The Use of Finite Element Analysis in the Design of Oil-Water Separators

Mohamed E. Wanas[1*], Yehia M. S. Elshazly[1], Dina A. ElGayar[1]
[1]Chemical Engineering Department, Faculty of Engineering, University of Alexandria; Alexandria; Egypt
[*] mwanas87@gmail.com, mohamed_ehab@ebgdco.com

Abstract: Oil-Water separators play an important role in several industries as well as in waste water treatment. However, no basic principles have been set to guarantee the designed separators would work according to the desired efficiency due to the effect various factors. Both time and money could be saved by simulating the separation process using Comsol software without the need of building a prototype and testing it.
This study aims to use Comsol software to simulate the separation process and test its accuracy by comparing the modeling and experimental results.
Upon confirmation of modeling accuracy, the study also aims to test the effect of different operational and design parameters on the efficiency of the separation process. Different conclusions drawn from this study could be very helpful in the design and operation of the separators.
Keywords: Computational Fluid Dynamics, Two phase flow, Comsol, oil- water Separators.

1. Introduction

Separation processes play an important role in several industries such as in oil and gas refineries. Gravity settling separators are used widely in oil and gas industry to separate water from oil produced from wells. These separators depend mainly on the difference in specific gravity between the two fluids[1, 2]. As the oil wells get older the oil-water separators' importance becomes even greater in order to overcome the oil fields aging phenomenon by which the water content in the oil produced from wells increases [3]. This increase causes corrosion problems to the piping and equipment used in the production and reduces the heating value of the produced oil.[4]

Another important use of gravity settling separators is in the treatment of wastewater produced from different industries or from ships bilge tanks which contain mixtures of different fluids along with water that could have harmful effects on the environment; hence, it has to be separated prior to the disposal to comply with environmental laws and regulations.[5-7]

Thus, the design of separators has achieved great attention as many factors interfere in the separation process. The design of such separators have only depended on previous experiences or study prototypes to achieve the desired separation efficiency.[8-10]

In the past few years, Computational fluid dynamics (CFD) and finite element analysis have been used as a replacement to the previous methods. CFD has many advantages such as cost, time and effort reduction over traditional methods. A major advantage is also that not all separator prototypes are easy to build or perform experimentally; however, these experiments could be easily simulated or applied using CFD modeling.[11-17]

In this study, Comsol was used to build horizontal oil-water separators and study the effect of different variables and designs on the separation process efficiency. Similar separator prototypes were built and tested at similar settings to those in the Comsol models in order to test their accuracy compared to the experimental results.

Moreover, a tracer (Methylene blue dye) was injected into the prototype using a syringe from the inlet nozzles and the color was monitored to study the flow pattern.

2. Laboratory Experiments

The laboratory experiments used a tank equipped with a rotating stirrer to mix transformer oil (who's viscosity was determined using a viscometer and found to be 0.0603Pa.s and a Pycnometer to measure the Specific gravity which was found to be 0.870) and tap water. A 0.5hp centrifugal pump was then used to pump the fluid into the horizontal separator. Valves controlled the flow which was measured using a rotating vane flow meter. (Fig.1)

The separator is a plexi glass cylindrical horizontal vessel of 45cm length and a diameter of 15cm [18]. It is equipped with half elliptical ends on both sides. The separator had one inlet nozzle at the center of one of its ends and two outlet nozzles: one at the top and one at the bottom, near the other elliptical end. The inlet and outlet nozzles had an inner diameter of 1.5cm. Samples were collected at the inlet and the two outlets for measurement (Fig.2). The samples were left to settle in graduated measuring cylinders to measure their oil in water volumetric fractions.

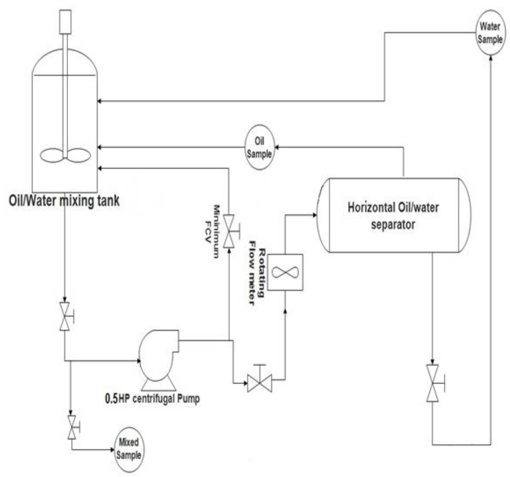


Figure 1. Process Flow Diagram for the prepared laboratory experiment

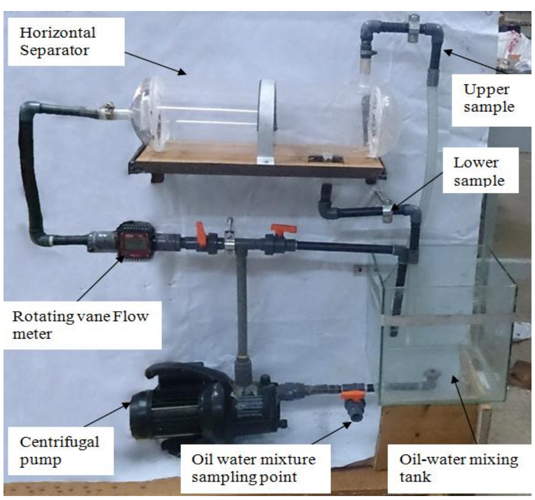


Figure 2. Laboratory setup for the experiment before placing into operation

3. Modeling Experiments

Reynolds number calculation confirmed the flow was turbulent, moreover, the presence of several obstacles in the system which would increase the level of turbulence. Thus, the Turbulent Two-Phase Flow, Level Set interface was selected to track the separation between the two liquids.

3.1 Separator Geometry

The separator model was built in similar dimensions to those used in the laboratory prototype. Vanes were added to some models at different location to study their effect on the process.

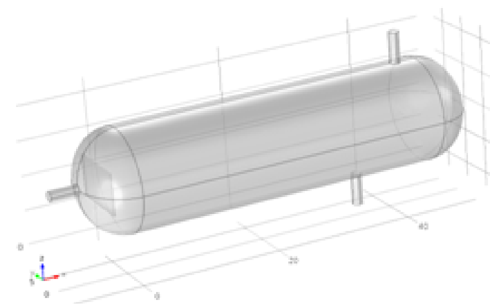


Figure 3. 3D separator model with inlet diverter.

Although 3D simulation would generate more precise results for cylindrical and spherical designs than 2D simulations. Moreover, the 2D model does not describe the actual cylindrical shape accurately. However, the results were close to those of the experiments. Therefore, there was no need for the excessive computer. Only 2D simulation was used in the study.

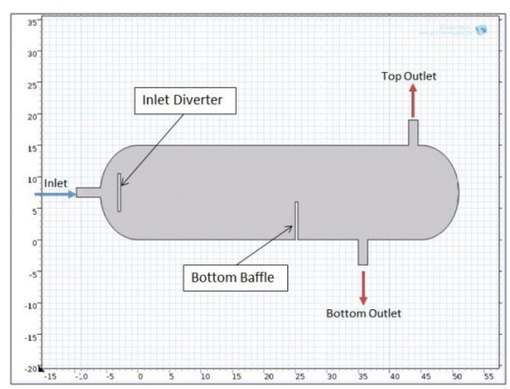


Figure 4. The simplified 2D separator model.

3.2 Governing Equations

The chosen Turbulent Two-Phase Flow, Level Set interface depends mainly on some basic concepts and equations: The conservation of mass and momentum. The software then solves Navier-Stokes equations in order to the track the interface between the two non mixing fluids.[19]

$$\rho\left(\frac{\partial u}{\partial t}+u.\nabla u\right)=-\nabla P+\nabla.\left(\mu(\nabla u+(\nabla u)^T)-\frac{2}{3}\mu(\nabla.u)I\right)+F$$

where u is the fluid velocity, p is the fluid pressure, ρ is the fluid density, μ is the fluid dynamic viscosity, I is the identity matrix and F is the external forces acting on the fluid.

The equation is built on the equalization between the different forces acting on the fluid: the inertial forces on one side and the sum of the pressure forces, viscous forces, and the external forces applied to the fluid on the other side. The turbulence effects are modeled using the standard two-equation k-ε model with realizability constraints.[20, 21]

Time dependent simulation was used to track the interface until the steady state conditions of operation is reached.

4. Variables studied

The relation between variables and the separation efficiency was monitored by comparing the bottom outlet sample compositions. Different variables studied were:

a. Inlet Composition

Fluids of different inlet mixture compositions 10%, 20% and 30% by vol. and the out coming fluids from the bottom outlet nozzles were analyzed to study the effect of the inlet composition on the separation process.

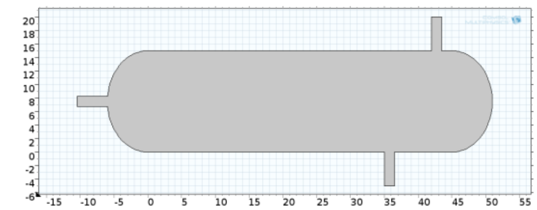
b. Inlet Velocity

Mixtures of a constant composition of 30% oil in water by Vol. were used at various inlet velocities in a separator with no internal baffles to study the effect of the inlet velocity on the separation process. The inlet velocities ranged between 0.2 to 1 m/s in laboratory experiments and between 0.5m/s and 3m/s in Comsol simulations. The velocity in laboratory experiments only reached 1m/s as this was the maximum velocity that could be reached by the used pump's power.

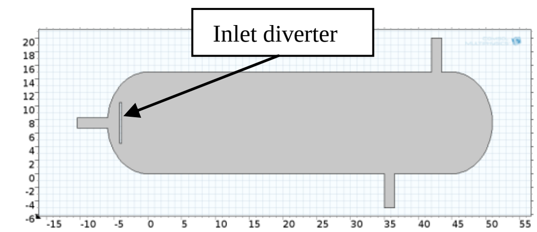
c. Separator internal design

i. Presence of baffles

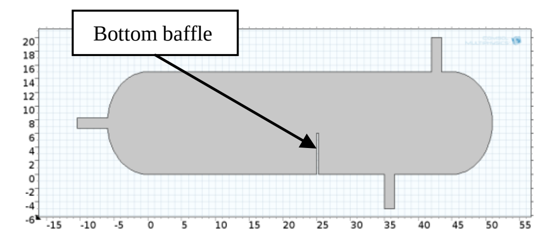
The effect of the presence of baffles (as inlet diverters or bottom baffles) on the separation efficiency has been studied. Experimental results were compared with modeling results to ensure the relationship is observed correctly.



a) Empty separator with no internal baffles



b) Separator with an inlet diverter



c) Separator with bottom baffle placed 10cm ahead of bottom outlet nozzle.

Figure 5. The three different separator designs.

ii. Baffle location inside the separator

The location of the bottom baffle ahead of the bottom outlet nozzle was then studied to test its effect on the composition of the bottom outlet.

5.Identification of the flow pattern

Another step taken in order to confirm the accuracy of Comsol multiphysics simulations has been done by injecting a tracer (Methylene blue dye) through the inlet nozzle of different separator designs and observing how the colored fluid flows then inside the separator. Comsol was then used to monitor the fluid flow pattern in similar separator models.

6.Mesh design

Different mesh sizes were tested. The results showed that the size of the mesh is irrelevant. Thus, the Coarse mesh size was used for the rest of the simulations.[22]

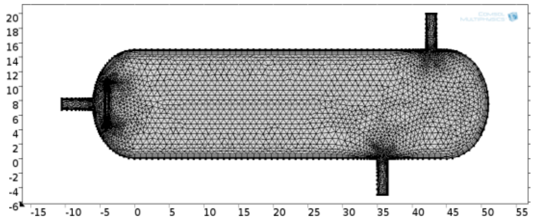


Figure 6. 2D discretization of the separator to a coarse mesh size.

7. Results

7.1 Effect of different variables

The first step taken to study the effect of different variables was developing an equation that would help monitoring the separation process efficiency and especially for different inlet composition where it is not logical to just compare the compositions of the bottom outlet samples. The percentage reduction in oil in the bottom outlet was considered as an indication for the separation efficiency. It was calculated as follows:

$$\%Reduction\ in\ oil = \frac{Inlet\ oil\ vol\% - Bottom\ outlet\ oil\ vol\ \%}{Inlet\ oil\ vol\%}$$

7.1.1 Effect of Inlet velocity

The results from laboratory experiments were close to those from the Comsol simulations. Fig.7 also shows clearly that as the inlet velocity increases the % reduction in oil decreases which also indicates a drop in the separation process efficiency.

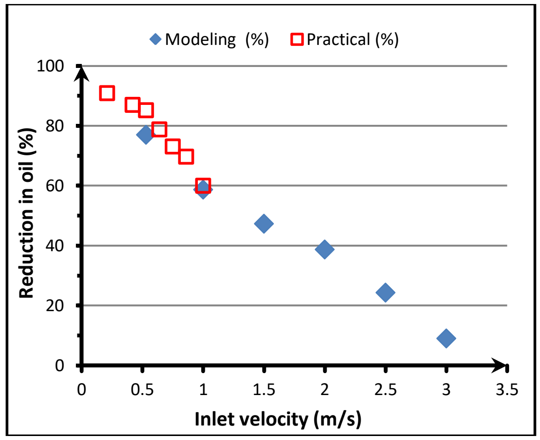


Figure 7. Percent reduction in oil results for experiments using an empty separator.

7.1.2 Effect of separator internal design

Fig. 8 indicates that experimental tests separators gave close results as those obtained from their corresponding models. Moreover, the figure also shows that both the inlet diverter and bottom baffle had a positive influence on the separation efficiency over the other separator with no baffles inside.

Although, the effect of both baffles seemed very much alike from Fig.8 which could be also confirmed from Comsol simulations in Fig.9. However, the location had another influence, the bottom baffle resulted in greater amounts of water to escape from the upper outlet along with the oil than when using the inlet diverter.

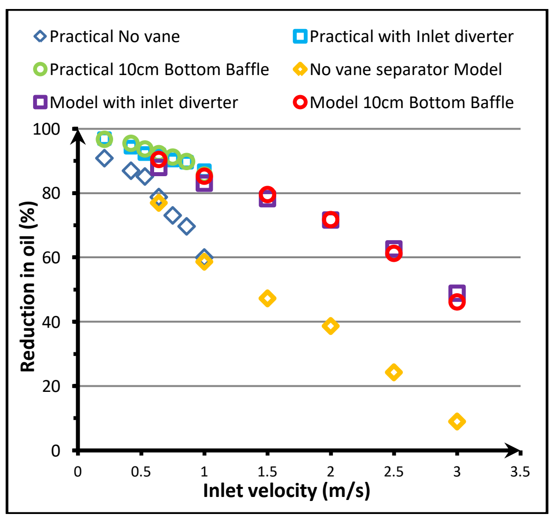


Figure 8. Percent reduction in oil results for separators of different internal designs.

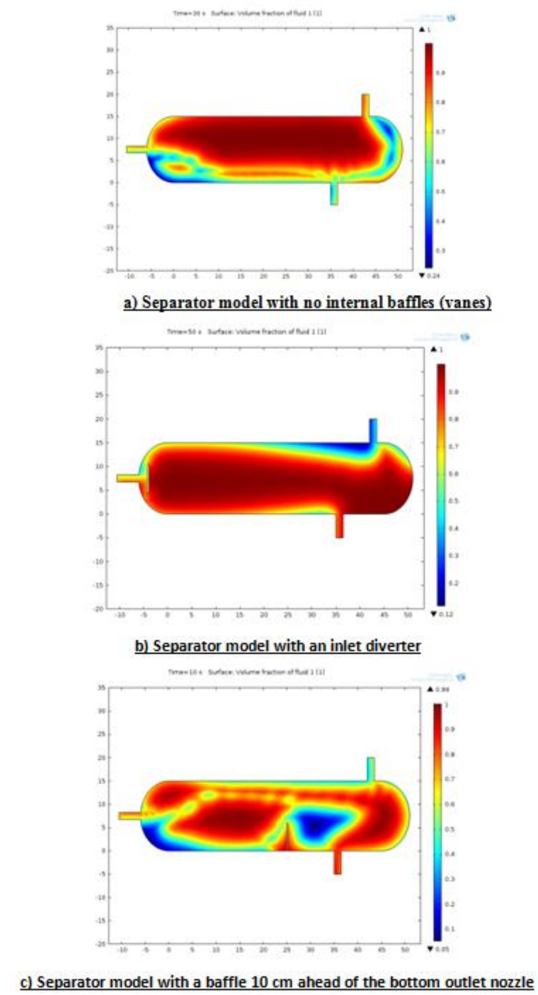


Figure 9. Simulations of different separator designs.

7.1.3 Effect of bottom baffle location inside the separator

Two bottom baffle locations were tested with reference to the bottom outlet nozzle. Locations of the bottom baffles were 10cm and 20cm ahead of the bottom outlet. Their results were compared to those from the no baffle separator.

At low velocities the results for both of the baffle locations were close, however, as the velocity increases the one placed closer to the bottom outlet (10cm distance) showed much better separation efficiency than the other one. The effect of the bottom baffle that was distant from the bottom baffle nearly vanished at high inlet velocities.

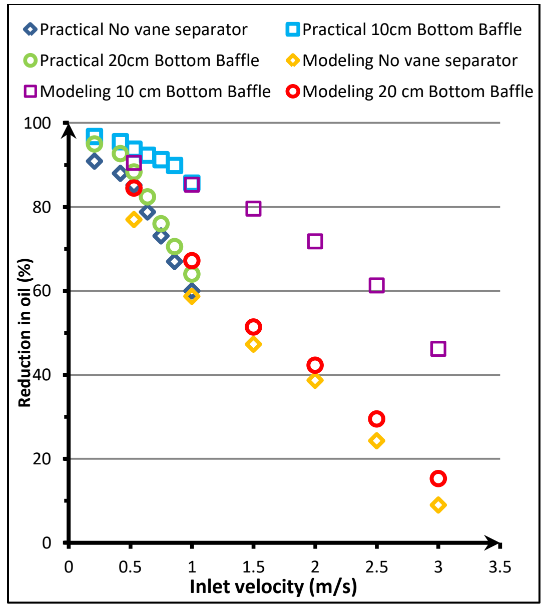


Figure 10. Percent reduction in oil results for separators with different bottom baffle locations.

7.1.4 Effect of mixture inlet composition

Fig.11 showed that the higher mixture compositions resulted in greater oil composition reductions than the lower mixture compositions.

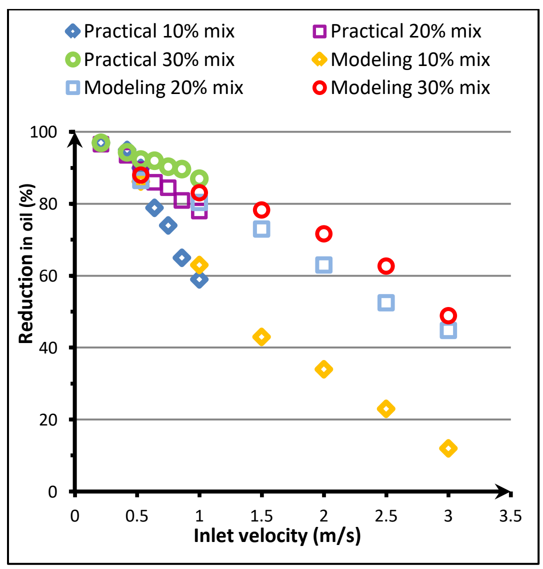


Figure 11. Percent reduction in oil results for different inlet mixture compositions.

7.2 Identification of the Flow Pattern

7.2.1 Separator with no internal vanes

Fig. 12 and Fig.13 indicate that the fluid enters with high momentum therefore, it does not spread for a certain distance inside the separator. After the momentum reduces gradually, the blue dye and the color in the model (Fig.13) starts spreading till filling the separator. Finally, the fluid reaches the outlet nozzles where the color by then the color reaches all the ends and fills the entire separator volume.

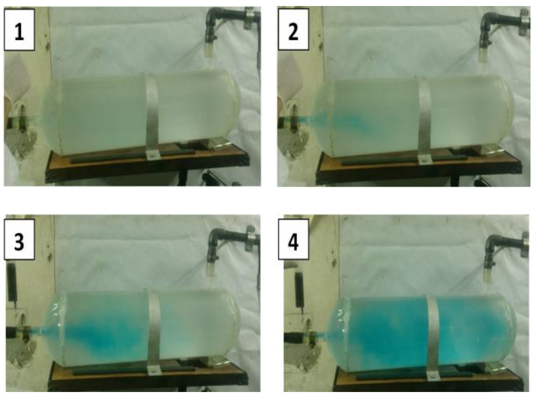


Figure 12. Laboratory experiments using a tracer for identification of flow pattern in an empty no vane separator.

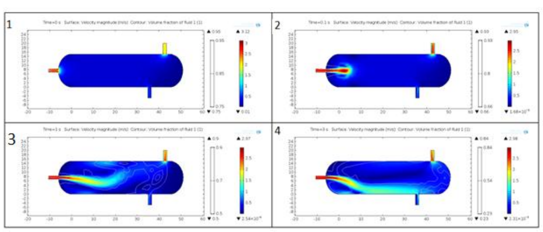


Figure 13. Identification of the flow pattern by Comsol simulations for an empty no baffle separator.

7.2.2 Separator with an inlet diverter

Fig. 14 and Fig.15 indicate that the fluid enters with high momentum but starts spreading rapidly to the sides due to the collision with the inlet diverter. The interference of the diverter, decreases the velocity and forms eddies at the center. This causes the color to much longer time to reach the outlet nozzles and fill the separator's entire volume.

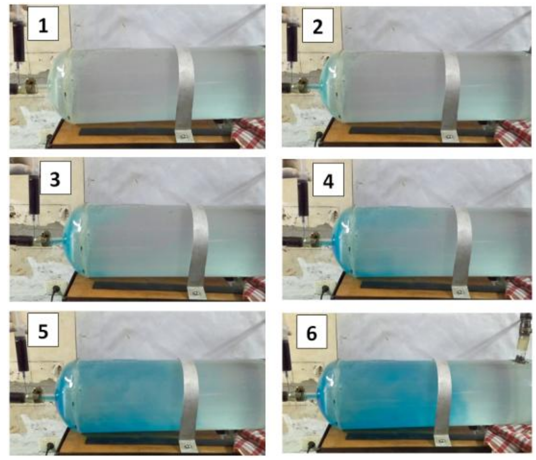


Figure 14. Laboratory experiments using a tracer to identify the flow pattern for a separator with an inlet diverter.

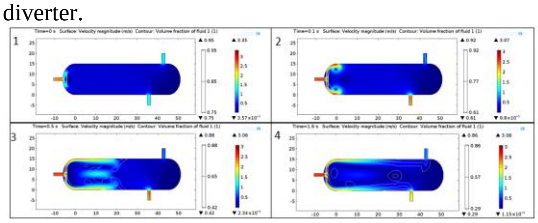


Figure 15. Identification of flow pattern by Comsol simulations for a separator with an inlet diverter.

8. Conclusions

After comparing the Comsol modeling results with the experimental results for different separator designs. A great resemblance was seen between the modeling and the experimental results. Referring to these results we could conclude that:

a) Comsol CFD simulations could be used with great confidence to replace the construction of a prototype.

b) The Introduction of a vane (as an inlet diverter or bottom baffle) plays an important role in enhancing the separation efficiency. The introduction of a vane could also offer a cheaper solution that would increase the separation efficiency than other possible solutions such as extending the separator size.

c) The location of the bottom baffle inside the separator should be studied with reference to the inlet mixture flow rate to achieve the desired separation efficiency.

d) Mixtures with low oil content need longer residence times to reach similar percentages reduction in compositions as those with higher oil contents.

9. References

1. Gas Processors Suppliers Association Engineering Data Book, Twelfth Edition ed., (2004).
2. J.L. Humphrey, G.E. Keller, Separation Process Technology, McGraw-Hill, (1997).
3. H.K. Abdel-Aal, M. Aggour, M.A. Fahim, Petroleum and Gas Field Processing, Taylor & Francis, (2003).
4. H. Devold, Oil and Gas Production Handbook: An Introduction to Oil and Gas Production, (2013).
5. J.H. Vandermeulen, S.E. Hrudey, Oil in Freshwater: Chemistry, Biology, Countermeasure Technology: Proceedings of the Symposium of Oil Pollution in Freshwater, Edmonton, Alberta, Canada, Elsevier Science, (2013).
6. M. Stewart, K. Arnold, Produced Water Treatment Field Manual, Elsevier Science, (2011).
7. C.A.S. Hall, K.A. Klitgaard, Energy and the Wealth of Nations: Understanding the Biophysical Economy, Springer, (2011).
8. E.G. Sinaiski, E.J. Lapiga, Separation of Multiphase, Multicomponent Systems, Wiley, (2007).
9. M. Stewart, K. Arnold, Gas-Liquid And Liquid-Liquid Separators, Elsevier Science, (2008).
10. C.E. Brennen, Fundamentals of Multiphase Flow, Cambridge University Press, (2005).
11. J. Tu, G.H. Yeoh, C. Liu, Computational Fluid Dynamics: A Practical Approach, Elsevier Science, (2012).
12. J. Wendt, J.D. Anderson, G. Degrez, E. Dick, R. Grundmann, Computational Fluid Dynamics: An Introduction, Springer Berlin Heidelberg, (2013).
13. J. Blazek, Computational Fluid Dynamics: Principles and Applications, Elsevier Science, (2015).
14. G. Venkatesan, N. Kulasekharan, S. Iniyan, Numerical analysis of curved vane demisters in estimating water droplet separation efficiency, Desalination, 339, (2014) 40-53.
15. G. Venkatesan, N. Kulasekharan, S. Iniyan, Influence of turbulence models on the performance prediction of flow through curved vane demisters, Desalination, 329, (2013) 19-28.
16. P.W. James, B.J. Azzopardi, Y. Wang, J.P. Hughes, The role of Drainage Channels In the Performance Of Wave-Plate mist eliminators, Chemical Engineering Research and Design, 81, (2003) 639-648.
17. C. Galletti, E. Brunazzi, L. Tognotti, A numerical model for gas flow and droplet motion in wave-plate mist eliminators with drainage channels, Chemical engineering science, 63, (2008) 5639-5652.
18. F.S. Manning, R.E. Thompson, R.E. Thompson, Oilfield Processing of Petroleum: Crude oil, PennWell Books, (1995).
19. H. Lomax, T.H. Pulliam, D.W. Zingg, Fundamentals of Computational Fluid Dynamics, Springer Berlin Heidelberg, (2013).
20. R.W. Pryor, Multiphysics Modeling Using COMSOL?: A First Principles Approach, Jones & Bartlett Learning, (2011).
21. Comsol, Comsol 4.3 CFD Module User's guide, DOI, (2012).
22. W.B.J. Zimmerman, Multiphysics Modelling with Finite Element Methods, London, (2006).