JOURNAL OF GAS TECHNOLOGY
Volume 6 / Issue 1 / Summer 2021 / Pages 30-42
Journal Homepage: http://jgt.irangi.org

Simulation of an Industrial Three Phase Boot Separator Using Computational Fluid Dynamics

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

1. Ph.D student, Department of Chemical Engineering, Faculty of Engineering, University of Sistan and Baluchestan, Zahedan, 98161, Iran
2. Corresponding Author: Department of Chemical Engineering, Faculty of Engineering, University of Sistan and Baluchestan, Zahedan, 98161, Iran
3. Department of Chemical Engineering, Faculty of Engineering, University of Sistan and Baluchestan, Zahedan, 98161, Iran

ARTICLE INFO

ORIGINAL RESEARCH ARTICLE

Article History:
Received: 28 April 2021
Revised: 26 May 2021
Accepted: 11 June 2021

Keywords:
Computational fluid dynamics
Multi-phase flow
simulation
Three phase boot separator
Discrete random walk model

DOR: 20.1001.1.25885596.2021.6.1.3.0

ABSTRACT

Three-phase separators are used to separate immiscible phases in petroleum industries. Computational fluid dynamics (CFD) simulation of industrial separators are rather limited in the literature and most of them are based on Eulerian-Eulerian (E-E) or Eulerian-Lagrangian (E-L) approaches with poor agreement between simulation and industrial data. In this research a coupled E-E and E-L method, i.e., the combination of the volume of fluid (VOF) and dispersed phase model (DPM) was developed to simulate an industrial three phase boot separator. Noted that despite the wide usage of boot separators in petroleum industry, no research has been performed on it. In order to develop the coupled model, effects of different sub-models including virtual mass force, droplet break up and also discrete random walk (DRW) model which was ignored in most of the researches, were considered. Liquid droplet entrainment in the gas outlet taken from data of Borzoyeh Petrochemical Company in the south of Iran, was the criteria for evaluating the CFD model. It is concluded that the coupled model using three mentioned sub-models with the high importance of applying DRW, is a successful way in predicting the separator efficiency so that considering all sub-models decreases the simulation error from 40.81% to 12.9%. Using the validated model, effects of inlet droplet size and flow rate on the separation performance were considered. Results demonstrated that decreasing droplet size (by 20%) and increasing flow rate (from 5800-6475 kg/hr), decreased the efficiency, such that the liquid entrainment in the gas outlet increased by 29% and 38 % respectively.

How to cite this article
Z. Khalifat, M. Zivda, R. Rahimi. Simulation of an Industrial Three Phase Boot Separator Using Computational Fluid Dynamics. Journal of Gas Technology. 2021; 6(1): 30 -42. (http://jgt.irangi.org/article_251664.html)

---

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

1. Introduction

Three phase gravity separators are the most important facilities which are widely used in petroleum industries to separate immiscible phases (Pourahmadi et al, 2012; Mostafaiyan et al, 2014). These separators have been developed in both vertical and horizontal orientations. Horizontal types used for high gas to liquid ratio mixtures are more common in Iran and can be categorized in two most important groups, i.e., weir type (when the water fraction is substantial) and boot type (when the water fraction is not substantial) (Pourahmadi et al, 2012). Inappropriate design of such equipment leads to inefficient separator performance, in which gases carry some liquid droplets whilst some gas bubbles are entrained by the liquid phases at the outlet. So, impure separated phases damage downstream equipment such as pumps and compressors (Pourahmadi et al, 2012; Qarot et al, 2014). Generally, separator designing is based on semi-empirical methods, but because of simplified assumptions such as not considering the effect of turbulence and internals, these methods are not completely acceptable (Monnery and Svrcek ,1994; Bothamley, 2013). Although the experimental study is a solution to this problem, the high-performance cost forces the researchers to use a more economical method, i.e., computational fluid dynamics (CFD) to modify the design problem and also debottleneck the separators (Ghafarkhah et al, 2017, 2018; Mc cleney et al, 2017; Kharoua et al, 2013b).

Exact Separator modeling using CFD is a complicated process which needs careful consideration to describe physical phenomena and estimate the separator efficiency well. So, investigating an appropriate CFD model leads to a powerful tool to aid in separator designing and also debottlenecking of the existing separators (Mc cleny et al, 2017). Two common strategies used in multiphase flow modeling are Eulerian-Eulerian (E-E) and Eulerian-Lagrangian (E-L) methods. In the E-E approach, all the phases are considered as continuous phases which interact with each other by solving the Navier-Stokes equation. E-E approach includes Volume of fluid (VOF), mixture and Eulerian models. Discrete phase model (DPM) which belongs to the model in the E-L approach, includes both continuous and discrete phases. In this approach, the Navier-Stokes equation is solved for the continuous phase while the discrete phases are tracked based on Newton's second law (Xu et al, 2013).

CFD simulation of industrial three-phase separators that their results have been compared with experimental data are rather limited in the literature, and most of them are usually based on the E-E or E-L method with low accuracy in estimating separator efficiency.

Kharoua et al. (2013 b) used Eulerian with k-Ɛ model for simulating an industrial weir separator. Because of considering a single average diameter for secondary phases (oil and water) and not taking in to account the droplet interaction, i.e., coalescence and breakup, the separator performance based on the mass of liquid droplets in the outlet were in a very poor agreement with field data.

Ahmed et al. (2017) used Eulerian model in one pilot separator with weir. Because of the limitations mentioned in Kharoua's work (Kharoua, 2013 b), a high error (35 to 50 %) were observed.

In another study performed by Kharoua et al. (2013 a) size distribution, coalescence and break up were considered using population balance model (PBM) coupled with the Eulerian model. Although this model revealed the importance of droplet size distribution and the results were in a better agreement with experimental data, because of the limitation of size distribution for just one secondary phase, the results again were not in a good agreement (about 50 to 85% error) with field data.

It should be noted that in spite of good estimating of some features like velocity and pressure profiles using models in E-E approach, these models are not capable of good estimating of separator efficiency (Qarot et al, 2014; Kharoua et al, 2013a, 2013b; Ahmed et al, 2017). In fact, in addition to the limitations pertinent to droplet size and interaction in Eulerian and mixture models, these models face problems in modeling the interfaces between phases. The VOF model, however, is special in tracking of sharp interfaces, but this model needs to track free surface around each droplet. Therefore, a prohibitively fine grid resolution is required. So, VOF model is not economical to be used in industrial scales. DPM can be a remedy to track droplets, where droplets are treated as point sources of momentum moving in the domain. Also, the size distribution can be used for all the secondary phases in this model (Cloete et al, 2009b; Kirveski, 2016), but both continuous oil and water phases accumulated on the bottom of the separators are neglected in DPM model, and this leads to abnormal results in separator efficiency (Pourahmadi et al, 2011). So, DPM model requires three background phases in three-phase separators to interact with droplets (Qarot et al, 2014; Pourahmadi et al, 2011). As VOF is exact in tracking interfaces between continuous phases, it is a candidate to be coupled with DPM model (Qarot et al, 2014; Cloete et al, 2009b). In spite of completely acceptable coupled VOF-DPM model in multiphase flow (Cloete et al, 2009 a, 2009 b), there are very limited researches which used this model in three-phase separators.

Pour Ahmadi et al. (2011,2012) applied a VOF-DPM with k-Ɛ model in a field separator with weir to debottleneck it for a better separation. Size distribution, coalescence and breakup were modeled for the secondary phases. It is important to note that the model was not validated due to the lack of experimental data for the studied three phase separator. The mesh independency test was another important factor which was neglected in their work. The model details for the CFD simulation to show which sub-model has the considerable effect, were not also investigated in this paper. The most important point in this study is neglecting the discrete random walk (DRW) model, which shows the effect of turbulence on the particle movement in the separators. Therefore, the CFD model is not exact for three-phase separators. In their work, in the absence of industrial data, the model was just validated with four two-phase small laboratory scale separators, which demonstrated a reasonable agreement between the mass distribution of liquid droplets in the outlet obtained by the model and experimental data. In fact, the defections of their work should be corrected to have a reliable CFD model. It should be noted that in the present work, all the mentioned corrections have been implemented in order to simulate an industrial three-phase boot separator.

Ghafarkhah et al. (2017, 2018) presented a VOF-DPM with the k-Ɛ model to compare two different semi-empirical methods in designing three-phase weir separator and also to evaluate its performance. Results showed that this model was good in estimating the appropriate dimensions of the separator.

Although three-phase boot separators have wide usage in petroleum industries, to the best of our knowledge no research has been performed by using CFD to investigate their performance. Gawas (2013) showed that multiphase flow behavior becomes different in three-phase flow (oil, water and gas) when the amount of water changes, due to different interaction between liquid phases. There is different multiphase behavior in boot separators compared to weir separators. Boot separators were not considered by any researcher, and should be addressed separately to evaluate their performance. Developing an appropriate model is the prerequisite for this aim to aid in separator design, and also debottlenecking the separators. Due to the poor agreement between the simulation and industrial data by using the E-E and E-L approaches, and because of the advantages of VOF-DPM model mentioned before, this model was selected in this work. Because of the limited works pertinent to this coupled model in three-phase separators, the model details and sub-models that lead to the best simulation results, have not been noticed in the literature yet. The aim of this work is to develop a comprehensive VOF-DPM model, and consider the effects of some sub-models, i.e., virtual mass force, breakup model, and also DRW model. This approach which was ignored in most of the separator modeling, was used to establish a suitable methodology for a realistic simulation of an industrial boot separator. The simulation results were compared with industrial data of a boot separator located in Borzoyeh Petrochemical Company in the south of Iran. Results demonstrated that the coupled model with the sub-models is capable of estimating three-phase boot separator efficiency very well. Also, the importance of applying DRW was highlighted in this research. By using the validated model, the effects of droplet size and flow rate on the separator efficiency were also investigated in this research.

2. Computational Fluid Dynamics Model

VOF model is used to create the background to show the total fluid flow profiles of three phases (oil, water and gas) and track their interfaces. Since the VOF model is not capable of tracking the droplets at an affordable grid resolution in industrial scale (Cloete et al, 2009b; Kirveski, 2016), DPM model is coupled with VOF to track droplets and force them to interact with the phases in the background through the momentum equation. For simulation, the commercial code, Ansys Fluent 16.2, was used.

2.1. Coupled VOF-DPM model

VOF model solves continuity equation for each phase and just one momentum equation with a shared velocity field for all the phases. The continuity and the momentum equations are shown respectively as (Cloete et al, 2009b):

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

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

Where α , ρ and $\vec{u}$ are the volume fraction, density and continuous phase velocity respectively. The subscript m is the representative of phase m. P is pressure; g is gravity acceleration and τ represents the shear stress. Noted that surface tension force between phases was considered as a source term ($\vec{F}$) by applying continuum surface force model proposed by Brackbill et al. (1992).

Tracking droplet is predicted by implementing force balance on each droplet using DPM model (Cloete et al, 2009b; fluent theory guide, 2016):

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

The subscript p is assigned for the particle. The particle acceleration is due to drag, gravity and additional forces. All the additional forces, $\vec{f}$, except virtual mass force were neglected (Ghafarkhah et al, 2017). Virtual mass force is introduced when the continuous phase is accelerated due to discontinuous phase motion. This force is (Cloete et al, 2009b; Saffari and Dalir, 2012):

$$F_{\vartheta_m} = \frac{1}{2}\frac{\rho}{\rho_p}\frac{d}{d_t}\left(\vec{u} - \overrightarrow{u_p}\right) \tag{4}$$

The drag force (F_D) and drag coefficient (C_D) are (Cloete et al, 2009b):

$$F_D = \frac{18\mu C_D\, Re}{24\rho_p {d_p}^2} \tag{5}$$

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \tag{6}$$

μ is the molecular viscosity and d_p is the particle diameter. a_1,a_2 and a_3 are constant in several ranges of Reynolds based on Morsi and Alexander method (Cloete et al, 2009b; Huang et al, 2018). Turbulence was modeled using the standard k-Ɛ model because of its simplicity in actual operating condition and industrial scale and also due to its suitable application in flow involving separation (HSU et al, 2017: Zhang et al, 2018).

Noted that movement of the particles is affected by the velocity of the continuous phases in the background which is included in the drag term of equation 3. In fact, in turbulent flow, the velocity of the background phase denotes as (Ghafarkhah et al, 2017, 2018):

$$u = \bar{u} + u' \tag{7}$$

Since the particle motion and dispersion is also influenced by the velocity fluctuation (u') not solely by the average velocity ($\bar{u}$), the stochastic tracking model, i.e., DRW model was applied to investigate the fluctuation effect on the particle movement. u' in this model is (Ghafarkhah et al, 2017):

$$u' = G\sqrt{\frac{2k}{\varepsilon}} \tag{8}$$

G is a random number which is distributed normally and remains constant during the time that the droplet passes through a turbulent eddy. This time is estimated as (Fluent theory guide, 2016; Cloete et al, 2009b):

$$\tau_e = c_L \frac{k}{\varepsilon} \tag{9}$$

k is turbulent kinetic energy and Ɛ is turbulent dissipation rate. c_L, which is constant is recommended to be 0.15 in the k-Ɛ model, but it can be altered as a tuning parameter (Fluent theory guide, 2016; Cloete et al, 2009b).

The Taylor Analogy Break up (TAB) model is used for the droplet break up. It is based on the analogy between an oscillating and distorting droplet and a spring-mass system. The resulted equation is (Kongre et al, 2010):

$$C_F \frac{\rho_c \, {u_p}^2}{\rho_{d.\,r}} - C_k \frac{\sigma}{\rho_d \;\, r^2}\, x - C_d \frac{\mu_d}{\rho_d \, r^2} = \frac{d^2 x}{dt^2} \tag{10}$$

x is the displacement of the droplet from its spherical position and σ is the surface tension. The subscripts c and d are assigned for continuous and dispersed phases. C_F, C_k and C_d are dimensionless constants. The droplet is assumed to break up if (Fluent theory guide, 2016):

$$x > C_b \, r \qquad , C_{b=0.5} \tag{11}$$

2.2. Geometry and material definition of the boot separator

The industrial three-phase boot separator which contains one slopped inlet diverter of 30° at the entrance, a gravity separation zone and a water boot at the bottom of the vessel is depicted in Fig. 1. These separators are used when the water flow rate is very low relative to the other phases (Pourahmadi, 2010). Generally, an inlet diverter is used at the entrance to change the flow direction and reduce the velocity abruptly to separate the bulk of liquid from gas. After the entrance zone, the dispersed liquid droplets which were not separated at the entrance, settle out of the gas due to gravity. The liquid collected at the bottom of the separator, provides the retention time for separation of gas bubbles from liquids and also for separation of two liquids from each other. Unlike the weir separators, water that settles out in the liquid section is collected in the boot to provide the retention time to separate oil from water. Thus, the main body diameter of the boot separators can be smaller relative to weir separators (Pourahmadi, 2010). The separator studied here is 11.9 m long, with the main body diameter of 3.6 m and the boot diameter of 1.5m. The feed inlet diameter is 0.51 m and the outlet diameters of gas, oil and water are 0.2, 0.25 and 0.1 m respectively. A tetrahedral/hybrid scheme shown in Fig. 2 for one segment of the domain was used to generate mesh for the separator.

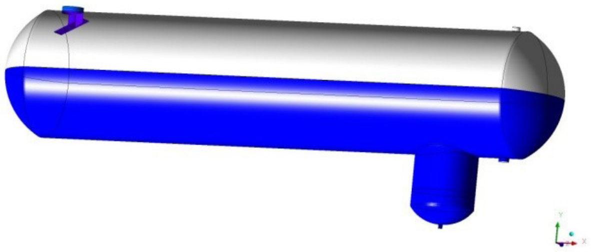

Figure 1. Three- dimensional model of the boot separator

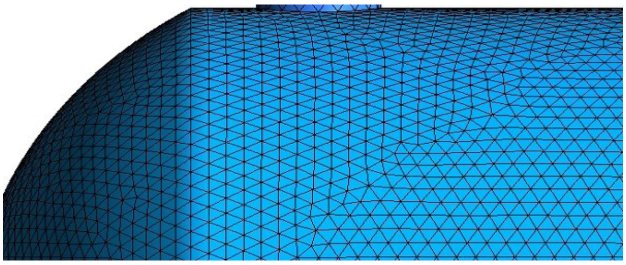

Figure 2. Grid type in one segment of the domain

The feed which was the mixture of hydrocarbon and water was simulated in Aspen Hysys v.9 to calculate the volume flow rate and physical properties of each phase. The results are shown in Table 1.

Table 1: Physical properties and volume flow rate of the feed in the boot separator at a temperature of 47°C and pressure of 19 bar

Phase	Density (kg/m³)	Viscosity (kg/m.s)	volume flow rate (m³/hr)
Gas	3.283	9.332e-6	1793
Liquid hydrocarbon (oil)	692.6	3.685e-4	340.1
Water	991.1	5.783e-4	11.93

The size distribution of oil and water droplets used in the DPM model were determined based on logarithmic Rosin-Rammler equation which is a conventional representative of droplet size distribution (Johansen et al, 2013).

$$Y_{(d)} = 1 - exp\left(\frac{-d}{\bar{d}}\right)^{n} \tag{12}$$

Where, $Y_{(d)}$ is the mass fraction of droplets, n is the spread parameter which describes the material uniformity and d is the particle diameter [26].

A comprehensive study that considers all the physical properties of the droplets showed that equations 13 and 14 could be used to determine the maximum and mean of oil and water droplet size (Pourahmadi, 2010).

$$d_{max} = 1.38\left(\frac{\sigma^{0.6}}{{\rho_c}^{0.3}{\rho_d}^{0.2}{\mu_c}^{0.1}}\right)\left(\frac{D^{0.5}}{{u_c}^{1.1}}\right) \times \tag{13}$$

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

$$d_{mean} = \ 0.4\ d_{max} \tag{14}$$

In this work, the maximum, minimum and mean diameter of oil and water are 3461,150,1384 µm and 6833,150,2733 µm, respectively. The spread parameter for both liquids were set as 2.6 (Ghafarkhah et al, 2017, 2018; Pourahmadi et al, 2011,2012).

2.3. Boundary condition and numerical solution

For the VOF model, a velocity inlet boundary condition and a pressure outlet for the gas outlet were set. To control the interfaces between gas-oil and oil-water phases, the velocity boundary type was utilized at both oil and water outlets (Ghafarkhah et al, 2017, 2018; Pourahmadi et al, 2011,2012). Turbulent parameter at each boundary was determined using turbulent intensity as (Pourahmadi et al, 2011):

$$I = \ .16\ Re^{-.125} \tag{15}$$

For the DPM model, the scape zones were selected for the inlet and outlets. It was assumed that the droplets which reach the walls surrounded by oil and water zones are trapped while those that reach the other walls reflect (Ghafarkhah et al, 2017, 2018; Pourahmadi et al, 2011,2012).

Discretization of the equations was performed using the finite volume method. The velocity-pressure coupling was utilized using the simple method (Ghafarkhah et al, 2017, 2018). The second order upwind method was selected to discretize the turbulent parameters and also the momentum equation. The presto scheme was chosen for the pressure interpolation in this work (Ghafarkhah et al, 2017, 2018; Fluent theory guide, 2016).

2. Results and Discussions

The coupled VOF-DPM model was selected for modeling of a three-phase boot separator. In this study the constant physical properties, unsteady DPM model, three-dimensional simulation and turbulent flow assumption were considered. The results of the model are as follows:

3.1. Grid test

The grid test was performed based on velocity profile and also the most important parameter in separator i.e., mass of liquid droplets that should be separated (Ghafarkhah et al, 2017,2018), to ensure the results. As an example, the analyze depicted in Figure. 3 was based on mass distribution of droplets that is necessary to be estimated for evaluating the separator performance. In this step, different vertical planes were modeled in different horizontal distance from the inlet and mass percentage of liquid (mass of liquid droplets that reaches each plane per total mass of droplets at the entrance) were recorded. Three different cell numbers were tested in this study and case 2 with 1182305 cells was selected due to not significant change from case 2 to case 3.

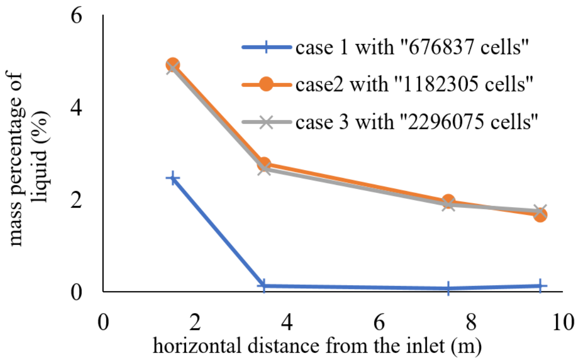

Figure 3. Grid test for mass percentage of liquid

3.2. Fluid flow profile

Figure. 4 depicts the volume fraction contour of the oil phase in the right view of the separator. The contour of density in the middle of the separator is also shown in Figure.5. As represented on Figures 4 and 5, three phases have been separated due to gravity by a clear interface at gas-oil and oil-water interfaces. The contour of pressure in the middle of the separator in Figure. 6 reveals that the separator works at constant pressure (except for variation due to liquid level) which is in complete accordance with the industrial data (Pourahmadi et al, 2011). Velocity vectors of the continuous phases at the entrance of the separator shown in Figure. 7 demonstrates that the flow direction changes and also the velocity magnitude reduces by passing from the slopped inlet diverter to the main part. To a better illustration, the velocity profile in the main part of the vessel are shown in different vertical planes located in the horizontal direction (x-direction) in Figure. 8 to 10. In fact, in separators, the velocity magnitude of the gas phase should be decreased sufficiently to let the droplets drop out by gravity easier due to more retention time of gas (Ghafarkhah et al, 2017, 2018; Pourahmadi et al, 2011). This trend is in accordance with the results shown in Figure. 7 to Figure. 10. So, the results show that this model is good in depicting the fluid flow profile.

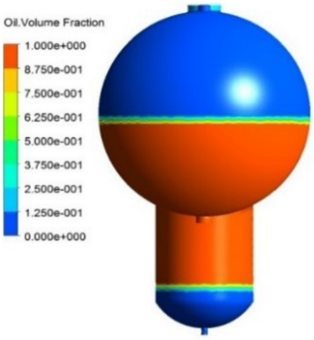

Figure 4. Contour of oil volume fraction

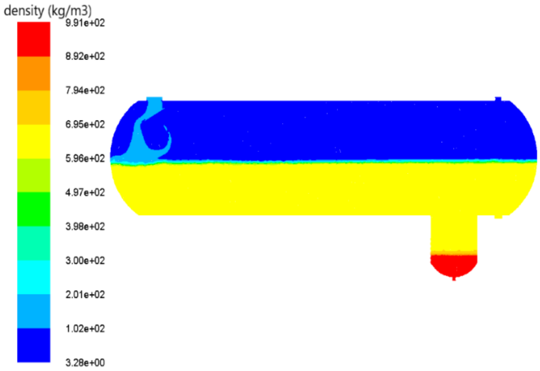

Figure 5. Contour of density in the middle of the separator

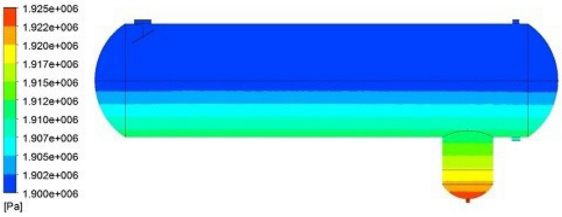

Figure 6. Contour of pressure in the middle of the separator

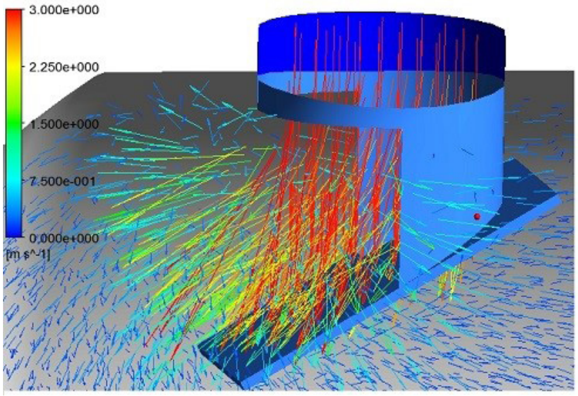

Figure 7. Velocity vector at the entrance

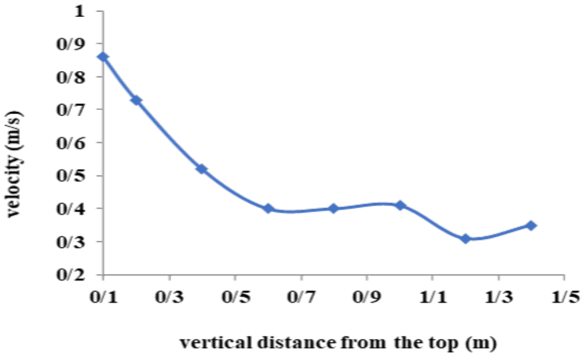

Figure 8. Velocity profile in x=1.5 m

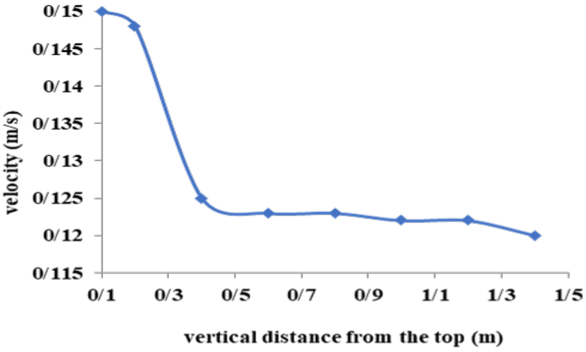

Figure 9. Velocity profile in x=5.5 m

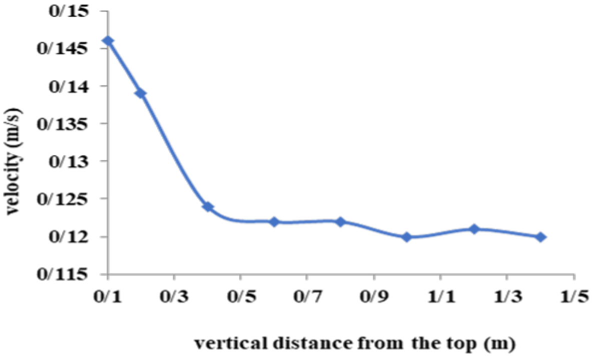

Figure 10. Velocity profile in x=9.5 m

3.3. Separator performance and data validation

The results of the simulation and industrial data were compared in Figure 11. In order to develop the appropriate model, effects of different sub-models including virtual mass force, DRW model and droplet break up, were considered. It should be noted that because of low volume fraction of water in the boot separators, no problem can be seen in liquid-liquid separation and the main problem is separating liquid droplets from the gas phase (Pourahmadi et al, 2011). The mass of oil droplet in the gas outlet were detected based on ASTM D1945 in Borzoyeh petrochemical company and were used to be compared with the model results. It can be observed from Figure. 11 that inclusion of the virtual mass compared to neglecting all the sub-models, shows a better prediction with experimental data. Despite the fact that the amount of virtual mass is significant only when the gas phase is dispersed, considering it in multiphase flow might modify the results to agree better with experimental data even if the gas phase is continuous (Saffari and Dalir, 2012). In this study the error of model decreased by 7.58% using virtual mass so that the simulation error relative to experimental data decreased from 40.81% to 33.23%. As depicted in Figure. 11, neglecting the effect of DRW model both in the presence and absence of virtual mass, decreases the amount of liquid at the gas outlet significantly. The reason is that neglecting DRW model leads to neglect velocity fluctuation in the background and as movement of droplets based on equations 3 and 10 is affected by this factor not solely by the average velocity, the results differ when neglecting it. So, it causes the mass of liquid droplets to go towards an ideal situation i.e., existing lower amount of liquid in the gas out let. In this research the simulation error using DRW model decreased from 33.23% to 16.6%. Thus, the model without DRW can't effectively estimate the separator performance which researchers paid less attention to it. Figure 11 shows that the error has been decreased using break up model. In fact, because of producing droplets with different size while using breakup model, the movement path of droplets is affected, so the results become different while neglecting it (Qarot et al, 2014). However, in this research break up model just decreased the error by 3.71% such that the simulation error relative to experimental data decreased from 16.6% to 12.9%. The error between simulation results and industrial data (12.9%) in this work relative to the previous studies (Kharoua et al, 2013 a, b, Ahmed et al, 2017) showed a reduction of at least 22.1%. Therefore, the coupled model used in this study is capable of estimating the boot separator performance very Comparison of different sub-models with experimental datal.

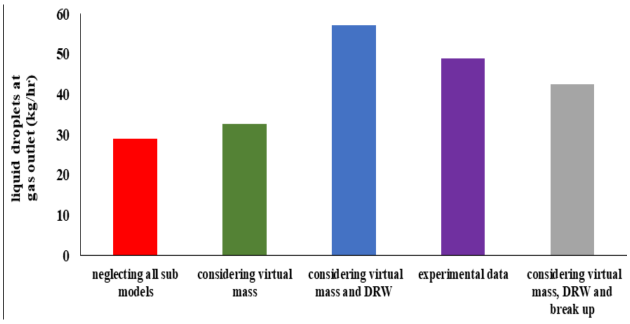

Figure 11. Comparison of different sub-models with experimental data

Note that, in addition to existing low amount of liquid droplets in the gas out let to have a good separator efficiency, appropriate diameter distribution of liquid droplets in the gas out let is of high importance since droplets less than 100 μm can be separated by applying an appropriate mist eliminator while those greater than 100μm might flood in mist eliminator and damage it (Ghafarkhah et al, 2018; Pourahmadi, 2010). So, size distribution at the out let should be checked to investigate the separator performance. As presented in Figures. 12 and 13., most of both oil and water droplets have a mean diameter between 10 and 100 μm using the model. Thus, its performance might increase by applying an appropriate mist eliminator to reduce the liquid droplets at the gas out let.

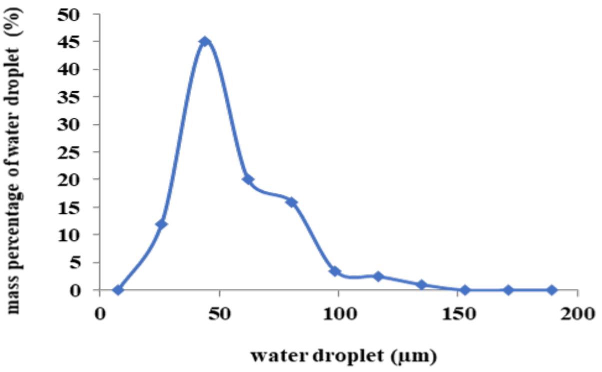

Figure 12. Water size distribution

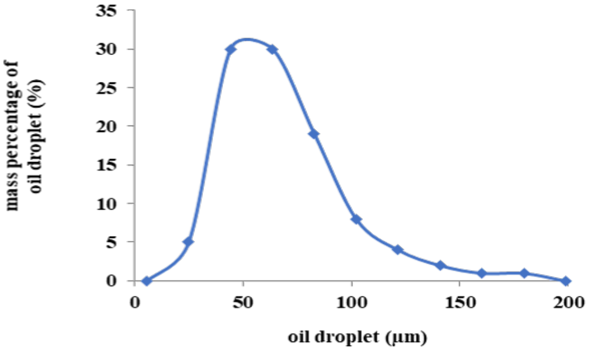

Figure 13. Oil size distribution

3.4. Effect of droplet size on the separator performance

Inlet droplet size of oil was changed ± 20% in this research to have small, medium and coarse droplets. Table 2 shows that injecting coarser droplets leads to a decrease (about 23%) in the droplet mass flow at the gas out let, while droplet mass flow increased by 29% when the inlet size was smaller. In fact, kharoua et al. (2013 a, 2013b) showed that settling out the small droplets due to gravity is hard and they can be moved easily by gas towards the outlet and leads to an increase in the liquid mass in the gas outlet, so a decrease in the separator performance can be observed which this trend is in accordance with the results presented in this section using the coupled VOF-DPM model. So, it is concluded that, appropriate size distribution that can be produced using suitable internals at the entrance to improve the separator efficiency (Kharoua et al, 2013a, 2013 b) is an effective parameter in separator efficiency that has paid less attention in separator modeling.

Table 2: Effect of droplet size on droplet mass flow in the gas outlet

Droplet size change	- 20%	Real size	+ 20%
Mass flow of droplet(kg/hr)	74	57.07	44.1

3.5. Effect of droplet size on the separator performance

The inlet flow rate of the gas phase was changed in the range of 5298-6475 kg/hr based on the field experience in the studied separator. Figure. 14 reveals that the liquid entrainment in the gas outlet increases by increasing the inlet flow rate. Noted that the increase in droplet mass at the gas outlet is more significant at higher flow rate range, i.e.,5800-6475 kg/hr, such that the liquid mass flow increases about 38% in the gas outlet, while no significant increase in the liquid mass can be illustrated by increasing the flow rate in the lower range, i.e., 5298-5800kg/hr. In fact, by increasing the flow rate, the retention time of the droplets decreases because of the higher velocity of the gas phase due to higher flow rate, so they have no sufficient time to be separated from the gas phase at the higher velocity and it leads to a reduction in separator efficiency (Mohammadi Ghaleni et al, 2012) which this trend is in accordance with the results found in this section.

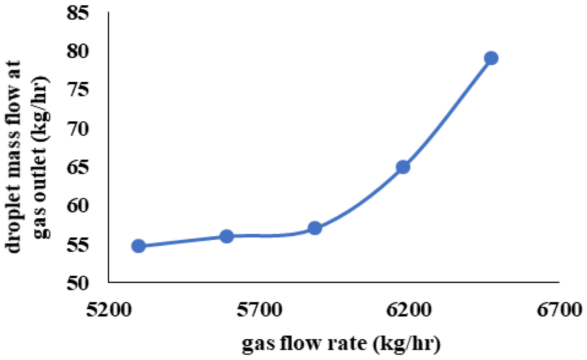

Figure 14. Oil size distribution

4. Conclusions

In this study a coupled VOF-DPM model was used to simulate an industrial three-phase boot separator. Considering virtual mass force, DRW and break up models led to a decrease in the simulation error relative to industrial data by16.6%, 7.58% and 3.71% respectively. So, the results underlined the importance of DRW in estimating separator efficiency. A good agreement of 12.9% error between simulation and industrial data using all the sub-models, reveals that this model is capable of estimating the separator efficiency and also the fluid flow profile. Droplet size distribution of liquids at the gas outlet showed that the average diameter of both oil and water were between 10 and 100 µm that requires an appropriate mist eliminator to increase the separator efficiency. Using the validated model, the effects of droplet size and flow rate on the separator performance were investigated. Results highlighted the effect of droplet size on the separation performance so that increasing the droplet size caused a better separator performance, since the liquid entrainment in the gas outlet decreased by 23%. The results pertinent to the effect of flow rate presented that the liquid entrainment in the gas outlet was highly influenced by the retention time at higher flow rate (5800-6475 kg/hr), such that increasing the flow rate increased the liquid mass by 38%, while the intended separator had the ability of changing the inlet flow rate in the range of 5298-5800 kg/hr without a significant increase in the liquid mass at the gas outlet.

Nomenclature	
C_D	Drag coefficient [-]
D	Pipe diameter of the flow [m]
dp	Particle diameter [m]
d_{max}	Maximum diameter [m]
$\bar{d}, d_{mean}$	Mean of diameter [m]
F_D	Drag force [N]
$\vec{F}$	Source term force [N/m^2]
$F_{\vartheta m}$	Virtual mass force [N]
$\vec{f}$	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]
r	Particle radius [m]
$\vec{u}$	Velocity of fluid [m/s]
$\vec{u}_m$	Velocity of phase m [m/s]
u_p	Particle velocity [m/s]
u_c	Velocity of continuous phase [m/s]
u’	Velocity fluctuation [m/s]

Greek letters	
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]
ρ_m	Density of phase m [kg/m^3]
τ	shear stress [N/m^2]

Acknowledgment

The authors wish to thank the Borzoyeh Petrochemical Company for its valuable contribution in providing all the necessary data in this research.

7. 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. Bothamley, M., 2013. Gas/liquid separators: quantifying separation performance-part 1. Oil and Gas. Fac., 2 (4), 21-29.
4. Bracill, J.u., Kothe, D.B., Zemach, c., 1992. A continuum method for modeling surface tension. J.Comput.Phys., 100, 335-356.
5. 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.
6. 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.

7. Gawas, K., 2013. Studies in low-liquid loading in gas/oil/water three phase flow in horizontal and near-horizontal pipes. Ph.D. dissertation, The University of Tulsa, Tulsa.
8. Ghafarkhah, A., Shahrabi, M.A., Moraveji, M.K., Eslami, H., 2017. Application of CFD for designing conventional three phase oilfeld separator. Egypt. J. Pet., 26 (2), 413–420.
9. 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.
10. Hsu, R.C., Chiu, C.K., Lin, S.C.,2017. A CFD study of the drawdown speed of floating solids in a stirred vessel. J Taiwan Inst Chem Eng, in press, 1-11.
11. Huang, A.N., Ito, K., Fukasawa, T., Fukui, K., Kuo, H.P., 2018. Effects of particle mass loading on the hydrodynamics and separation efficiency of a cyclone separator. J Taiwan Inst Chem Eng, in press, 1-7.
12. Johansen, Q., Brandvik, P.J., Faroot, u.,2013. droplet break up in sub sea oil release -part2: prediction of droplet size distribution with and without injection of chemical dispersants. Ma. Poll, Bull., 173, 327-335.
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. Kharoua, N., Khezzar, L., Saadawi, H., 2013b. CFD simulation of three-phase separator: effects of size distribution. ASME FEDSM. Nevada, USA.
15. Kirveski, L., 2016. Design of Horizontal three-phase separator using computational fluid dynamics. MSC Dissertation, Alato university school of chemical technology.
16. Kongre, U.v., Sunnap war, V.K., 2010. CFD modeling and experimental validation of combustion in direct ignition fueled with diesel. International Journal Applied Engineering Research, 1 (3), 508-517.
17. MC cleney, A.B., Owston, R.A., Green, S.T., Viana, F., and Nelson, S.M., 2017. modeling of a full -scale Horizontal liquid-liquid separator under condition of varying flow rate, water cut and viscosity with experimental validation. off shore technology conference, Texas.
18. 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.
19. Monnery, W.D., Svrcek, W.Y., 1994. Successfully specify 3-phase separators. Chem. Eng. Prog, 90 (6), 29–40.
20. Mostafaiyan, M., Saeb, M.R., Alorizi, A.E., Farahani M., 2014. Application of evolutionary computational approach in design of horizontal three-phase gravity separator. Journal of petroleum science and engineering,19, 28-35.
21. Pourahmadi Laleh, A., 2010. CFD Simulation of Multiphase Separators. Ph.D. Dissertation, University of Calgary, Canada.
22. 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.
23. 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.
24. 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 Engineerinh Congress and Exhibition, Canada.

25. Saffari, H., Dalir, N., 2012. Effect of virtual mass force on prediction of pressure changes in condensing tubes. thermal science, 16 (2), 613-622.

26. 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.

27. Zhang, B., Kong, L., Jin, H., He, G., Yang, S., Guo, x., 2018. CFD simulation of gas-liquid flow in a high-pressure bubble column with a modified population balance model. CHINESE J CHEM ENG, 26, 1350-1358.

شبیه سازی جداکننده سه فازصنعتی دارای بوت با استفاده از دینامیک سیالات محاسباتی

- زهره خلیفات[۱]، مرتضی زیودار[۲*]، رهبر رحیمی[۳]

۱. دانشجوی دکتری مهندسی شیمی، گروه مهندسی شیمی، دانشگاه سیستان و بلوچستان، زاهدان، ایران
۲. استاد، گروه مهندسی شیمی، دانشکده مهندسی شهید نیکبخت، دانشگاه سیستان و بلوچستان، زاهدان، ایران
۳. پروفسور، گروه مهندسی شیمی، دانشکده مهندسی شهید نیکبخت، دانشگاه سیستان و بلوچستان، زاهدان، ایران
(ایمیل نویسنده مسئول: mzivdar@eng.usb.ac.ir)

چکیده

جداکننده های سه فازی برای جدایش فازهای غیر قابل امتزاج در صنایع نفتی مورد استفاده قرار می گیرند. شبیه سازی این جداکننده های صنعتی با استفاده از دینامیک سیالات محاسباتی در مراجع بسیار محدود بوده و اکثر کارها ی انجام شده مربوط به شبیه سازی با استفاده از دیدگاه اولر-اولر یا اولر-لاگرانژ با تطابق ضعیف بین نتایج مربوط به شبیه سازی و داده های صنعتی بوده است . در این کار یک مدل که ترکیبی از دو مدل حجم سیال(FOV)و مدل فاز ناپیوسته (MPD) که ترکیب دو دیدگاه اولر-اولر و اولر-لاگرانژ بوده، برای شبیه سازی یک جداکننده سه فاز صنعتی دارای بوت توسعه داده شده است. لازم به ذکر است که با وجود کاربرد گسترده جداکننده های دارای بوت در صنایع نفتی، تا کنون هیچ پژوهشی روی این نوع از جداساز ها انجام نشده است .به منظور توسعه این مدل ترکیبی در این کار، اثر زیر مدل های مختلف شامل نیروی جرم مجازی، شکست قطرات و مدل گام تصادفی (WRD) که در اکثر شبیه سازی ها از آن صرف نظر شده، مورد بررسی قرار گرفته است. میزان جرم قطرات در گاز خروجی از داده های مربوط به جداکننده موجود در شرکت پتروشیمی برزویه واقع در جنوب ایران به عنوان معیاری برای ارزیابی مدل مورد نظر، مورد استفاده قرار گرفته است. نتایج نشان داد که مدل ترکیبی مورد نظر با در نظر گرفتن هر سه زیر مدل و تاثیر بالای زیرمدل WRD یک مدل موفق در تخمین بازده جداکننده بوده است، به طوری که با استفاده از زیر مدل ها خطای نتایج شبیه سازی از۴۰/۸۱ ٪ به ۱۲/۹٪ کاهش پیدا کرده است. در این کار همچنین با استفاده از مدل اعتبار سنجی شده به بررسی اثر اندازه قطرات و دبی ورودی روی عملکرد جدا کننده پرداخته شده است. نتایج شان دادند که کاهش اندازه قطرات(به اندازه ۰۲٪) و افزایش دبی(۶۴۷۵kg/hr -۵۸۰۰) باعث کاهش راندمان جداساز شده است به طوری که میزان جرم قطرات مایع در گاز خروجی به ترتیب به اندازه ۲۹٪ و ۳۸٪ افزایش یافته است.

واژگان کلیدی: دینامیک سیالات محاسباتی، جریان چند فازی، شبیه سازی، جداکننده سه فاز دارای بوت، مدل گام تصادفی.