VOL. 11, NO. 22, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

VALIDATION OF SIMULATION PROCEDURE FOR VERTICAL OIL/WATER FLOW

Iylia E. A. Jamil and Hussain H. Al-Kayiem
Mechanical Engineering Department, Universiti Teknologi Petronas, Bandar Seri Iskandar, Perak, Malaysia
E-Mail: iyliaelena@gmail.com

ABSTRACT

This paper presents the validation done on the simulation procedure for a vertical oil/water flow. A 3-D steady state simulation was carried out to predict the behaviour of the oil/water flow in the wellbore with presence of a hydrocyclone separator. The procedure is then validated against a previous experimental work of oil/water flow in a vertical pipe. The volume fraction distribution profile for the simulation is compared against the profile obtained from the experimental work for three cases of mean volume fraction: 0.068, 0.135 and 0.205. The simulation results for oil volume fraction distribution agrees with the results with error of difference falling below 12%. This agreement shown in the results concludes that the procedure applied is acceptable.

Keywords: simulation, oil and water flow, multiphase.

INTRODUCTION

The increasing cost of oil production has driven the development for new technologies to improve the efficiency and output of existing oil production systems. One of these technologies is the Downhole Oil/Water Separation (DOWS), where a hydrocyclone is installed inside the wellbore to separate the mixture of oil and water. Work on DOWS has begun since the early 1990s, and has continued to this day due to its promising benefits [1].

Water production is undesirable and is a major concern for oil and gas producers. In any oil well, while water production reduces the oil output, its handling cost at the surface is an even bigger concern. This is the main reason why DOWS application has gained global interest. In an ideal application of the system, the downhole liquid-liquid hydrocyclone separator receives mixture of oil and water from the inside the wellbore, separates the mixture into oil-rich and water-rich streams and where oil is pushed to the surface for processing and water is injected back into the reservoir formation. Typically, at least one Electrical Submersible Pump (ESP) in installed as a part of the DOWS system configuration to provide the required pressure to drive this. Figure-1 shows the model of the wellbore with installed hydrocyclone.

In a recent work [2], DOWS system without the use of ESP has been suggested. Theoretically, the system requires relative positioning where a higher-pressure production zone is at the top and a lower-pressure dumping zone is at the bottom. Here, 3-D simulation of the hydrocyclone separator has been done to leverage on the pressure difference between the production zone (where oil and water mixture enters the wellbore) and the dumping zone (where water is injected back into the reservoir formation).

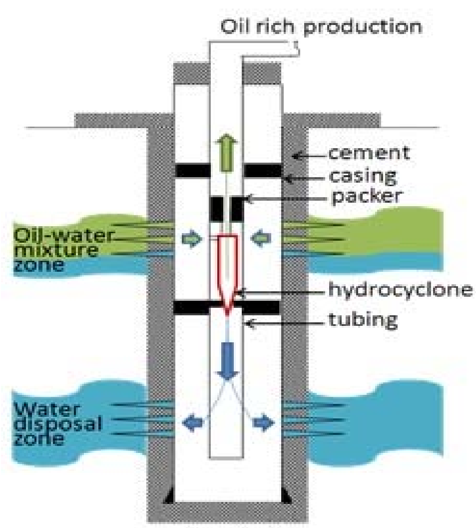

Figure-1. The wellbore model.

While work in [2] focuses on the hydrocyclone performance optimization using CFD, attention on the pre-separator and post-separator zones is deemed necessary. This has been followed by [3] where CFD simulation in 3-D has been done using ANSYS-Fluent 14 software as a fundamental investigation on the mixture behavior inside the production zone. The results from this work has been shared in a previous paper with more details.

A validation process for the simulation [3] is necessary to substantiate that the model behaves with satisfactory accuracy consistent with the study objectives while used within its applicable domain [4]. Model validity has also been described as determining that the theories and assumptions underlying the model are correct and the model is able to reasonably represent problem in terms of its structure, logic, and mathematical and causal relationships for its intended purpose. [5]

With regards to the importance of model validation, in this paper, the model described in [3] is

ARPN Journal of Engineering and Applied Sciences

validated. The same procedure is applied on an experimental model of oil/water flow in a vertical pipe and the results of volume fraction for the flow from both numerical simulation and experiment are compared to validate the simulation procedure. The full description of the experimental work may be referred [6].

THE EXPERIMENTAL MODEL

In the experimental work done [6], a dual sensor conductance probe was used to measure the local oil volume fraction distribution in vertical, oil in water bubbly flows in an 80 mm diameter vertical pipe. The pipe is 2.5 m tall with diameter of 80 mm. The two fluids used are water and oil. Oil density is taken to be 790 kg/m3 while oil viscosity is 2 mm2/s. The tests were carried out for values of water superficial velocity in the range 0.276 ms-1 to 0.417 ms-1 and for values of oil superficial velocity in the range 0.025 ms-1 to 0.083 ms-1. The measured oil volume fraction is 0.068, 0.135 and 0.205.

THE SIMULATION PROCEDURE

The simulation procedure is divided into geometry creation, mesh generation, boundary conditions set-up and post-processing.

Geometry creation. This stage is done using ANSYS Design Modeler. The geometry for the 3-D model is a simple vertical cylinder with 2.5 m height and inside diameter of 80 mm.

Mesh Generation. The next stage after geometry creation is meshing and this was done using ANSYS Meshing tool. For a simple geometry, a structured assembly may be applied. Typically comprising hexahedral (brick) elements, this type of mesh can provide higher quality solutions with fewer cells than compared to tetrahedral mesh. In Meshing, the inlet and outlet of the geometry is specified. Bottom of the cylindrical fluid volume is set as inlet and top of it is set as outlet.

Problem Set-up. The pressure-based solver was chosen for the simulation and set to solve in steady-state condition. In Fluent, there are two main approaches for the numerical calculation of the multiphase flow: The Euler-Lagrange approach and the Euler-Euler approach. The Euler-Lagrange approach is applied in the Lagrangian Discrete Phase model in Fluent. With the fundamental assumption made that the dispersed phase occupies a very low volume fraction this approach is not unsuitable for our model. Using the Euler-Euler multiphase model, there are three models to choose from: The Volume-of-Fluid (VOF) model, the Mixture model and the Eulerian model. The VOF model is unsuitable because it is for stratified or free-surface flows. To ensure a more stable solution, The Eulerian model was chosen for computation with the Mixture model being used during initialization for higher computation stability.

Boundary Conditions. For the model, the inlet type is set as velocity inlet and outlet type is set as pressure outlet. The properties of fluid used in the simulation are taken from the experiment [6]. The manipulative variable for the simulation is the inlet velocity of the flow and the mean oil volume fraction in the mixture. Summary of the model parameters is in Table-1.

Solution and Post Processing. For steady-state solutions, a small under-relaxation factor for the volume fraction is used to begin the calculation. Using the default Courant number, under relaxation factors were reduced by an average of 0.2 to avoid divergence in the solution. As mentioned, the option to start with a mixture multiphase calculation is applied before switching to the Eulerian multiphase model.

The solution method selected for the multiphase system was the Phase Coupled SIMPLE. Monitors and Solution Initialization were the same as for single phase. Second order discretization scheme was opted over the first order to achieve more accuracy.

Table-1. Summary of model parameters used in the simulation.

	Parameter	**Value**
Geometry	Diameter Height	80 mm ID 2.5 m
Fluid Properties	Water density Water viscosity Oil density Oil viscosity	1000 kg/m^3 1 mm^2/s 790 kg/m^3 2 mm^2/s
Flow Properties	Oil velocity Water velocity Outlet pressure Mean oil volume fraction	0.270 to 0.417 m/s 0.025 to 0.033 m/s 101.325 kPa 0.068, 0.135, 0.205

The residual plots were monitored while running the calculations to note when the values have reached the specified tolerance. Convergence was assumed when the residual difference was reaches 10-4 for all variables.

RESULTS AND DISCUSSIONS

In the experiment [6], the measurement probe was located at approximately 1.5 m from the inlet of the test section. In the simulation post-processing, a horizontal plane was created to cross-intersect the model at 1.5 m from the bottom. A line was drawn on the plane through the diameter of the model. The results for volume fraction distribution is plotted on this line and compared with the plot from the experiment [6]. For mean oil volume fraction 0.068, 0.135 and 0.205, the results are shown in Figure-2 to Figure-4.

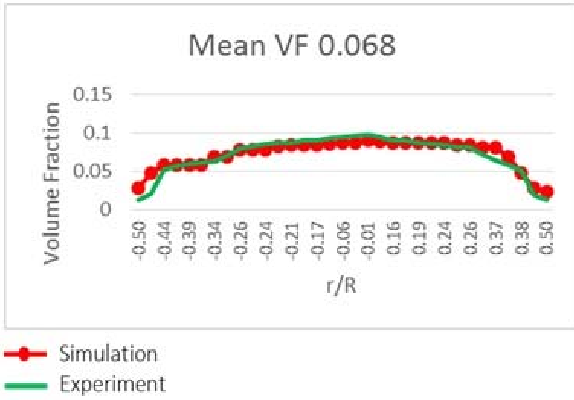

Figure-2. Oil volume fraction distribution for mean volume fraction 0.068.

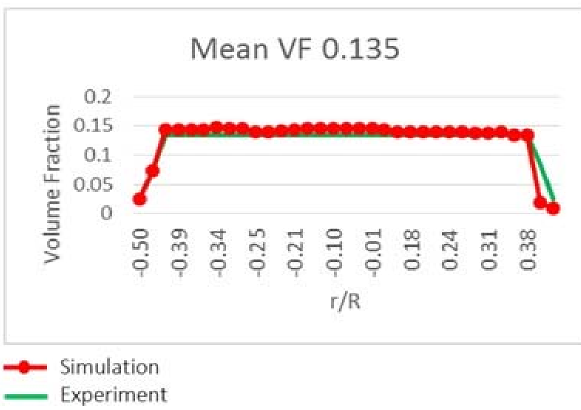

Figure-3. Oil volume fraction distribution for mean volume fraction 0.135.

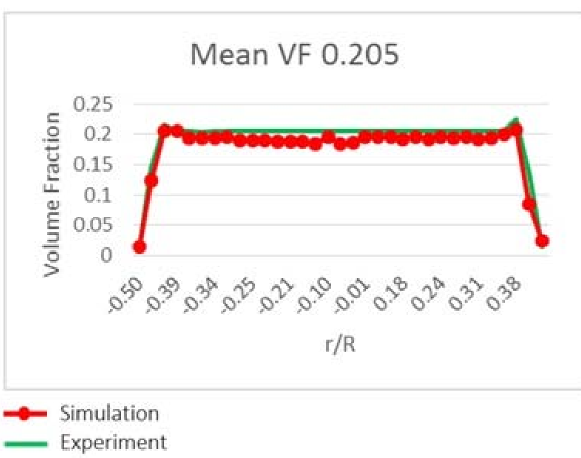

Figure-4. Oil volume fraction distribution for mean volume fraction 0.205.

The error calculated from the simulation against experiment is shown in Table-2.

Table-2. Error for different mean oil volume fractions.

Mean oil volume fraction in mixture	Averaged calculated error across pipe diameter
0.068	11.87%
0.135	2.980%
0.205	6.502%

From the results shown in Figure-2 to Figure-4, the volume fraction distribution obtained from the simulation is in agreement with the volume fraction distribution profile observed and measured from experiment. At different mean oil volume fractions, the shape of the plots vary significantly as has been described [6]. The averaged error across the pipe diameter has also been compared and the errors fall below 12%. Based on this, the simulation model is accepted to be used to predict the oil/water flow behavior in the downhole.

CONCLUSIONS

The CFD simulation procedure for a vertical oil/water flow has been validated against an experimental work. The simulation was done in 3-D steady-state Euler-Euler multiphase model and plots of oil volume fraction distribution across the diameter of the model has been compared with the same parameter from the experiment. The agreement is determined from the profile of the volume fraction distribution and the error calculation. From this, the simulation model is accepted for use in the

ACKNOWLEDGEMENTS

The authors acknowledge Universiti Teknologi PETRONAS for sponsoring the research under the YUTP-FRG 015-3AA-A34 research grants.

REFERENCES

[1] Peachey, B.R., and C.M. Matthews 1994, “Downhole Oil/Water Separator Development,” Journal of Canadian Petroleum Technology, 33(7): 17-21.

[2] Khor, Y.Y., Al-Kayiem, H.H. and Pao, W. 2013, “Numerical Study of Water Control with Downhole Oil-Water Separation Technology,” MATEC Web of Conferences, Volume 13, 2014. http://dx.doi.org/10.1051/matecconf/20141302029R.

[3] Jamil, I. E. and Al-Kayiem, H. H. 2015, “Simulation Analysis of the Oil/Water Structure in the Downhole with Presence of Hydrocyclone Separator,” IOP Conference Series: Materials Science and Engineering, Volume 100, Number 1, 2015, pp. 12025-12033(9) http://dx.doi.org/10.1088/1757-899X/100/1/012025.

[4] Haffke B., Möller R., Melz T., and Strackeljan, J. 2015, “Validation of Simulation Models without Knowledge of Parameters Using Differential

ARPN Journal of Engineering and Applied Sciences

Algebra," Mathematical Problems in Engineering, vol. 2015, Article ID 793216, 9 pages, 2015. doi:10.1155/2015/793216.

[5] Sargent, R. G. 2008, "An Introductory Tutorial on Verification and Validation of Simulation Models," Proceedings of the 2015 Winter Simulation Conference.

[6] Lucas, G.P. and Panagiotopoulos, N. 2008, "Measurement and Modelling of Oil Volume Fraction and Velocity Profiles in Vertical, Bubbly Oil-in-Water Flows," 11th Int. Conf. on Multiphase Flow in Industrial Plants, Palermo, Italy, 2008.