JOURNAL OF GAS TECHNOLOGY

Volume 5 / Issue 1 / Summer 2020 / Pages 57-69

Journal Homepage: http://jgt.irangi.org

Application of CFD for Troubleshooting and Hydrodynamic Analysis in an Industrial Three-Phase Gravity Separator

Zohreh Khalifat[1], Mortaza Zivdar[2*], Rahbar Rahimi[3]

1. Ph.D student, Chem. Eng. Dept., Faculty of Eng., University of Sistan and Baluchestan, Zahedan, Iran
2. Corresponding Author: Chem. Eng. Dept., Faculty of Eng., University of Sistan and Baluchestan, Zahedan, Iran
3. Chem. Eng. Dept., Faculty of Eng., University of Sistan and Baluchestan, Zahedan, Iran

ARTICLE INFO

ORIGINAL RESEARCH ARTICLE

Article History:
Received: 26 May 2020
Revised: 22 August 2020
Accepted: 03 September 2020

Keywords:
Three phase gravity separator
Volume of fluid
Dispersed phase model
Troubleshooting
Hydrodynamic analysis

DOR: 20.1001.1.25885596.2020.5.1.6.6

ABSTRACT

Multiphase separation in gravity separators is one of the important processes in different industries. This study presents a computational fluid dynamics (CFD) simulation of an industrial three-phase boot separator applying a coupled volume of fluid (VOF)-dispersed phase model (DPM) method for hydrodynamic analysis and troubleshooting of the separation process. Noted that despite the wide application of the boot separator in different industries, no research has been performed on this type of separator to investigate the macroscopic and microscopic behavior of the separation process. The results of numerical calculations based on three-phase flow profile, secondary phase behavior, separator performance, and size distribution of the droplets were investigated in this research. Results showed that the CFD model is well capable of estimating the separation behavior in a three-phase boot separator. Troubleshooting of the studied separator was also investigated to detect the parameters that might decrease the separation performance. Based on the results, it is concluded that the separator suffers from the type of the inlet diverter, lack of an efficient mist extractor at the gas outlet and also lack of an appropriate vortex breaker at the oil outlet. The effect of increasing the inlet water flow rate on the separator performance was another parameter that was studied in this research. Results demonstrated that increasing the inlet water flow rate from 11823-47295 kg/hr caused an increase in the mass of droplets at the gas outlet from 0.09 to 1.6 kg/hr, but this increase did not lead to a significant decrease in the separation efficiency.

How to cite this article
Z. Khalifat, M. Zivdar, R. Rahimi. Application of CFD for Troubleshooting and Hydrodynamic Analysis in an Industrial Three-Phase Gravity Separator. Journal of Gas Technology. 2020; 5(1): 57 -69. (http://www.jgt.irangi.org/article_251661.html)

---

*Corresponding author.
E-mail address: mzivdar@eng.usb.ac.ir (M. Zivdar)

1. Introduction

Three-phase gravity separators are one of the initial and important process equipment used for separating phases with different densities (oil, water, and gas) in different industries. Inappropriate design of such apparatus leads to a decrease in the separation efficiency of the separators and also causes the downstream equipment which is placed after the separators to be damaged. For example, the presence of liquid droplets with gas or bubble gas with liquids reduces the pump and compressor efficiency. The presence of water in oil phases also causes corrosion in tubes and requires the high cost for repair and maintenance of the equipment. So, appropriate design for such separators leads the industry's performance to be more efficient and economical. These separators are used in two horizontal and vertical types which the horizontal types are more common in Iran and can be categorized in two common types, i.e., weir type (when the water fraction is substantial) and boot type (when the water fraction is not substantial). Generally, semi-empirical methods which are based on the droplet settling theory are used to design the separators (Pourahmadi et al, 2012). Although these methods provide useful guidelines, simplified assumptions used in these methods such as considering a single droplet size with constant velocity in the droplet settling theory and also not considering the effects of turbulence and separator internals lead these approaches not to be completely acceptable (Monnery and Svrcek, 1994; Bothamley, 2013a, b; Ghafarkhah et al, 2017,2018). So, semi-empirical methods need to be improved. One reliable method to overcome the problems in semi-empirical methods is performing experimental studies. Noted that, because of the high-performance cost and technical problems in measuring the internal flow behavior using experimental studies, applying a more complete and economical method such as computational fluid dynamics (CFD) in analyzing the quality and quantity of the separation process and also debottlenecking of the separators is necessary (Ghafarkhah et al, 2018).

Eulerian-Eulerian (E-E) and Eulerian-Lagrangian (E-L) approaches are two common methods used in CFD simulation of multi-phase flows. In the E-E approach which includes the volume of fluid (VOF), mixture and Eulerian models, all the phases are considered as continuous phases which interact with each other. The Navier-Stokes equations are solved in this approach. In the dispersed phase model (DPM) which belongs to the E-L approach, one continuous phase and two or more discrete phases are considered. In this approach, the Navier-Stokes equation and the Newton second's law are solved for the continuous and discrete phase, respectively (Pourahmadi et al, 2012). Although CFD simulations of gravity separators were the subject of several types of researches, most of them were pertinent to the simulation of two-phase separators and limited works studied three-phase separators due to the complicated behavior and also the high calculation time in simulating three-phase flows. Noted that among the studies performed on three-phase separators, most of them used the E-E approach in evaluating the separation process.

Ahmed et al. (2017) used VOF and Eulerian models separately to simulate one pilot plant three-phase separator with a weir. The k-Ɛ model was used to consider the turbulent flow in this research. Because of the assumptions such as considering a single average diameter for liquid droplets, and not considering the interaction between liquid droplets in Eulerian model and also due to not considering suitable grid cell number for tracking the interfaces between phases in the VOF model, a high simulation error (30-50%) relative to field data were observed.

Kharoua et al. (2013 a) used Eulerian with the k-Ɛ model to investigate the flow behavior and separation performance in one three-phase industrial separator equipped with a weir. Because of considering liquid droplets with a fixed diameter, neglecting coalescence and breakup of the droplets and also the weakness of this model in the exact tracking of the interfaces between phases, unreliable results such as existing more water in the oil outlet were observed which were not in agreement with the field data.

In another study performed by Kharoua et al. (2013b) population balance model (PBM) was coupled with the Eulerian model to overcome the problems in their previous work. In this study, the droplet size distribution of the liquid droplets and also the coalescence and breakup of the droplets were taken into account. Although the results were in a better agreement with the field data, due to the limitation in this model pertinent to considering the droplet size distribution for just one secondary phase, the difference between simulation and industrial data were not negligible.

Considering the results of the studies on the models in the E-E approach showed that these models were not successful in exact estimating of the separator performance. Noted that among the models in the E-E approach, the VOF model is suitable in tracking the interfaces between phases and also the interfaces between the droplets and the continuous phase, but this model needs to track free surface around each droplet for the exact estimating of the droplet behavior and also the separator performance. So, a very fine grid is obligatory to achieve the exact simulation results which are not economical to be used in industrial scales. Using the DPM model is a solution to the problem encountered in the VOF model in which the droplets in the DPM model are treated as source terms that move into the domain (Cloete et al, 2009; Kirveski et al, 2013). It is noted that the coalescence and breakup and also the droplet size distribution of all the secondary phases can be considered in this model, but two continuous phases (oil and water) in three-phase separators which accumulate at the bottom of the separator are neglected in this model and just one gas phase as continuous phase and oil and water droplets as dispersed phases are modeled which leads to unreliable results in simulating three-phase separators (Pour Ahmadi et al, 2011). Therefore, DPM model needs three phases at the background to consider all three continuous phases and also to be available for droplets to interact with them. Because of the exact tracking of the interfaces between phases in the VOF model, it is a good candidate to be coupled with the DPM model for modeling three phases at the background (Pourahmadi et al, 2011; Qarot et al, 2014). Although the coupled VOF-DPM model is completely acceptable in simulating multiphase flow (Cloete et al, 2009a, b), very limited researches were performed on simulating industrial three-phase separators using this model.

Pourahmadi et al. (2011, 2012) used VOF-DPM with k-Ɛ turbulence model to simulate an industrial three-phase (oil, water, and gas) separator with a weir to improve its performance. The droplet size distribution of the secondary phases and also coalescence and breakup were taken into account in this study. Results showed that this model was good at estimating the separator performance to debottleneck the separator.

In another study performed by Ghafarkhah et al. (2017,2018), two different semi-empirical methods were used to design a pilot plant three-phase separator with a weir using VOF-DPM- k-Ɛ model to show which method is more realistic. Results demonstrated that the mentioned coupled model was successful in estimating the best dimension of the separator.

As it was mentioned before, in spite of different studies performed on the CFD simulation of multiphase separators, most of them considered the simulation of two-phase separators and limited works studied three-phase separators in industrial scales due to the complicated behavior and also the high calculation time in simulating three-phase flows in industrial scales. Three-phase boot separators are one of the important types of separators used when the volume fraction of water is very low relative to the other phases, but among the studies performed on three-phase separators, no research has been carried out on the CFD simulation of these types of separators to investigate their performance. So, the main object of this research is using a CFD model for investigating the hydrodynamic analysis to consider the microscopic and macroscopic treatments of the separation process in one industrial three-phase boot separator. Noted that the studied industrial separator is located in the Borzoye petrochemical Company in the south of Iran. Troubleshooting of the mentioned industrial separator was another important parameter which was considered in this research to detect the factors that might decrease the separation efficiency. Results highlighted the need for changing or adding some internals in the industrial separator to achieve better separation. To simulate the intended separator, a coupled VOF-DPM model because of its advantages over the other models mentioned in the introduction, was chosen and the commercial CFD package, Ansys Fluent 16.2 was selected for this purpose. The VOF model in this study was used to show the total fluid flow profile at the background (continuous oil, gas and water phases) and the DPM model was used to consider the behavior of the droplets in the separator. The k-Ɛ model was selected as the turbulence model in this study. Three-phase flow profile, secondary phase behavior, separator performance, and the size distribution of the droplets were the results of the numerical calculations. Results expressed that the VOF-DPM model is successful at estimating the separation behavior of the three-phase boot separator. It is noted that the effect of increasing the inlet water flow rate on the separator performance, due to the need to change the inlet water flow rate in the industry, was also investigated in this research.

2. Mathematical Modeling

The model used in this study is the combination of the VOF model to simulate three continuous phases at the background to track the interfaces between phases and also the DPM model to track the dispersed phases while interacting with the continuous phases.

1.2. VOF Model

The VOF model is used when tracking the interfaces between phases is important. In this model, one continuity equation for each phase to track the volume fraction of phases and also one momentum equation with a shared velocity field for all the phases are solved. The continuity equation for each phase is as (Cloete et al, 2009b; Xu et al, 2013).

$$\frac{\partial}{\partial t}(\alpha_m \rho_m) + \nabla \cdot (\alpha_m \rho_m \vec{u}_m) = 0 \qquad (1)$$

Where the subscript m is denoted as phase m. In this equation, $\vec{u}$, α and ρ are the average velocity, volume fraction and density of the continuous phase, respectively. The momentum equation is expressed as (Ansys Fluent, 2016; Bracill et al, 1992):

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla \cdot (\rho \vec{u}.\vec{u}) = -\nabla P + \nabla \tau + \rho g + \vec{F} \qquad (2)$$

Where τ is the shear stress, P is pressure and $\vec{F}$ is a source term denoted as the surface tension force between phases by applying the continuum surface force model proposed by Brackbill et al (Ansys Fluent, 2016; Bracill et al, 1992).

2.2. DPM Model

In this model, tracking of droplets in the E-L approach is predicted by implementing the Newton second’s law on each droplet. The particle acceleration in this equation is because of the drag, gravity and additional forces that are exerted on the droplets due to the existing of the continuous phase. This equation is shown as (Xu et al, 2013; Ansys Fluent, 2016):

$$\frac{d\overrightarrow{u_p}}{dt} = F_D(\vec{u} - \overrightarrow{u_p}) + \frac{g(\rho_p - \rho)}{\rho_p} + \vec{f} \qquad (3)$$

Where the subscript p represents the particles. The additional forces $(\vec{f})$ in this equation are mainly virtual mass, Brownian and thermophoretic forces (Xu et al, 2013; Ansys Fluent, 2016).

F_D is the drag force which is shown as (Xu et al, 2013):

$$F_D = \frac{18\mu C_D\, Re}{24\rho_p {d_p}^2} \qquad (4)$$

μ, d_p and ρ_p are the molecular viscosity, the diameter, and density of the particles, respectively. C_D is the drag coefficient which is calculated based on the Morsi and Alexander for spherical particles (Xu et al, 2013).

3-2. Turbulence Equation

The multiphase model in this work is coupled with the k-Ɛ model to consider the effect of turbulence on the separation process. Two different equations are solved using the k-Ɛ model for calculating turbulent kinetic energy (k) and turbulent dissipation rate (Ɛ) (Ghafarkhah,2017; Ansys Fluent, 2016):

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(u_i \rho k)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu + \mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] - \rho\varepsilon + \beta \qquad (5)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(u_i\rho\varepsilon) = \frac{\partial}{\partial j}\left[\left(\mu + \frac{\mu_t}{\sigma\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + \beta C_1\frac{\varepsilon}{k} - \rho C_2\frac{\varepsilon^2}{k} \qquad (6)$$

β, is the kinetic energy production term due to velocity gradient, σ is the surface tension and μ_t, is the turbulent viscosity which is calculated by equation 7 (Ghafarkhah et al, 2017).

$$\mu_T = \rho C_M \frac{k^2}{\varepsilon} \qquad (7)$$

The constants in equations 6 and 7 are as (Ansys Fluent, 2016):

$$C_1=1.44\ ,\ C_2=1.92\ ,\ C_M=0.09\ ,\sigma_k=1\ ,\ \sigma_\varepsilon=1.3 \qquad (8)$$

3. Fluid Properties

To show the separation process in this research, data of one industrial three-phase boot separator located in the Borzoye petrochemical Company were used. The mentioned separator operates at a temperature of 47°C and a pressure of 19 bar. To calculate the volume percentage and physical properties of each phase, the separator was simulated using Aspen Hysys V.9 (Khalifat et al, 1396) and the results were used to perform the present CFD simulation. The results of the Hysys simulation showed that the densities of the gas, oil, and water phases were 3.28, 692.6 and 991.1 kg/m^3, and their corresponding viscosities were 9.332e-6, 3.685e-4, and 5.783e-4 (kg/m.s), respectively. It should be noted that the volume percentages of each phase at the inlet mixture were 83.59%, 15.86%, and 0.56%, respectively.

The droplet size distributions of the secondary phases in the DPM model were estimated using logarithmic Rosin-Rammler equation as (Pourahmadi, 2010):

$$Y_{(d)} = 1 - exp\left(\frac{-d}{\bar{d}}\right)^n \qquad (9)$$

Where $Y_{(d)}$ is the mass fraction of particles, n is the spread parameter and d is the particle diameter (Pourahmadi et al, 2010).

The maximum and mean of droplet sizes were calculated based on one equation taken from a comprehensive study that considers all the physical properties of the fluid, which is shown as (Pourahmadi , 2010):

$$d_{max} = 1.38\left(\frac{\sigma^{0.6}}{{\rho_c}^{0.3}{\rho_d}^{0.2}{\mu_c}^{0.1}}\right)\left(\frac{D^{0.5}}{{u_c}^{1.1}}\right) \times \qquad (10)$$

$$\left(1 + 0.5975\left[\frac{\mu_d({\mu_c}^{0.25}{u_c}^{2.75}{\rho_c}^{-0.25}D^{-1.25}d_{max})^{\frac{1}{3}}}{\sigma}\right]\sqrt{\frac{\rho_c}{\rho_d}}\right)^{0.6}$$

$$d_{mean} = 0.4\, d_{max} \qquad (11)$$

The subscripts c and d are denoted as continuous and dispersed phases. D is the internal diameter of the tube which flow passes through it. The max and mean in the above equations are the representative of the maximum and mean of the droplets.

4. Physical Modeling and Grid Generation

Three-dimensional (3-D) modeling of the intended boot separator, which is shown in (Figure1), is performed in Gambit 2.4.6. The separator is equipped with one slopped inlet diverter at the entrance and one boot vessel to store the water at the bottom. As it was mentioned before, these separators are used when the amount of water is negligible compared to the other phases (water flow rate should be less than 20% of the total mass flow rate). Generally, the inlet diverters at the entrance are used to change the velocity and flow direction to help the bulk of liquids separate from the gas phase and move towards the bottom of the separators. At the next zone, some liquid droplets which were not separated in the first zone, have the opportunity to be separated from the gas phase due to gravity. It should be noted that the two continuous oil and water phases that accumulate at the bottom of the separator provide the required time to separate the gas phase from the liquid and also one liquid from the other liquid phase. Unlike the weir separators, water collects at the boot, not at the main vessel. So, the main vessel diameter of the boot separator is smaller than the weir separators (Pourahmadi, 2010). The length, main body diameter and the boot diameter of the studied separator are 11.9 m, 3.6m, and 1.5 m, respectively. To generate the grid for the geometry, the vessel was divided into different volumes, and a tetrahedral/hybrid scheme was used. The quality of the produced mesh based on the skew factor is shown in (Figure 2). Based on the results, only a few percentages of the cells (0.1%) have skew factor more than 0.8 which shows that the studied grid is of high quality. The mesh independence test in this study was performed by increasing the cell number until the same results were observed. In this study, the separator with 1,182,305 cells was selected as the case with the optimum cell number.

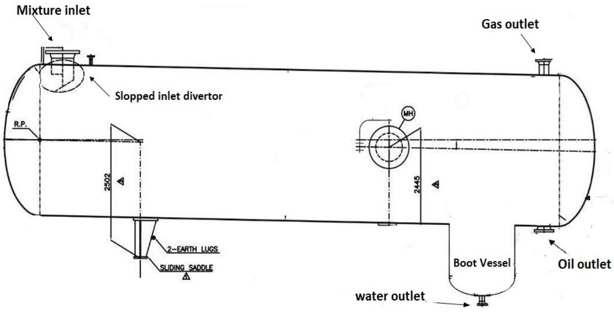


Figure 1. Schematic of the industrial boot separator.

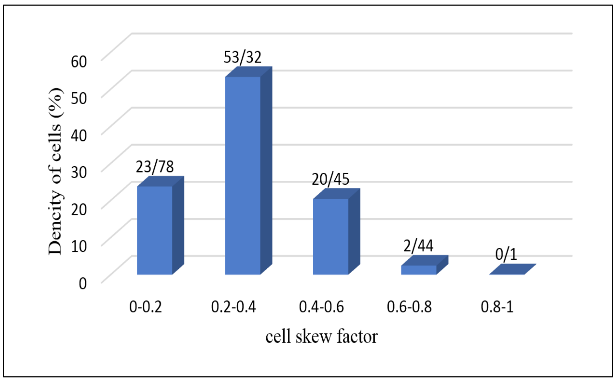


Figure 2. The quality of the produced mesh.

5. Boundary Condition

A velocity inlet for the inlet mixture and the pressure outlet for the gas phase were selected as the boundary condition types in the VOF model. For the liquids at the outlets, the velocity boundary type was utilized to control the interfaces between phases (Ghafarkhah et al 2017,2018; Pourahmadi et al 2011,2012). For the DPM model, the escape zone boundary condition was selected for the inlet and the outlets. In this model, the droplets which reach the wall surrounded by the liquid phases are assumed to be trapped, and those which reach the walls in the gas zone reflect and lose their momentum (Ghafarkhah et al 2017,2018; Pourahmadi et al 2011,2012).

6. Discretization and Numerical Methods

The equations used in the modeling of the separators were discretized using the finite volume method. The simple algorithm (Ghafarkhah et al 2017,2018; Ansys Fluent, 2016), was used in the Navier Stokes equation to couple the pressure and velocity. Turbulent kinetic energy parameters and the momentum equation were discretized using the second order upwind method. To interpolate the pressure at the numerical cell faces, the presto scheme due to the accordance with the VOF model was utilized (Ghafarkhah et al, 2017; Xu et al, 2016; Ansys Fluent, 2016).

7. Results and Discussion

In this research, the VOF model was used to show the total fluid flow profile on the macroscopic scale. To make a realistic simulation, the DPM model was coupled with the VOF model to track the droplets and investigate the microscopic behavior. So, to consider both the macroscopic and microscopic features of the separation process, the governing equations for both continuous and dispersed phases were solved simultaneously. The assumptions used in this study were, considering constant physical properties, 3-D model simulation, and turbulent flow. The simulation results based on three-phase fluid flow profiles, secondary phase behavior, separator performance, and droplet size distribution of the secondary phases and also the result for troubleshooting of the separation performance are as follows:

1.7. Three-phase fluid flow profiles

The CFD simulation results based on pressure, volume fraction contours and velocity vector are depicted in (Figures 3 to 5) to show the total fluid flow behavior. The simulation results based on the pressure contour in (Figure 3) show that the separator works at constant pressure (except for the variation due to the levels of the liquids) which this result is in a complete agreement with the industrial behavior (Pourahmadi et al, 2011; Mohammadi Ghaleni et al,2012). The oil volume fraction contour in (Figure 4), reveals that all the phases have been separated from each other by a clear interface because of the gravity force. The almost stratified gas-oil and oil-water interfaces predicted by the numerical calculations show the low foaming tendency in the studied separator.

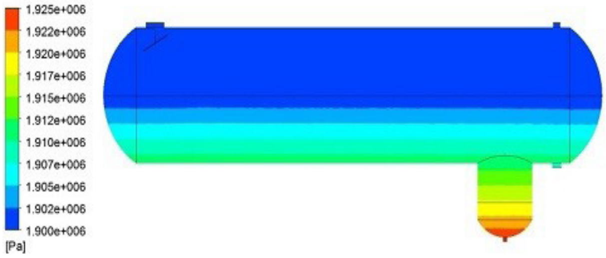


Figure 3. Contour of pressure in the separator.

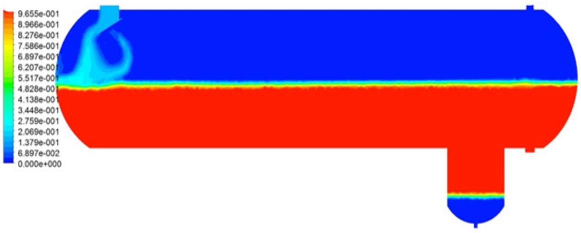
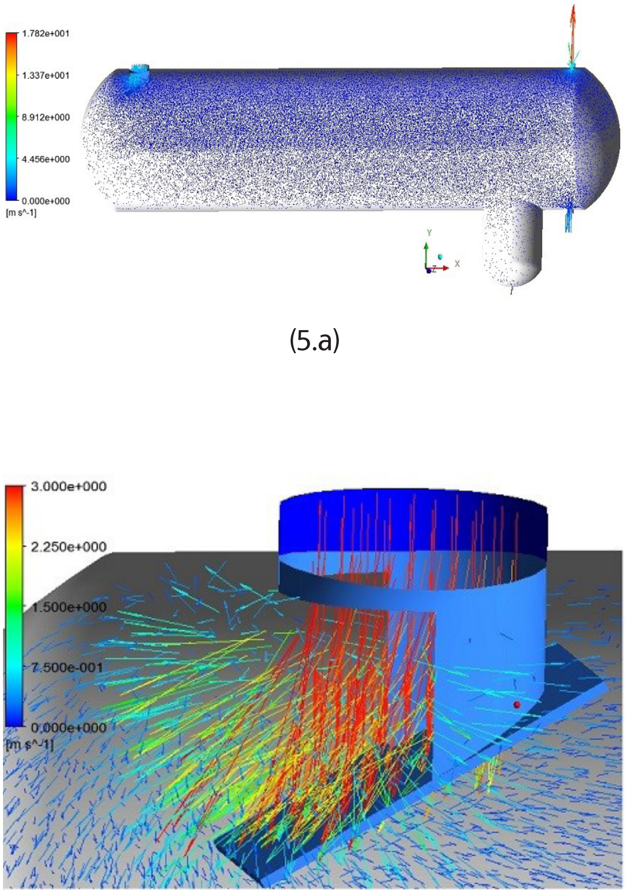


Figure 4. Contour of oil volume fraction.

The velocity vectors are drawn in (Figure 5). Based on (Figure 5-a), the velocity magnitude in the middle of the separator is much lower than the inlet and outlets. As it is more evident in (Figure 5.b), the flow direction changed and the velocity magnitude decreased by passing from the inlet diverter which shows the reduction in momentum flow. As it was mentioned, the main role of an inlet diverter is changing the flow direction and reducing the velocity magnitude to have a good separator performance (Pourahmadi, 2010), so the trend demonstrated in (Figure 5.b) shows that the used CFD model can appropriately predict the flow behavior by hitting the inlet diverter. To show the velocity magnitude along the separator with a high resolution, multiple vertical planes which are shown in (Figure 6), were modeled and the average velocity was recorded at each plane. The results of the velocity profile along the separator are illustrated in (Figure 7). In fact, the gas phase velocity should be decreased sufficiently from the inlet to the outlet (mostly at the first zone due to the existence of the inlet diverter) to help the droplets settle out by gravity easier due to more retention time of gas caused by low gas velocity along the separator (Ghafarkhah et al, 2017,2018; Pourahmadi, 2010). Thus, the decreasing trend in the velocity magnitude observed in (Figures 5 and 7) shows that the studied CFD model is good at evaluating the fluid flow profile in separators.



(5.a)

(5.b)

Figure 5. Velocity vector at (a): the separator (b): the entrance.

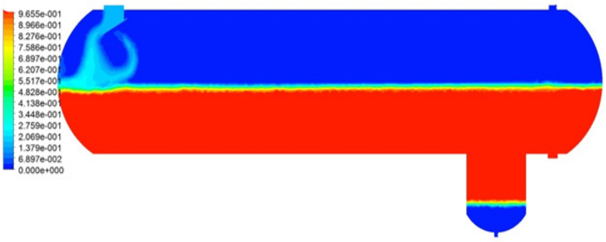


Figure 6. The planes which are modeled at the horizontal direction (x).

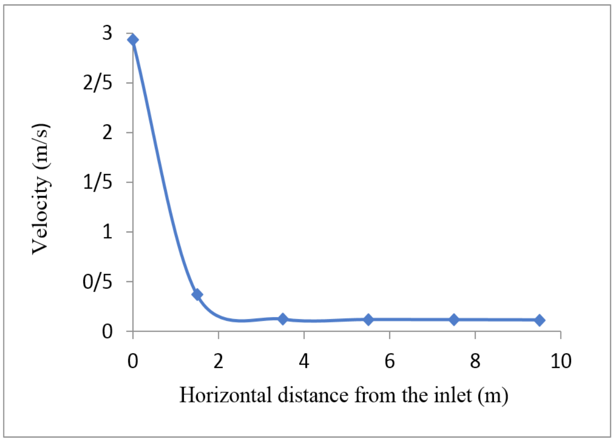


Figure 7. Velocity profile along the separator.

2.7. Secondary phase behavior

Kinetic energy which is completely related to the motion of the particles is a concept that can be used to evaluate the microscopic feature of the discrete phases in the separator. This feature can be investigated by tracking the droplets applying the DPM model (Ghafarkhah et al, 2017). The kinetic energy of the oil droplets in the gas-rich zone of the studied boot separator is presented in (Figure 8). In fact, in a separator, the kinetic energy of the droplets should be decreased from the inlet towards the outlet to let the droplets separate easier and have a good separator performance so that the presence of the liquid at the gas outlet be at the minimum amount (Ghafarkhah et al, 2017). As it is illustrated in (Figure 8), the kinetic energy of the droplets in the intended boot separator decreased from the inlet to the outlet and shows that the CFD model with a decreasing trend in the kinetic energy along the separator, is applicable of good estimating of the separator performance.

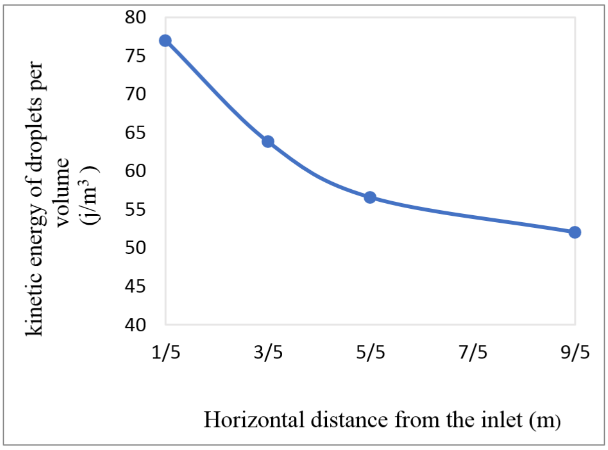


Figure 8. Kinetic energy of oil droplets.

3.7. Separation Performance

To validate the simulation results, the performance of the mentioned separator based on the presence of the oil droplets along the separator was considered. Noted that, due to the sufficient retention time of water in the boot, no problem can be seen in the liquid-liquid separation, and the main problem is the gas-liquid separation (Pourahmadi, 2010). Generally, the mass of liquid droplets should be decreased from the inlet to the outlet to have a good gas-liquid separation performance (Ghafarkhah et al, 2017). (Figure 9) illustrates the mass percentage of the oil droplets along the separator which is calculated by tracking the droplets using the DPM model. In fact, different vertical planes were modeled along the gas-rich zone (Figure.6) and the mass percentage of the droplets (mass of liquid droplets that reach each plane per total mass of the droplets at the inlet) was recorded at each plane. As it is seen, the mass percentage of the oil droplets along the separator decreased. The descending trend shown in (Figure 9), is in complete accordance with (Figure 8), i.e., decreasing the kinetic energy of the droplets. As mentioned earlier, decreasing the kinetic energy of the droplets let the droplets separate easier and reduce the mass of the liquid droplets from the inlet towards the outlet (Ghafarkhah et al, 2017). So, the complete accordance between (Figures 8 and 9), validates the use of the CFD model at estimating the quality of the separation.

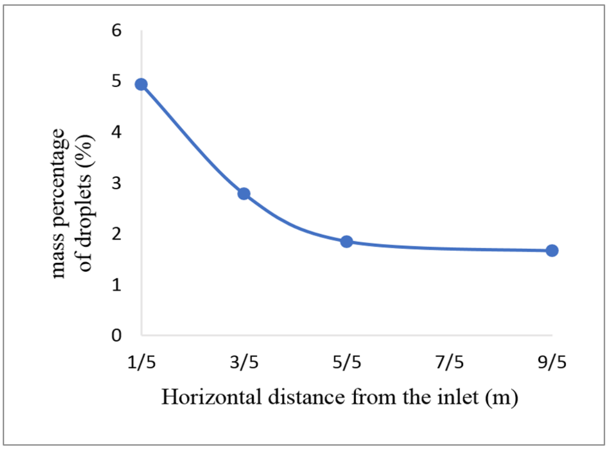


Figure 9. Mass percentage of oil droplets along the separator.

4.7. Droplet Size Distribution of the Secondary Phases

One of the important parameters to achieve the best quality of the separation process, except existing the minimum amount of liquid droplet mass at the gas outlet, is the appropriate droplet size distribution of the liquid droplets at the gas outlet. Appearing droplet size more than 100 µm shows that the separator does not work properly and suffers from the appropriate design (Arnold and Stewart, 2008; Pourahmadi, 2010). Tables 1 and 2 show the droplet size distribution of oil and water droplets at the gas outlet. As presented in Tables 1 and 2, the most percentage of both oil and water droplets have diameters less than 100 µm using the model.

Table 1. Droplet size distribution of oil droplets in the gas outlet

droplet size (µm)	25	45	63	83	102	122	141	160	179
Mass percentage of oil droplets (%)	5	30	30	19	8	4	2	1	1

Table 2. Droplet size distribution of water droplets in the gas outlet

droplet size (µm)	25	44	62	80	98	116	134
Mass percentage of water droplets (%)	12	45	20	16	3.5	2.5	1

5.7. Troubleshooting of the Separation Performance

As it was mentioned previously, CFD simulation has the priority over the experimental studies in that the internal flow behavior can't be investigated at each point using experimental works due to the technical problem in measuring the internal flow features and also because of the high experimental cost. So, the CFD simulation can be an economical method in investigating the internal flow to detect the imperfections of the separation process. Troubleshooting of the intended separator was investigated in this research by considering the fluid flow behavior using the CFD model. Considering the inlet diverter zone in the volume fraction contour shown in (Figure 4), revealed that the mixture flow (mostly the liquid phases which reached the interface) had a backward direction towards the gas-rich zone. This behavior shows that the momentum of the flow had not been sufficiently decreased by the used inlet diverter. So, the bulk of liquid can't effectively be separated at the first zone of the separator (the inlet diverter zone) and the performance of this zone will be reduced. Thus, changing the type of inlet diverter will be suggested. By a closer look at the oil outlet which is magnified and shown in (Figure 10), a small vortex is detected. A vortex can suck some gas from the gas rich-zone and re-entrain it in the oil outlet (Arnold and Stewart, 2008). So, the separator performance will be reduced by this phenomenon and causes the downstream equipment to encounter many problems. This problem can be overcome by implementing an appropriate vortex breaker.

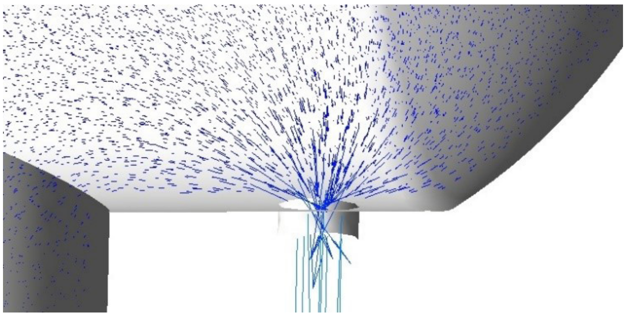


Figure 10. The velocity vector at the oil outlet.

Since the appropriate diameter distribution of the liquid droplet in the gas outlet is one of the key parameters which shows that the separator works properly, it should be investigated at each study to check the performance of the studied separator. In fact, appearing droplet size more than 100 μm shows that the separator does not work properly and suffers from the appropriate design (Arnold and Stewart, 2008; Pourahmadi, 2010) but droplets less than 100 μm in the gas outlet can be separated by applying an appropriate mist eliminator to improve the separation performance (Ghafarkhah et al, 2018; Pourahmadi, 2010). It should be noted that appearing droplets with diameter greater than 100μm might cause a flood in mist eliminator and damage it (Arnold and Stewart, 2008; Ghafarkhah et al, 2018). Based on the results of Tables 1 and 2, most of both oil and water droplets have a diameter less than 100 μm using the model. Thus, its performance might increase by applying an appropriate mist eliminator to reduce the liquid droplets at the gas outlet.

6.7. Effect of Altering the Inlet Water Flow Rate on the Separator Performance

AChanging the inlet flow rate is one of the important parameters which has been paid less attention while designing a separator. Totally, a separator should be designed so that changing the inlet flow rate (in a limited range based on the field experience) doesn't lead to a significant reduction in the separator efficiency. But generally, this is a problem in the industry which requires the separator to have a new design while changing the flow rate, which leads to paying the high cost (Ghafarkhah et al,2017,2018; Pourahmadi, 2010). In this section, the inlet flow rate of the water in the boot separator was changed to see its effect on the separator performance. The water flow rate was increased in the range of 11823-47295 kg/hr based on the field experience (until the water flow rate is less than 20% of the total mass flow rate in a boot separator). (Figure 11) shows the effect of increasing the water flow rate on the separation of the water droplets from the gas phase. It is depicted that increasing the inlet flow rate leads to an increase in the liquid droplets at the gas outlet which causes a decrease in the separation efficiency. In fact, Increasing the flow rate decreases the required retention time for separating the droplets from the gas phase, so an increase in the mass of liquid droplets at the gas outlet will be achieved (Mohammadi Ghaleni, 2012). Noted that although the increase in the water flow rate in the present separator leads to an increase in the liquid droplet mass at the gas outlet, the droplet mass at the maximum flow rate is just 1.6 kg/hr which doesn't significantly decrease the separator efficiency.

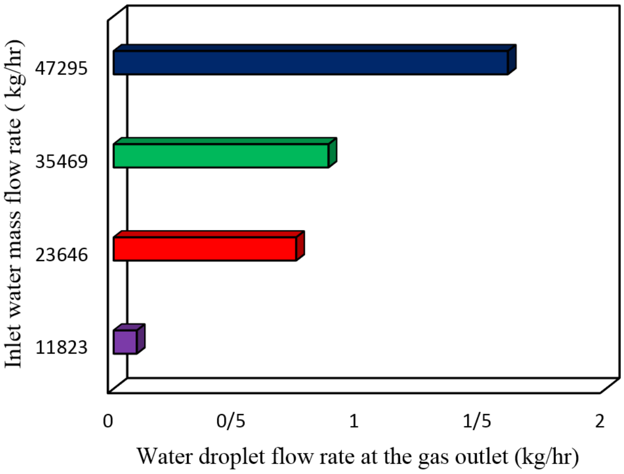


Figure 11. Effect of inlet water flow rate on the separator performance.

8. Conclusion

A 3-D VOF-DPM model was used in this research to show the macroscopic and microscopic features of the separation process in one industrial three-phase boot separator. The results of the fluid flow profiles in the macroscopic scale using the VOF model show that the separator works at constant pressure with a decrease in velocity magnitude by passing from the slopped inlet diverter towards the outlet. The results in the microscopic scale reveal the decrease in the kinetic energy of the droplets and also the decrease in the presence of liquid droplets from the inlet to the outlet. Results of the fluid flow profile and the complete agreement between the trends of the kinetic energy and the mass percentage of liquid droplets along the separator show that the CFD model is successful at good estimating of the separation process in the separator. Troubleshooting of the mentioned separator was another work that was considered in this research. Results demonstrated that due to the backward flow at the entrance and also because of the vortex detected at the oil outlet, the separator suffers from an appropriate design. So, applying an appropriate inlet diverter at the entrance and also a proper vortex breaker at the oil outlet were the suggestions at this research. The droplet size distribution of liquids at the gas outlet showed that the average diameter of both oil and water were less than 100 μm that requires an appropriate mist eliminator to increase the separator efficiency by omitting the liquid droplets at the gas outlet. The effect of increasing the water flow rate on the separator efficiency was also considered in this research. It is concluded that increasing the water flow rate from 11823-47295 kg/hr causes the increase in the water droplet mass at the gas outlet from 0.09 to 1.6 kg/hr, due to the reduction in the required retention time for separating droplets from the gas phase. Results demonstrate that although the increase in the water flow rate increases the water droplet mass at the gas outlet, due to less amount of water droplet mass at the gas outlet even at the maximum inlet water flow rate, a significant decrease in the efficiency can't be observed. So, the intended separator has the ability to change the inlet water flow rate in the range of 11823-47295 kg/hr without a significant decrease in the separation efficiency.

Nomenclature

C_D	Drag coefficient [-]
D	Pipe diameter of the flow [m]
dp	Particle diameter [m]
d_{max}	Maximum diameter [m]
d, d_{mean}	Mean of diameter [m]
F_D	Drag force [N]
$\vec{F}$	Source term force [N/m^2]
$\vec{f}$	Additional force per particle mass [m/s^2]
g	Gravity acceleration [m/s^2]
k	Turbulent kinetic energy [m^2/s^2]
P	Pressure [N/m^2]
$\vec{u}$	Velocity of fluid [m/s]
$\vec{u}_m$	Velocity of phase m [m/s]
u_p	Particle velocity [m/s]
α_m	Volume fraction of phase m [-]
ε	Turbulent dissipation rate [m^2/s^2]
μ	Molecular viscosity [pa.s]
μ_c	Molecular viscosity of continuous phase [pa.s]
μ_d	Molecular viscosity of dispersed phase [pa.s]
ρ	Density [kg.m^{-3}]
ρ_c	Density of continuous phase [kg/m^3]
ρ_d	Density of dispersed phase [kg/m^3]
ρ_p	Density of particle [kg/m^3]
μ_m	Density of phase m [kg/m^3]

Acknowledgements

The authors thank the Borzoyeh Petrochemical Company for providing all the required data in this research.

9. References

[1] Ahmed, T., Hamed, F., Russell, P.A., 2017. The use of CFD simulation to compare and evaluate different sizing algorithm for three - phase separator. OTC offshore technology conference. Brazil, 24-26.

[2] ANSYS Fluent version 16.2, 2016, Fluent Theory Guide.

[3] Arnold, K., Stewart, M., 2008. Surface production operations: design of oil handling systems and facilities. Amsterdam: Elsevier.

[4] Bothamley, M., 2013a. Gas/liquid separators: quantifying separation performance-part 1. Oil and Gas. Fac., 2 (4), 21-29.

[5] Bothamley, M. 2013b. Gas/liquid separators: quantifying separation performance-part 2. Oil and Gas. Fac., 2 (5), 35-47.

[6] Bracill, J.u., Kothe, D.B., Zemach, c., 1992. A continuum method for modeling surface tension. J.Comput.Phys., 100, 335-356.

[7] Cloete, S., Eksteen, J.J., Bradshaw, S.M., 2009 a. A mathematical modelling study of fluid flow and mixing in full scale gas stirred ladles. Computational Fluid Dynamics, 9(6), 345-356.

[8] Cloete, S., Olsen, J.E., Skjetne, P. 2009b. CFD modeling of plume and free surface behavior resulting from a sub-sea gas release. Applied Ocean Research. 31, 220-225.

[9] Ghafarkhah, A., Shahrabi, M.A., Moraveji, M.K., Eslami, H., 2017. Application of CFD for designing conventional three phase oilfield separator. Egypt. J. Pet., 26 (2), 413-420.

[10] Ghafarkhah, A., Shahrabi, M.A., Moraveji, M.K., Eslami, H., 2018. 3D Computational-Fluid-Dynamics Modeling of Horizontal Three-Phase Separators: An Approach for Estimating the Optimal Dimensions. Oil and Gas. Fac., 33 (4), 1-17.

[11] Khalifat, Z., Zivdar, M., Rahimi, R., 1396. Simulation of three-phase separator in Borzoye petrochemical company, the first conference in science and engineering, Mashhad, in Persian.

[12] Kharoua, N., Khezzar, L., Saadawi, H., 2013b. CFD simulation of three-phase separator: effects of size distribution. ASME FEDSM. Nevada, USA.

[13] Kharoua, N., Khezzar, L., Saadawi, H., 2013a. CFD Modelling of a Horizontal Three-Phase Separator: A Population Balance Approach. Am. J. Fluid Dyn., 3 (4), 101-118.

[14] Kirveski, L., 2016. Design of Horizontal three-phase separator using computational fluid dynamics. MSC Dissertation, Alato university school of chemical technology.

[15] Mohammadi Ghaleni, M., Zivdar, M., Nemati, M.R., 2012. Hydrodynamic Analysis of two-phase separator by computational fluid dynamic (CFD). 6th international conference on Advanced Computational Engineering and Experimenting. Istanbul, Turkey.

[16] Monnery, W.D., Svrcek, W.Y., 1994. Successfully specify 3-phase separators. Chem. Eng. Prog, 90 (6), 29-40.

[17] Pourahmadi Laleh, A., 2010. CFD Simulation of Multiphase Separators. Ph.D. Dissertation, University of Calgary, Canada.

[18] Pourahmadi Laleh, A., Svrcek, W.Y., Monnery, W.D., 2011. Computational Fluid Dynamics Simulation of Pilot Plant-Scale Two-Phase Separators. Chem.Eng.Tech., 34 (2), 296-306.

[19] Pourahmadi Laleh, A., Svrcek, W.Y., Monnery, W.D., 2012. Computational Fluid Dynamics-Based Study of an Oilfield Separator--Part I: A Realistic Simulation. Oil and Gas Fac., 1(6), 57-68.

[20] Qarot, Y.F., Kharoua, N., Khezzar L., 2014. Discrete phase modeling of oil droplets in the gas compartment of a production separator. ASME International Mechanical Engineering Congress and Exhibition, Canada.

[21] Xu, Y., Liu, M., Tang, C., 2013. Three-dimensional CFD-VOF-DPM simulations of effects of low-holdup particles on single-nozzle bubbling behavior in gas-liquid-solid systems. Chem.Eng, 222, 292-306.

کاربرد دینامیک سیالات محاسباتی به منظور عیب یابی و تحلیل رفتار هیدرودینامیکی در یک جداکننده گرانشی سه فازه صنعتی

- زهره خلیفات[1]، مرتضی زیودار[2*]، رهبر رحیمی[3]

1. دانشجوی دکتری مهندسی شیمی، گروه مهندسی شیمی، دانشگاه سیستان و بلوچستان، زاهدان، ایران
2. دانشیار گروه مهندسی شیمی، دانشگاه سیستان و بلوچستان، زاهدان، ایران
3. استاد گروه مهندسی شیمی، دانشگاه سیستان و بلوچستان، زاهدان، ایران

(ایمیل نویسنده مسئول: mzivdar@eng.usb.ac.ir)

چکیده

جدایش جریان چند فازی در جداکننده های گرانشی یکی از فرایند های مهم در صنایع مختلف به شمار می رود. در این مطالعه از یک شبیه سازی دینامیک سیالات محاسباتی (CFD) با استفاده از یک مدل ترکیبی VOF-DPM برای عیب یابی و تحلیل رفتار هیدرودینامیکی فرایند جدایش در یک جداکننده سه فازی صنعتی دارای بوت، استفاده شده است. لازم به ذکر است که با وجود کاربرد گسترده جداکننده های بوت در صنایع مختلف تاکنون هیچ پژوهشی روی این نوع از جداسازها برای بررسی رفتار ماکروسکوپی و میکروسکوپی فرایند جدایش ارائه نشده است. در این پژوهش، نتایج محاسبات عددی بر حسب پروفایل های جریان سه فازی، رفتار فازهای ثانویه ، بازده جداکننده و توزیع اندازه قطرات مورد بررسی قرار گرفته است. نتایج نشان داد که مدل CFD مورد نظر قادر به تخمین خوبی از رفتار جدایش در یک جداکننده سه فاز دارای بوت است. عیب یابی فرایند جدایش در جداکننده مورد نظر برای تشخیص پارامترهایی که منجر به کاهش راندمان جداکننده می شود نیز مورد بررسی قرار گرفته است. نتایج نشان دهنده کاربرد نامناسب نوع هدایت کننده ورودی، نیاز به وجود نم گیر مناسب در خروجی گاز و همچنین نیاز به یک گرداب شکن در خروجی مایع بوده است. اثر افزایش میزان دبی آب ورودی روی عملکرد جداساز، یکی از پارامترهای مهم دیگری است که در این مطالعه مورد بررسی قرار گرفته است. نتایج نشان داد که افزایش دبی آب ورودی در محدوده ۱۱۸۲۳-۴۷۲۹۵ kg/hr منجر به افزایش میزان جرم آب در خروجی گاز از ۰/۰۹ به ۱/۶ kg/hr شده است ولی این مقدارافزایش باعث کاهش چشمگیری در عملکرد جدایش در جداساز مورد نظر نشده است.

واژگان کلیدی: جدا کننده گرانشی سه فازی، مدل حجم سیال ، مدل فاز ناپیوسته، عیب یابی، تحلیل هیدرودینامی