sec, the changes in the number of the elements had no effect. Thus, at a certain velocity, the quantity of the elements was performed successfully.

5. Results of the computational investigation of the mixed-phase flow in the horizontal pipeline

5. 1. Drop Ratio in Oil-to-Water Mixtures

Current experiments in this analysis were regarded on the basis that an average speed of 1.5, 2.0 and 2.5 m/s for the oil-water combination was checked for the results of a leveling factor followed by a shear-lifting power portion operation in the cumulative balance of force in the central equation. The modeling findings for each unit length (i.e. pressurization pipe incline) are contrasted by Fig. 3, which displays simulation projections where the CL coefficient was set to 0.5 and the oil content input is 0.2. The contrast in the existing flow environment shows that the proportions expected for pressure declines are higher than the predicted proportions of the pressure drops in the analysis owing to the computational simulation of the present CL=0.5 coefficient. When the diameter Dd is roughly 5 mm, CL=0.5 was the best fit for the experimental outcomes.

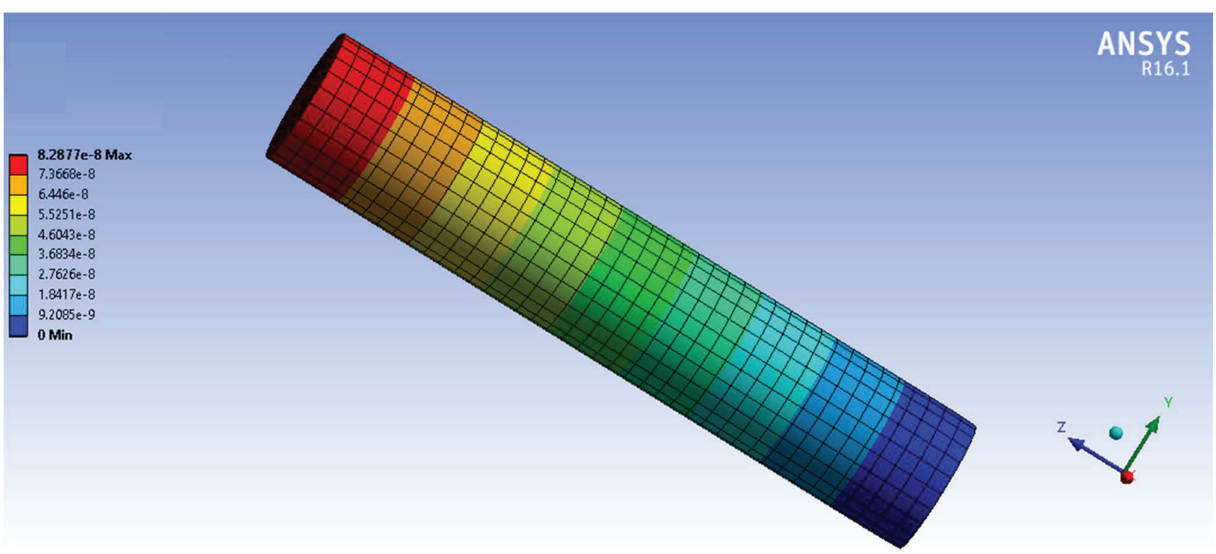

Fig. 3. Pressure drop simulation in the water and mixed phases (oil and water)

Fig. 4, 5 showed a moderate under-projected atmosphere, subject to a *CL*=0.5 coefficient. A more precise estimation of the decrease in pressure ratio is always obtained, whether the average speed of the flow is higher, based on the analysis (greater Reynolds number). Such a result may be related to the fact that the Euler-Lagrange method is used (discrete phase model).

Fig. 4 showed the effect of the pressure drop on the velocity to the fraction diameters. Wherein this study, two percent of oil fractions were considered to be analyzed. Fig. 5 showed the velocity of the input fraction. The results stated that the maximum velocity occurred at maximum input fraction and pressure drop as well.

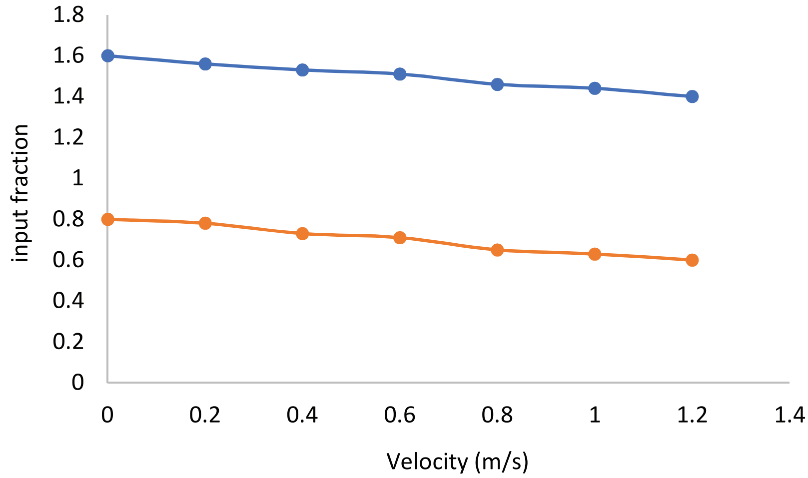

Fig. 5. Pressure drop to the particular diameter for each unit length based on the input fraction

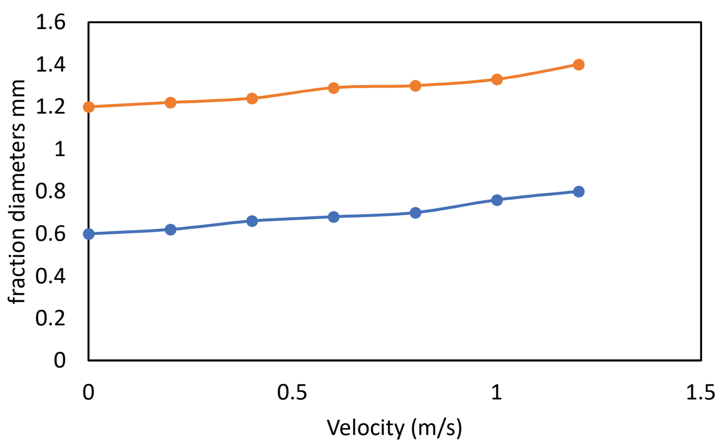

Fig. 4. Pressure drop to the particular diameter for each unit length

5. 2. Wall shear stress for mixture phases

An investigation into how shear stress affects the pipe's radial structure is needed. A data collection line was established at the pipe's opening to capture the effects of simulations on the pipe's performance as shown in Fig. 6, 7. Because this oil fraction has strong results, the 'Shear Tension scenario' where the average flow rate *Um*=2.7 m/s is 0.4 for the input oil and *Dd*=6 for the oil. With only a slight hint of increased oil droplet strength along the tube's wall, the shear stress curve depicts a pressure that is rather uniform across the pipe's diameter. Gravity has been factored into the current study.

Wall shear stresses have been obtained from numerical analyses accordingly. Fig. 7 showed the relation between the velocity and the wall shear stress in terms of oil fraction. The results stated that the maximum wall shear stress happened at the maximum velocity.

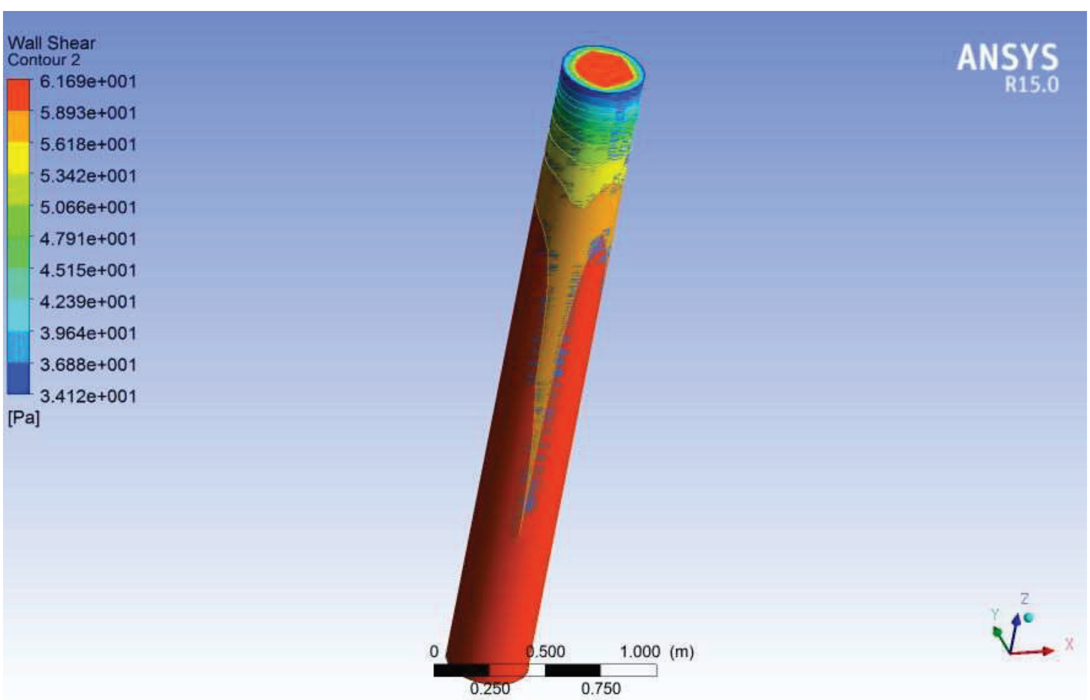

Fig. 6. Shear stress simulation in a two-phase mixture

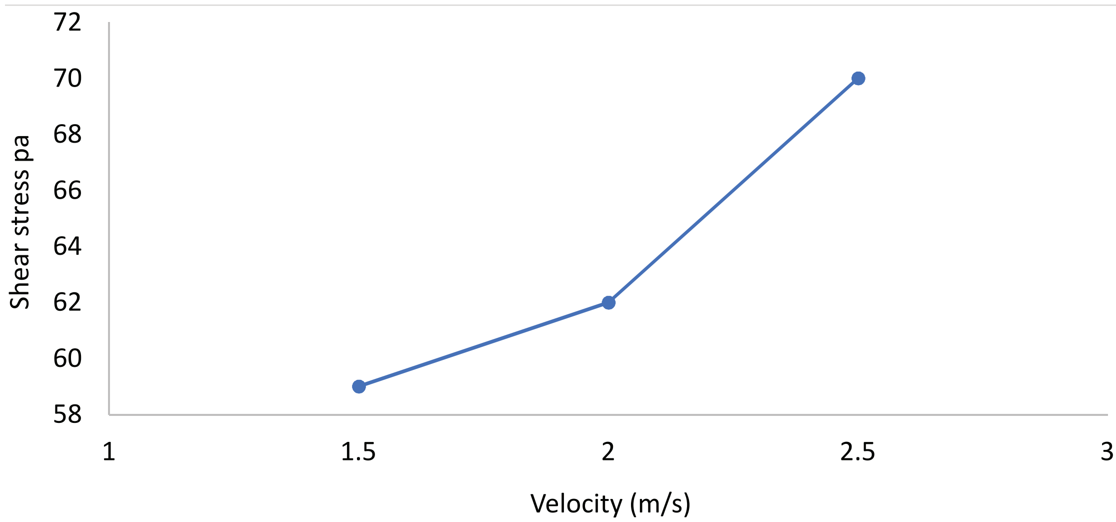

Fig. 7. Oil fraction shear stress-velocity relationship

6. Discussion of the computational investigation of the mixed-phase flow in the horizontal pipeline

In this research, the computational fluid dynamics (CFD) approach based on the finite element method has been used to predict the data of the pressure drops and wall shear stresses. Fig. 3 showed the effect of the pressure drop ratio through the motion of the fluid. The findings have shown that the maximum value of the wall shear stress reached 6.1 MPa due to the motion of the fluent. Fig. 6 shows the graphical distribution of the shear stress of the inner surface of the pipeline.

The results of this study have been compared to those of earlier investigations. In contrast to [28], the comparison has been made in terms of technique and results, respectively. The wall shear stress, as well as the pressure drop ratio, was taken into account in these results. The methods of investigation and the results of the numerical analysis were taken into consideration.

The investigation of the influence of the pressure due to motion and explanation of the distribution of the shear stress effect on the inner part of the pipeline. These analyses specify and reveal where the maximum shear stress and pressure drop can happen in order to solve them accordingly.

This study is limited to certain kinds of crude oil as defined in the boundary condition in the case of the two-phase flow. As well as only two main parameters have been analyzed and investigated and these parameters are pressure drop and wall shear stresses with respect to velocity.

Several disadvantages can be noticed and can be eliminated in the future, and the main disadvantages are the accuracy of the data of crude oil and mesh setup.

In this research, some difficulties have been encountered and the main issue was how to get accurate data on crude oil. Another difficulty in the simulation process is in defining data in the library of the program.

7. Conclusions

1. The pressure drop ratio of the mixed flow with a certain diameter *Dd* predicted to be 4 mm has been calculated. Numerical results have provided better predictions of pressure drops when the mixture is moving at a faster speed, this analysis and numerical simulation show a significant relationship. It has been proven that shear stress in the pipeline portion of 2 and 4 m/s is 0.4.

2. Wall shear stress due to the velocity of the fluids has been calculated for the mixed flow of crude oil. Numerical results have predicted that the greater shear stress value was found at the pipe's upper end and was 70 Pa.

References

1. Burlutskiy, E., Turangan, C. K. (2015). A computational fluid dynamics study on oil-in-water dispersion in vertical pipe flows. Chemical Engineering Research and Design, 93, 48–54. doi: https://doi.org/10.1016/j.cherd.2014.05.020
2. Hu, B., Angeli, P. (2008). Phase Inversion and Associated Phenomena in Oil-Water Vertical Pipeline Flow. The Canadian Journal of Chemical Engineering, 84 (1), 94–107. doi: https://doi.org/10.1002/cjce.5450840113
3. Saleh, Z. S., Sheikholeslami, R., Watkinson, A. P. (2005). Fouling Characteristics of a Light Australian Crude Oil. Heat Transfer Engineering, 26 (1), 15–22. doi: https://doi.org/10.1080/01457630590890049
4. Abdul Jalil, N. A.., Kadhim Sharaf, H.., Salman, S. (2017). A Simulation on the Effect of Ultrasonic Vibration on Ultrasonic Assisted Soldering of Cu/SAC305/Cu Joint. Journal of Advanced Research in Applied Mechanics, 36 (1), 1–9. Available at: https://akademiabaru.com/submit/index.php/aram/article/view/1792
5. Hu, H., Cheng, Y. F. (2016). Modeling by computational fluid dynamics simulation of pipeline corrosion in CO2-containing oil-water two phase flow. Journal of Petroleum Science and Engineering, 146, 134–141. doi: https://doi.org/10.1016/j.petrol.2016.04.030
6. Wu, W.-T., Aubry, N., Antaki, J. F., Massoudi, M. (2017). Normal stress effects in the gravity driven flow of granular materials. International Journal of Non-Linear Mechanics, 92, 84–91. doi: https://doi.org/10.1016/j.ijnonlinmec.2017.03.016
7. Schümann, H., Fossen, M. (2018). Oil-water dispersion formation, development and stability studied in a wheel-shaped flow loop. Journal of Petroleum Science and Engineering, 162, 567–576. doi: https://doi.org/10.1016/j.petrol.2017.10.066

8. Yang, Z., Velthuis, J., Veltin, J., Twerda, A. (2013). Cold restart of viscous multiphase flowline by hot water flushing. 16th International Conference on Multiphase Production Technology. Cannes.
9. Höhne, T., Porombka, P. (2018). Modelling horizontal two-phase flows using generalized models. Annals of Nuclear Energy, 111, 311–316. doi: https://doi.org/10.1016/j.anucene.2017.09.018
10. Parekh, J., Rzehak, R. (2018). Euler–Euler multiphase CFD-simulation with full Reynolds stress model and anisotropic bubble-induced turbulence. International Journal of Multiphase Flow, 99, 231–245. doi: https://doi.org/10.1016/j.ijmultiphaseflow.2017.10.012
11. Elahe, D., Alireza, H. (2018). A Review on Separation Techniques of Graphene Oxide (GO)/Base on Hybrid Polymer Membranes for Eradication of Dyes and Oil Compounds: Recent Progress in Graphene Oxide (GO)/Base on Polymer Membranes-Related Nanotechnologies. Clinical Medical Reviews and Case Reports, 5 (8). doi: https://doi.org/10.23937/2378-3656/1410228
12. Liu, J., Cheng, L., Huang, S. (2013). Numerical simulation study of gas-cap reservoir barrier water injection fluid interface migration laws. Metalurgia International, 18 (10), 23–27. Available at: https://bib.irb.hr/datoteka/644822.644822.MI2013NO10_MOSUROVIC_SOLESA_RAJSMAN.pdf
13. Sellman, E., Sams, G., Mandewalkar, S. P. (2013). Improved Dehydration and Desalting of Mature Crude Oil Fields. All Days. doi: https://doi.org/10.2118/164289-ms
14. Duan, S. (2009). Progressive water-oil transition zone due to transverse mixing near wells. Louisiana State University. Available at: https://digitalcommons.lsu.edu/cgi/viewcontent.cgi?article=4763&context=gradschool_dissertations
15. Zhang, D., Tan, J., Yang, D., Mu, S., Peng, Q. (2019). The Residual Potential of Bottom Water Reservoir Based upon Genetic Algorithm for the Relative Permeability Inversion. Journal of Geoscience and Environment Protection, 07 (04), 192–201. doi: https://doi.org/10.4236/gep.2019.74012
16. Bayat, M., Aminian, J., Bazmi, M., Shahhosseini, S., Sharifi, K. (2012). CFD modeling of fouling in crude oil pre-heaters. Energy Conversion and Management, 64, 344–350. doi: https://doi.org/10.1016/j.enconman.2012.05.003
17. Tkalich, P. (2006). A CFD solution of oil spill problems. Environmental Modelling & Software, 21 (2), 271–282. doi: https://doi.org/10.1016/j.envsoft.2004.04.024
18. Lin, Z., Sun, X., Yu, T., Zhang, Y., Li, Y., Zhu, Z. (2020). Gas–solid two-phase flow and erosion calculation of gate valve based on the CFD-DEM model. Powder Technology, 366, 395–407. doi: https://doi.org/10.1016/j.powtec.2020.02.050
19. Sharma, S. L., Ishii, M., Hibiki, T., Schlegel, J. P., Liu, Y., Buchanan, J. R. (2019). Beyond bubbly two-phase flow investigation using a CFD three-field two-fluid model. International Journal of Multiphase Flow, 113, 1–15. doi: https://doi.org/10.1016/j.ijmultiphaseflow.2018.12.010
20. Sergeev, V., Vatin, N., Kotov, E., Nemova, D., Khorobrov, S. (2020). Slug Regime Transitions in a Two-Phase Flow in Horizontal Round Pipe. CFD Simulations. Applied Sciences, 10 (23), 8739. doi: https://doi.org/10.3390/app10238739
21. Dempster, W., Alshaikh, M. (2018). CFD Prediction of Safety Valve Disc Forces Under Two Phase Flow Conditions. Volume 3A: Design and Analysis. doi: https://doi.org/10.1115/pvp2018-84745
22. Burlutskii, E. (2018). CFD study of oil-in-water two-phase flow in horizontal and vertical pipes. Journal of Petroleum Science and Engineering, 162, 524–531. doi: https://doi.org/10.1016/j.petrol.2017.10.035
23. Sarkar, S., Singh, K. K., Shenoy, K. T. (2019). Two-phase CFD modeling of pulsed disc and doughnut column: Prediction of dispersed phase holdup. Separation and Purification Technology, 209, 608–622. doi: https://doi.org/10.1016/j.seppur.2018.07.020
24. Swery, E. E., Meier, R., Lomov, S. V., Drechsler, K., Kelly, P. (2015). Predicting permeability based on flow simulations and textile modelling techniques: Comparison with experimental values and verification of FlowTex solver using Ansys CFX. Journal of Composite Materials, 50 (5), 601–615. doi: https://doi.org/10.1177/0021998315579927
25. Nosrati, K., Tahershamsi, A., Seyed Taheri, S. H. (2017). Numerical Analysis of Energy Loss Coefficient in Pipe Contraction Using ANSYS CFX Software. Civil Engineering Journal, 3 (4), 288–300. doi: https://doi.org/10.28991/cej-2017-00000091
26. Santana, H. S., da Silva, A. G. P., Lopes, M. G. M., Rodrigues, A. C., Taranto, O. P., Lameu Silva, J. (2020). Computational methodology for the development of microdevices and microreactors with ANSYS CFX. MethodsX, 7, 100765. doi: https://doi.org/10.1016/j.mex.2019.12.006
27. Sharma, D., Mistry, A., Mistry, H., Chaudhuri, P., Murugan, P. V., Patnaik, S. et. al. (2020). Thermal performance analysis and experimental validation of primary chamber of plasma pyrolysis system during preheating stage using CFD analysis in ANSYS CFX. Thermal Science and Engineering Progress, 18, 100525. doi: https://doi.org/10.1016/j.tsep.2020.100525