Al-Rafidain Engineering Journal (AREJ) Vol.29, No.2, September 2024, pp. 10-18

Three Phase Oil Separator Simulation Using CFD Analysis: A Review Study

Ali Ahmed Ibrahim
ali.21enp17@student.uomosul.edu.iq

Younis M. Najim
mahalyounis@uomosul.edu.iq

Amir Sultan Dawood
asdawood@uomosul.edu.iq

Department of Mechanical Engineering, College of Engineering, University of Mosul, Mosul, Iraq

Received: August 13th, 2023 Received in revised form: September 24th, 2023 Accepted: October 6th, 2023

ABSTRACT

The three-phase separator, a cornerstone of oil and gas production, is pivotal for maintaining high-quality oil output and minimizing environmental impact. Efficient separation of water, oil, and gas ensures superior oil quality and economic viability by removing impurities and contaminants from the extracted mixture. Moreover, the environmental significance lies in preventing contamination of water bodies through proper disposal of produced water. This review paper presents a comprehensive investigation on the simulation of three-phase oil separators utilizing Computational Fluid Dynamics (CFD) analysis. The emergence of CFD as a powerful tool has revolutionized separator design by unraveling complex flow patterns and interface structures. CFD aids in understanding turbulent flow structures resulting from inlet diverter interactions, leading to enhanced separation efficiency. This innovation reduces design costs by allowing engineers to simulate various configurations, resulting in optimized separator designs. Essential design parameters encompass oil properties influencing separation behavior, the design of the inlet diverter affecting flow dynamics, mean residence time for phase separation, separator diameter determined using techniques like Monner and Svrcek or Arnold and Stewart, weir height for phase interface control, and droplet size and distribution which significantly impact separation efficiency are also covered in this work.

Keywords:

CFD; Oil separator; three phase separators; multiphase flow; CFD simulation.

https://rengj.mosuljournals.com
Email: alrafidain_engjournal2@uomosul.edu.iq

===

1. INTRODUCTION

In oil and gas production processes, the three-phase separator plays a pivotal role in ensuring the quality of the extracted resources. This integral piece of equipment is employed to separate the extracted mixture into its three main components: water, oil, and gas. The significance of the three-phase separator lies in its ability to enhance the overall quality of the oil output, minimize environmental impact, and adhere to regulatory standards. The quality of oil extracted from reservoirs is directly linked to the efficiency and performance of three-phase separators. Efficient separation of water, oil, and gas ensures that the oil output is of a higher quality, with reduced levels of impurities and contaminants. By removing unwanted elements, such as water and gas, from the oil stream, these separators optimize production processes and enhance the economic viability of oil and gas operations.

Apart from its impact on oil quality, the three-phase separator also plays a crucial role in minimizing the environmental footprint of oil and gas activities. Effective separation of water and gas components ensures that the produced water can be properly treated before disposal, preventing contamination of water bodies and soil. This helps meet environmental regulations and safeguards ecosystems from potential damage [1].

Three-phase separators are ingeniously designed based on the principle of gravity-based phase separation. The mixture enters the separator at a high velocity and is allowed to slow down within the vessel. This controlled slowdown allows the heavier water phase to settle at the bottom, the lighter oil phase to accumulate above the water, and the gaseous phase to rise to the top. Internal components, such as baffles and weirs, aid in facilitating efficient phase separation.

Computational Fluid Dynamics (CFD) tools have revolutionized the design process of three-phase separators. Through parametric CFD analysis, engineers can simulate various design configurations and assess their performance virtually. This approach significantly reduces design costs by eliminating the need for multiple physical prototypes. Furthermore, CFD analysis provides insights into complex flow patterns, phase interactions, and interface structures within the separator. This information aids in fine-tuning the separator's design for optimal separation efficiency, ensuring that oil, water, and gas phases are accurately partitioned [2].

There are primarily two main types of three-phase oil separators widely employed in the oil and gas industry: horizontal and vertical separators [3]. Vertical separators feature a tall, vertical design that accommodates high flow rates and offers enhanced separation efficiency due to the increased length for phase separation. On the other hand, horizontal separators, shown in Fig. 1) are characterized by their cylindrical shape and horizontal orientation, facilitating gravity-driven separation of phases (water, oil, and gas). These separators are well-suited for applications with moderate flow rates and where space constraints might dictate their installation. The choice between these two types depends on factors such as flow rates, available space, and desired separation performance [4].

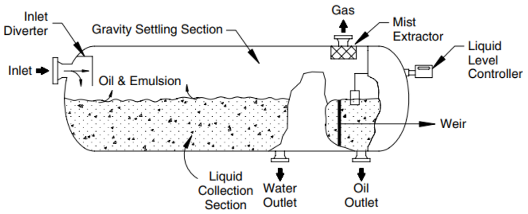


Fig. 1 Cutaway view of a horizontal three-phase separator with interface level control and weir

In this work, many previous studies that used computational fluid dynamics in modeling and simulating the oil separator will be addressed, taking into account some parameters which effect on the performance of the oil separator, as droplet size, weir height, mean residence time, separator design techniques, oil properties, and separator motion.

2. CFD MODELING

The application of Computational Fluid Dynamics (CFD) modelling in simulating three-phase separators, involving intricate interactions among oil, water, and gas, has received significant attention in various industries, particularly the oil and gas sector. CFD has emerged as a highly influential and indispensable tool for numerous critical aspects of separator design, analysis, and performance prediction. Its pivotal role during the design phase allows engineers to explore diverse configurations and optimize designs to achieve optimal performance. Through the simulation of fluid behavior within the separator, CFD enables the identification of potential flow issues, such as recirculation, turbulence, or dead zones, which can adversely impact efficiency. As a result, the integration of CFD during the design phase contributes to the development of robust and efficient separators.

Furthermore, the use of CFD in modeling three-phase separators presents a significant cost advantage over traditional experimental setups. The construction of physical prototypes and the conduction of experimental tests can be financially burdensome, time-consuming, and sometimes impractical due to the complexities inherent in three-phase flow studies. CFD simulations eliminate the need for costly physical setups, making it a preferred and cost-effective alternative for comprehending and optimizing separator behavior.

Previous studies can be classified into several classifications according to the effect of some parameters that affect to the separation efficiency and separator performance in general. This work is constructed to cover the main design parameters of the oil separators as follows:

A. Oil Property

Oil properties are directly involved in the design of all oil separators, but there is very little previous research that has studied the effect of oil properties on separation efficiency.

Carvalho et al. [5] conducted a comprehensive investigation on the dynamic behavior of fluids within a horizontal three-phase oil separator and explored the influence of different oil properties on this behavior. The research revealed significant variations in fluid dynamics when altering the physical properties of the oil. Specifically, differences in oil density and viscosity directly affected the initial separation process. Increased viscosity and density led to reduced withdrawal between liquids and hindered sedimentation, adversely impacting the separation process and exacerbating its complexity. The simulations employed three distinct cases of oil with varying properties. Two cases demonstrated promising results, while the simulation of the heavier oil yielded unsatisfactory outcomes. Notably, the results indicated that the volume fraction of oil in the outlet was 99.607% for 40.28 API (light oil), 99.66% for 35.8 API (medium oil), and 89.5% for 30.99 API (heavy oil). The CFD simulations highlighted an inverse relationship between viscosity and separation efficiency, as higher viscosity levels correlated with decreased separation rates between liquid phases.

B. Separator Motion

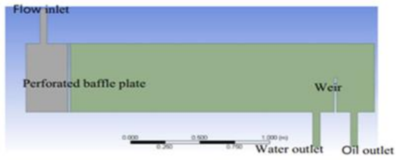
The angular motion of an oil separator along various axes can also exert an influence on the separator's performance. In a study conducted by Le et al. [6], a transient Eulerian computational fluid dynamics (CFD) model was developed to investigate the effects of three angular motions (namely, rolling, yawing, and pitching) on the separation efficiency of an [air–water–oil] separator. The separator configuration was characterized by a flat cylindrical design encompassing key components such as an inlet for the mixed fluid, a section facilitating gravitational separation of water and oil through coalescence, and a segment designed for mist removal.

The CFD simulation entailed the incorporation of equations governing fluid flow continuity, momentum, and the standard k-ε turbulence model. Additionally, a moving reference frame was employed to account for the angular motions introduced. Realism in the CFD model necessitated meticulous management of exit pressures at both the water and oil discharge outlets. To achieve this, a user-defined function was implemented for exit pressure control, as the absence of such control led to unrealistic backflow from the water outlet.

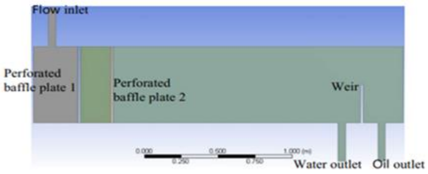
Subsequently, various CFD scenarios were replicated, including cases involving no motion, rolling, yawing, pitching at 2°, and pitching at 4°. Each scenario had a duration of 8 seconds and maintained controlled exit pressure conditions. The simulations involving angular movements with pressure control exhibited consistent cyclic patterns, resulting in notably effective oil separation. However, it was observed that during the 4° pitching motion, the oil recovery rate decreased to 93%, and the water outlet exhibited a purity of 77%.

C. Mean Residence Time

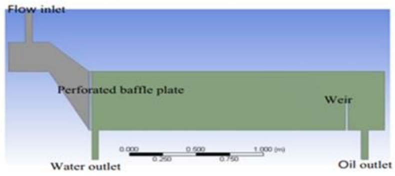
To incorporate the effect of mean residence time of the fluids inside the separating vessel on the separation efficiency, in their research, T. Acharya et al. [7] investigated three different geometries of oil separators to analyze the impact of these geometries on the mean residence time of the hydrocarbon phase, considering varying water-cuts. The three geometries examined (illustrated in Fig. 2)) included: a first configuration featuring a single perforated baffle plate at the entrance, a second configuration incorporating two perforated baffle plates at the entry area, and a third configuration with a sloping throat section connecting the initial area with the gravity area, all having identical sizes. Upon conducting simulations, the results indicated that the first and second geometries led to an increase in the mean residence time with an elevation in the percentage of water-cut, thereby enhancing the separator's efficiency. However, in the case of the last geometric shape, the mean residence time showed an initial increase at a 21% water-cut, followed by a decrease at a 31% water-cut, and then a subsequent increase at a 57% water-cut. These findings suggested that horizontal separators incorporating a sloping throat section demonstrated higher efficiency compared to horizontal separators lacking such a feature, especially at low water-cut levels.



First geometry with single perforated baffle



Second geometry with double perforated baffle



Third geometry with throat section

Fig. 2 Different Geometries of The Separator

Furthermore, Baghi and Karimi [8], conducted a CFD analysis of a two-phase oil-water separator. The study focused on evaluating the influence of four distinct inlet deflectors on the mean residence time. These inlet deflectors consisted of a spherical design and three plate configurations, each characterized by varying angles. The goal was to discern their respective effects on the separator's performance.

The validation of computational fluid dynamics (CFD) simulations was performed based on the average residence time, which was determined through pilot separator experimentation. The simulation utilized the multiphase Volume of Fluid (VOF) theory, implemented via Ansys Fluent software. The initial phase of this study involved an examination of the impact of halving the separator's geometry along its line of symmetry. The objective was to assess the viability of using this reduced geometry in place of the complete geometry within the simulation.

Among the plate inlet deflectors, it was observed that the configuration with an angle of 105 degrees between the plates yielded the highest mean residence time, whereas the configuration with an angle of 135 degrees between the plates resulted in the lowest mean residence time. This divergence in mean residence time outcomes is attributed to the intricate interplay between fluid conduction within the separator and its interactions with the internal walls of the separator. Notably, the introduction of the spherical inlet deflector led to an 8.9% increase in the mean residence time compared to the mean residence time observed in the absence of the inlet deflector.

In another paper, T. Acharya et al. [9] conducted an experimental investigation involving a horizontal gravity separator, which has previously been explored by other researchers. They then performed computational fluid dynamics (CFD) simulations on this identical geometric setup, using identical operating conditions. The results of these simulations showed a noteworthy agreement in terms of qualitative agreement with previous experimental results.

In line with previous experiments, the CFD results highlighted a positive relationship between the mean residence time (MRT) of the organic phase and the water cutting escalation. Furthermore, analysis of the residence time distribution (RTD) characteristics revealed a remarkable similarity in trends between the CFD-derived results and the experimental observations mentioned previously.

D. Component Modification

Other studies use CFD to study the effect of modifications to some components of the separator, especially the entrance baffles configuration which control the speed of fluids at entry and change the kinetic energy of the inlet flow. In this context, in their study, Ping Yu et al. [10] utilized CFD simulations to replicate the internal flow of a three-phase separator, examining three distinct inlet components: baffle-type, upper holes box type, and lower holes box type (as depicted in Fig. 3. The research findings indicated that the fluid flow exhibited greater stability when passing through the lower holes box type inlet component, as observed across all three cases tested.

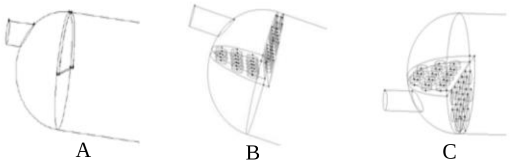


Fig. 3 (a) Baffle-type entrance; (b) Upper holes box type; (c) Lower holes box type

Also T. Acharya et al. [7] As mentioned previously studied three different geometries of oil separators to analyze the effect of these geometries on the average residence time of the hydrocarbon phase, taking into account varying water cuts. The results described in the previous section were obtained.

In a separate investigation, S. Yayla et al. [11] conducted a comprehensive two-phase computational fluid dynamics (CFD) analysis of three-dimensional turbulent flow within a horizontal separator employed within the petroleum industry. This examination encompassed two distinct geometric configurations as shown in Fig. 4: one entailing a straight plate located atop the separator, and the other incorporating a straight plate positioned adjacent to the separator. The study aimed to elucidate the impact of various factors, namely, the location of the separator, the gap between the separator inlet and the conversion plate, and the inlet velocity, on the efficacy of separation. The researchers employed the conventional k-ε turbulence model for this purpose. To carry out the investigation, they explored three distinct distances between the straight adapter plate and the separator inlet (0.1 m, 0.15 m, 0.2 m) and four different inlet velocities (0.25 m/s, 0.5 m/s, 0.75 m/s, and 1 m/s), considering the Euler mixture model to account for the two-phase flow.

The results revealed that the highest achievable separation efficiency reached 99.772% when the mixture was introduced into the separator from the top with an inlet velocity of 0.25 m/s, while the plate was positioned 0.2 m away from the inlet section of the separator. Importantly, the findings unveiled an inverse relationship between the inlet velocity and the separation efficiency, whereby an increase in the inlet velocity resulted in a reduction in separation efficiency.

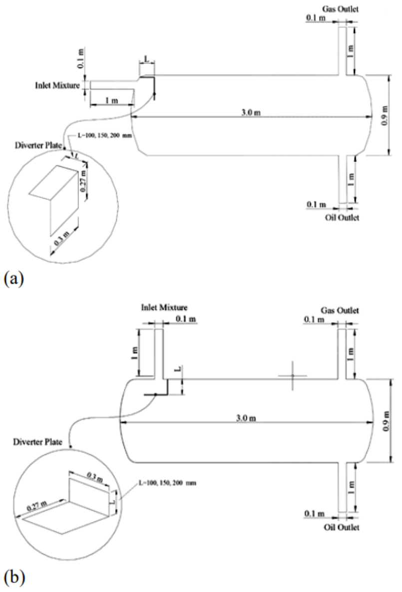


Fig.4 Geometry and dimensions of two different separator arrangement a) separator with side inlet pipe, b) separator with top inlet pipe.

E. Separator Design Techniques

The Arnold and Stewart [1] and Monnery and Svrcek [12] techniques are well-known semi-empirical techniques used to predict the oil separator preliminary dimensions. Ahmadreza et al. [13] applied these empirical techniques to estimate the dimensions of a multi-phase separator. The most important equations of this techniques are summarized as:

- **Monner and Svrcek Techniques**

Monner and Svrcek's [12] research has analyzed both theoretical and practical aspects of oil separator design, addressing critical factors such as residence time, geometry, and the influence of inlet deflectors. Their work has provided valuable insights for engineers and researchers seeking to enhance the efficiency and effectiveness of oil-water separators in industries like oil and gas. Overall, their contributions have played a pivotal role in advancing the field of oil separator design techniques, with a strong emphasis on empirical validation and practical application. They suggest that the separator main diameter can be estimated according to:

$$D = \left[\frac{4\,(v_h+v_s)}{0.5\,\pi\left(\frac{L}{D}\right)}\right]^{\frac{1}{3}} \ldots\ldots\ldots\ldots\ldots\ldots\ldots 1$$

where D is separator diameter, v_h is hold volume, v_s is surge volume and (L/D) is slenderness ratio.

$$L_{ss} = \frac{v_h+v_s}{A_T-A_g-(A_{HL}-A_{LL})} \ldots\ldots\ldots\ldots\ldots 2$$

where, L_{ss} denotes the length measured from one seam of the separator to the other, A_g signifies the portion of the separator's area that is filled with gas, A_{HL} represents the interface region occupied by the denser or heavier liquid, A_T signifies the overall cross-sectional area of the separator, and A_{LL} indicates the area at the interface specifically taken up by the less dense or lighter liquid.

- **Arnold and Stewart Techniques**

Their studies have shed light on critical parameters, including residence time, separator geometry, and the impact of various inlet deflector configurations which has proven invaluable to engineers and researchers seeking to enhance the efficiency and efficacy of oil-water separators, particularly in industries like oil and gas.

$$DL_{eff} = 0.345\left[\frac{T\,Z\,Q_g}{P}\right]\left[\left(\frac{\rho_g}{\rho_l-\rho_g}\right)\frac{C_D}{d_p}\right]^{\frac{1}{2}} \ldots\ldots 3$$

where L_{eff} is the vessel effective length, T is operating temperature, gas flow rate Q_g, operational pressure P, and gas compressibility Z.

$$D^2\,L_{eff} = 0.042\,[Q_w\,t_{rw} + Q_o\,t_{ro}] \ldots\ldots 4$$

Where Q_w represents the water flow rate, t_{rw} signifies the water retention time, Q_o denotes the oil flow rate, and t_{ro} stands for the oil retention time. The ultimate dimensions of the vessel were determined with reference to the slenderness ratio, which is defined as the ratio of its length.

A series of equations pertaining to continuity and momentum were systematically solved, ensuring that the various phases involved were constrained from intermixing with each other. The continuity equation for one of the phases is given as:

$$\frac{\partial}{\partial t}(\alpha_m\rho_m) + \nabla.(\alpha_m\rho_m \mathrm{U_m}) = S_m \ldots\ldots 5$$

Where U_m denoting the velocity of phase (m), S_m represent the mass source term, and α_m is volume fraction of phase (m). The momentum equation for VOF model is:

$$\frac{\partial}{\partial t}(\rho U) + \nabla.(\rho U.U) = -\nabla P + \nabla\tau + \rho g + b \ldots\ldots\ldots\ldots\ldots\ldots 6$$

Where τ is the viscous stress tensor, U is the average fluid phase velocity, and b is the external body force.

To assess the separation efficiency and complex fluid behavior within the three-phase separator, two multiphase models, Volume of Fluid (VOF) and Discrete Phase Model (DPM), were integrated with the k-ε turbulence model. The

Discrete Random Wall (DRW) model was also used to account for the influence of background phase velocity variation on the movement of secondary phase particles. The CFD simulations yielded results indicating that the Monnery and Svrcek [12] example exhibited higher values of turbulent kinetic energy (k) and velocity magnitude compared to the Arnold and Stewart [1] scenario. Consequently, the Monnery and Svrcek [12] separator experienced more liquid carryover. The simulation density profiles further projected the Monnery and Svrcek [12] vessel's inefficiency in oil-water separation. In terms of the mass distribution of oil and water, the Arnold and Stewart [1] separator demonstrated superior performance compared to the Monnery and Svrcek [12] vessel. DPM was used to analyze the kinetic energy of particles in the gas-rich zone of each separator. Higher particle kinetic energy led to a greater spread of secondary phase droplets, causing an increase in oil droplets at the gas outlet and a decrease in the effectiveness of oil-water separation, as indicated by the simulation results. Although numerical simulations did not fully cover optimization results, valuable understanding was gained by contrasting outcomes from various setups. This method bolstered confidence in combining CFD with semi-empirical methods during the design phase.

F. Weir Height

Using CFD to explore the effect of overflow weir on the separator performance is studied by Nabil Kharoua et al. [14] by investigating the impact of the overflow weir height on the performance of a three-phase horizontal separator situated in the Al-Bab oil field in Abu Dhabi. To ensure higher accuracy, they employed the computational fluid dynamics (CFD) method, utilizing the Eulerian-Eulerian model coupled with the (k-ε) turbulence model for analysis. The study revealed that reducing the relative height of the overflow weir resulted in decreased water presence in the gaseous phase at the exit. However, it simultaneously led to an increase in the amount of water present in the oil at the exit. Notably, the researchers assumed a specific diameter for the bubbles in the liquid. When comparing the obtained results with the fieldwork of the separator, numerous contradictions emerged. These inconsistencies were attributed to the utilization of overly simplistic hypotheses and insufficient consideration of the effect of bubble size on the overall separator performance.

Song et. al. [15] introduced a dynamic simulator designed for three-phase gravity separators within oil production facilities, focusing on the significant influence of weir height on separator performance. The simulator incorporates a mass conservation equation to compute key parameters such as pressure, water level, and oil level, along with a mass balance equation for the dispersed phase to determine oil-water separation efficiency.

To effectively control the separator's water level, oil level, and pressure, proportional integral controllers are employed, adjusting the opening of the three outlet valves for oil, gas, and water. The model's validity was established through field data, incorporating prescribed valve openings and proportional integral controller parameters.
With a verified simulator in hand, they investigated the dynamic nature of the separator filling process. Notably, they explored the consequences of changes in pressure, oil level, and water level setpoints on the separator's operational state. The analysis provided detailed insights into variations in liquid levels, pressure levels, and the opening of the three outlet valves. Furthermore, they delve into the impact of operating conditions, such as inlet flow, water setpoints, and notably, weir height, on the separation efficiency.

G. Droplet Size and Distribution

The efficacy of oil separators is profoundly influenced by the dimensions and arrangement of liquid droplets. In the study conducted by N. Kharoua et al. [2], a comprehensive research investigation was conducted to examine the impact of water droplet size distribution on the operational performance and internal flow dynamics of a gravitational separator. Computational Fluid Dynamics (CFD) simulations were employed, incorporating both turbulence k-ε and multiphase Eulerian-Eulerian models, to investigate the separation dynamics of various phases within the gravitational separator. The investigation encompassed considerations such as the distribution of droplet sizes, coalescence phenomena, and the breakup patterns of secondary phases through the utilization of the population balance model (PBM). Through a comparative analysis of scenarios featuring diverse distributions of water phases (some characterized by multiple-sized droplets and others with uniform-sized droplets), the researchers observed that employing the population balance model for the water phase resulted in a more realistic reduction in the extent of water admixture with the oil in contrast to real-world experiments. Furthermore, the study revealed that droplets tended to coalesce in specific regions within the separator, notably in the Schoepentoeter, the mixing compartment, and to a lesser extent, in the

Spiraflow. The non-uniform distribution, primarily consisting of smaller droplets, yielded a consistent presence of notable concentrations beyond the mixing compartment, which contrasted with the standard Rosin-Rammler distribution that led to a predominant concentration of larger droplets just before the baffles.

Assessing droplet size and distribution is crucial in gauging the efficacy of separation within a two/three phase separator. In a laboratory context, Kul Pun et al. [16] scrutinized the impact of fluid flow and oil pad thickness on droplet size. Their findings revealed a correlation wherein heightened inflow rates of oil and water correlated with diminished droplet size, whereas thicker oil cushions correlated with larger droplet sizes. Subsequently, the collected data underwent fitting via a straightforward Gaussian model, yielding determinations for mean, standard deviation, and amplitude coefficients. Analysis of these fitted parameters the Reynolds number yielded trends, offering a potential avenue for refining initial parameters in population models for segregation simulations via computational fluid dynamics (CFD) packages. Remarkably, the central predictive parameter dictating the positioning of the Gaussian distribution was identified as the mean droplet size.

3. CONCLUSION

In conclusion, this paper provides a comprehensive review of prior scholarly investigations employing computational fluid dynamics (CFD) to analyze the performance of three-phase oil separators. Furthermore, it scrutinizes various factors known to influence separation efficiency, which have been subjects of prior research endeavors. Additionally, the manuscript delves into diverse methodologies employed for the design of three-phase separators. Collectively, the consistent application of CFD tools in scrutinizing fluid dynamics within separators has yielded compelling findings in previous scholarly undertakings, underscoring the reliability and efficacy of these computational methodologies. A synthesis of the reviewed studies yields the following key findings:

1. Augmented oil viscosity and density have been found to impede the separation process by diminishing the inter-liquid withdrawal and hindering sedimentation. This exacerbates the complexity of the separation process [5].
2. Simulations involving angular movements under controlled pressure conditions exhibit recurring cyclic patterns, resulting in optimal oil separation. Nevertheless, it was observed that during a 4° pitching motion, oil recovery diminished to 93%, and the water outlet exhibited a purity level of 77% [6].
3. The number of inlet diverters, their angles, and their shapes have been established as factors influencing an increase in the mean residence time, subsequently enhancing the efficiency of the oil separator [7][9].
4. The results indicate that fluid flow experiences enhanced stability when traversing through a square-type inlet component at the bottom hole, a phenomenon observed consistently across all three examined cases [10].
5. The high speed of the mixture entering the separator negatively affects the efficiency of the separator, while the distance of the inlet transformer from the separator inlet increases the efficiency significantly [11].
6. Reducing the relative height of the weir has been demonstrated to reduce the presence of water in the gaseous phase at the exit of the separator [14].
7. Higher flow rates of water and oil reduce the size of the droplets, and an increase in the thickness of the oil cushions also leads to an increase in the size of the droplets [16].

NOMENCLATURE

D	separator diameter (m).
v_h	hold volume (m$^{3)}$
v_s	surge volume (m$^{3)}$
L/D	slenderness ratio
A_g	separator area occupied by gas (m^2)
A_{HL}	interface area occupied by heavy liquid (m^2)
A_T	separator cross section area (m^2)
A_{LL}	interface area occupied by light liquid (m^2)
L_{eff}	effective length of the vessel (m).
T	operating temperature (K)
Q_g	gas flow rate (scm/h)
P	operation pressure (Kpa)
Z	gas compressibility
Q_w	water flow rate (m^3/s)
t_{rw}	water retention time (min)
Q_o	oil flow rate (m^3/s)
t_{ro}	oil retention time (min)
U_m	velocity of phase (m)
S_m	mass source term (kg/s.m^3)
α_m	volume fraction of phase (m)
τ	viscous stress tensor (N/m^3)
b	external body force (N/m^3)
CFD	computational fluid dynamics
DRW	discrete random wall
VOF	volume of fluid
DPM	discrete phase model
PBM	population balance model

REFERENCES

[1] K. Arnold and M. Stewart, "Surface Production Operations Design of Oil Handling Systems and Facilities," Third Edition ed., vol. 1, 2008.

[2] N. Kharoua, L. Khezzar and H. Saadawi, "CFD Modelling of a Horizontal Three-Phase Separator: A Population Balance Approach," American Journal of Fluid Dynamics, vol. 3(4), pp. 101-118, 2013.

[3] [1] H. K. Abdel-Aal, M. Aggour, and M. A. Fahim, Petroleum and Gas Field Processing. New York: Marcel Dekker, 2003.

[4] A. Kayode Coker "Petroleum Refining Design and Applications Handbook" Vol. 3, 2022.

[5] A. J. G. Carvalho, D. C. Galindo, M. S. C. Tenório and J. L. G. Marinho, "Modeling and Simulation of a Horizontal Three-Phase Separator: Influence of Physicochemical Properties of Oil," Brazilian Journal of Petroleum and Gas, vol. 14 n. 4, pp. 205-220, 2020.

[6] L. T. Thi, N. S. Ich, L. Young, P. Chi-Kyun, L. B. Don, K. B. Gook and L. D. Ha, "Three-phase Eulerian computational fluid dynamics of air–water–oil separator under off-shore operation," Journal of Petroleum Science and Engineering, vol. 171, pp. 731-747, 2018.

[7] [1] T. Acharya and T. Potter, "A CFD study on hydrocarbon mean residence time in a horizontal oil–water separator," SN Applied Sciences, vol. 3, no. 4, Mar. 2021. doi:10.1007/s42452-021-04483-x.

[8] M. D. Y. Baghi, M. Karimi "Investigating the Mean Residence Time in A Two-Phase Oil-Water Separator Using Volume of Fluid Multiphase Theory in Computational Fluid Dynamics Simulation", Spe Prod & Oper 38 (03):527–536.

[9] T. Acharya; L. Casimiro "Evaluation of flow characteristics in an onshore horizontal separator using computational fluid dynamics" Journal of Ocean Engineering and Science 5 (2020) 261-268.

[10] P. Yue, S. Liua, Y. Wanga, W. Lina, Z. Xiaoa and C. Wanga, "8. Ping Yue, Shilei Liua, Yaohua Wanga, WStudy on Internal Flow Field of the Three-phase Separator with Different Entrance Components," Procedia Engineering, vol. 31, p. 145 – 149, 2012.

[11] S. Yayla, K. Kamal and S. Bayraktar "Numerical Analysis of a Two-Phase Flow (Oil and Gas) in a Horizontal Separator used in Petroleum Projects" Journal of Applied Fluid Mechanics, Vol. 12, No. 4, pp. 1037-1045, 2019.

[12] W.Y. Svrcek.; W.D. Monnery, "Successfully Specify Three-Phase Separators," Chemical Engineering Progress, pp. 29-40, 1994.

[13] A. Ghaffarkhah, M. A. Shahrabi, M. K. Moraveji and H. Eslami, "Application of CFD for designing conventional three phase oilfield separator," Egyptian Journal of Petroleum, 2016.

[14] N. Kharoua, L. Khezzar and H. Saadawi, "Application of CFD to Debottleneck Production Separators in a Major Oil Field in the Middle East," Society of Petroleum Engineers, 2012.

[15] Shangfei Song, Xuanzhang Liu, Chenxuan Li, Zhe Li, Shijia Zhang, Wei Wu, Bohui Shi, Qi Kang, Haihao Wu, and Jing Gong, "Dynamic Simulator for Three-Phase Gravity Separators in Oil Production Facilities" American Chemical Society, 2023, 6078–6089

[16] K. Pun, F. A. Hamad, T. Ahmed, J. O. Ugwu, J. Eyers, G. Lawson and P. A. Russell, "Investigation of Droplet Size Produced in Two-Phase," International Journal of Chemical and Molecular Engineering, vol. 17, 2023.

محاكاة فاصل نفطي ثلاثي الطور باستخدام تحليل ديناميكية الموائع الحسابية: مراجعة

علي احمد ابراهيم
ali.21enp17@student.uomosul.edu.iq

يونس محل نجم
mahalyounis@uomosul.edu.iq

امير سلطان داود
asdawood@uomosul.edu.iq

قسم الهندسة الميكانيكية، كلية الهندسة، جامعة الموصل، الموصل، العراق

تاريخ الاستلام: 13 اغسطس 2023 استلم بصيغته المنقحة: 24 سبتمبر 2023 تاريخ القبول: 6 اكتوبر 2023

الملخص

يعتبر الفاصل ثلاثي الطور هو الحجر الزاوية في إنتاج النفط والغاز، وذلك للحفاظ على إنتاج نفط عالي الجودة وتقليل التأثير البيئي. يضمن الفصل الفعال للمياه والنفط والغاز جودة النفط الفائقة والجدوى الاقتصادية عن طريق إزالة الشوائب والملوثات من الخليط المستخرج. علاوة على ذلك، تكمن الأهمية البيئية في منع تلوث المسطحات المائية من خلال التخلص السليم من المياه المنتجة. تقدم ورقة المراجعة هذه تحقيقًا شاملاً حول محاكاة فواصل النفط ثلاثية الطور باستخدام تحليل ديناميكية الموائع الحسابية. حيث أحدث ظهور ديناميكية الموائع الحسابية كأداة قوية ثورة في تصميم الفاصل من خلال كشف أنماط التدفق المعقدة وهياكل الواجهة. يساعد ديناميكية الموائع الحسابية في فهم هياكل التدفق المضطرب الناتجة عن تفاعلات محول المدخل، مما يؤدي إلى تحسين كفاءة الفصل. يقلل هذا الابتكار من تكاليف التصميم من خلال السماح للمهندسين بمحاكاة التكوينات المختلفة، مما يؤدي إلى تصميمات فاصل محسّنة. تشمل معلمات التصميم الأساسية خصائص النفط التي تؤثر على سلوك الفصل، وتصميم محول المدخل الذي يؤثر على ديناميكيات التدفق، ومتوسط وقت الإقامة لفصل الطور، وقطر الفاصل المحدد باستخدام تقنيات مثل Monner وSvrcek أو Arnold وStewart، وارتفاع السد للتحكم في واجهة الطور، وكذلك حجم القطرة والتوزيع الذي يؤثر بشكل كبير على كفاءة الفصل يتم تناوله أيضًا في هذا العمل.

الكلمات الداله:

فواصل النفط، محاكاة، جريان متعددة الاطوار، فواصل ثلاثية الطور، ديناميكبة الموائع الحسابية.