Simulation analysis of the oil/water structure in the downhole with presence of hydrocyclone separator

I E Jamil and H H Al-Kayiem
Mechanical Engineering Department, Universiti Teknologi Petronas, 32610 Bandar Seri Iskandar, Perak, Malaysia.

E-mail: hussain_kayiem@petronas.com.my

Abstract. The maturing oil fields with increasing water production can pose a challenge in terms of produced water handling and disposal issues. This paper presents the modelling and simulation procedure of the two-phase flow of water/oil in a downhole using ANSYS-FLUENT 14 software. The developed procedure successfully simulated the production zone and the interaction of the two fluids in a natural environment where the reservoir pressure is the sole driving force. The results show significant difference of volume distribution in the flows with different oil content. The model can become an essential tool to assist in prediction of the behavior of oil/water mixture flow in the wellbores, and to serve in designing downhole oil/water separators.

1. Introduction
The largest volume waste stream associated with oil and gas production is produced water [1, 2]. Virtually, water production in mature field is unavoidable. Oil is commonly accompanied by an underlying aquifer. As production rate is increasing, oil water contact (OWC) also increases until water breakthrough into the wellbore. The unavoidable premature water breakthrough or water coning could happen due to high pressure drawdown around the wellbore and high mobility of water than oil, where water viscosity much lesser than the oil, and water relative permeability much greater than oil. The consequences of increased water production translates to high operating expenses and produced water disposal costs, increased in environmental risk associated with large volumes of produced water, additional requirement for produced water lifting and significantly reduced surface processing facilities for oil. Water production can also limit the well production life via fines migrations and high water production inhibits oil production to production tubing, hence causing low recovery at oil layer [3-5].

Treatment and disposal of produced water represent significant costs for operators. A new technology, down-hole oil/water separators (DOWS), has been developed to reduce the cost of handling produced water [1, 2]. DOWS is a hydrocyclone-based system for downhole separation of produced oil and water and subsequent disposal of the produced water by reinjection within the same wellbore using electric submersible pump [6]. However, concurrent with the project presented in this paper, is the exploitation of downhole separation without the use of electric submersible pump. Theoretically such concept requires relative positioning of the zones of a high pressure production layers on top and a low pressure water zone below to accomplish hydrocyclone separation.

In order for better prediction of the behavior of the flow entering the hydrocyclone separator, we take a step back to study the behavior of the oil and water flow inside the wellbore. In this
investigation, computational fluid dynamics (CFD) is adopted to obtain quantitative description of the oil and water flow behavior from the reservoir into the wellbore. The wellbore is modelled in 3 dimensions using ANSYS-Fluent 14 software, while the fluid properties were taken from Bayan offshore field in Miri.

2. Model formulation of the downhole zone
The design of the wellbore is depicted from real well. As seen in figure 1, production in the well occurs in the upper zone and dumping in the lower zone [1]. The wellbore model is broken down into three parts: the upper zone (production zone) with high pressure, the lower zone (dumping zone) with low pressure, and the liquid-liquid hydrocyclone separation zone. The investigation presented in this paper focuses on the upper zone.

The wall of the zone is perforated with 12 shots per foot for the inflow of production fluid into the well. Table 1 and figure 2 show the design and parameters of the upper zone model of the wellbore.

Figure 1. The wellbore model.  
Figure 2. Model sketch of the upper zone.

Table 1. Parameters of upper zone design.

| Parameter                        | Value                      |
|----------------------------------|----------------------------|
| Casing OD                        | 9.5/8” (0.244m)            |
| Casing ID                        | 8.535” (0.217m)            |
| Casing wall thickness            | 0.545” (0.014m)            |
| Casing length (packer to packer) | 50’ (15.240m)              |
| Inner tubing OD                  | 3.5” (0.089m)              |
| Perforation                      | 12 shots per foot, 120 degrees phasing |

3. Numerical implementation
In ANSYS-FLUENT, 5 stages of simulation are need to be followed which are geometry, mesh, setup, solution and results. The order of the simulation is as such, so it is a top to down sequence. First, a 3
A dimensional (3-D) model of the production zone is created using ANSYS Design Modeler. With the dimensions stated in table 1, the 2-D outline of the model’s vertical cross-section is first sketched then revolved into a 3-D solid model. The perforated holes on the walls and the flow outlet are included later, drawn at a separate plane then extruded and cut out from the solid wellbore model.

| Details of Mesh                  | Description                  |
|----------------------------------|------------------------------|
| **Sizing**                       |                              |
| Use Advanced Size Function       | Off                          |
| Relevance Centre                 | Coarse                       |
| Element Size                     | Default                      |
| Initial Size Seed                | Active Assembly              |
| Smoothing                        | Low                          |
| Transition                       | Slow                         |
| Span Angle Centre                | Coarse                       |
| Minimum Edge Length              | 3.1919e-002m                 |
| **Inflation**                    |                              |
| Use Automatic Inflation          | Program Controlled           |
| Inflation Option                 | First Aspect Ratio           |
| First Aspect Ratio               | 5                            |
| Maximum Layers                   | 5                            |
| Growth Rate                      | 1.2                          |
| View Advanced Options            | No                           |
| **Assembly Meshing**             |                              |
| Method                           | None                         |

**Table 2.** First mesh setting.

Figure 3 shows the completed 3-D model for the wellbore. The next step leading to simulation is the meshing of the model as shown in Figure 4. In mesh, the inlets (perforated holes) and outlets (exit
into separator) of the model as well as the sizing, smoothing, inflation and all other mesh options are set. Table 2 shows the first mesh setting used to create mesh model. Other elements unspecified are using the default setting of the program. The statistics for this mesh model is 349, 690 nodes and 1, 832, 214 elements.

4. Mesh quality

Before going straight with the first mesh model to generate solution, there is great importance in carrying out the mesh independency study to check the quality of the mesh. We may gain a benefit in terms of computing time if the number of elements can be reduced but still maintaining the quality of the mesh. A high-quality mesh with lesser number of elements would require less computing time while giving equally good results. The changes made to the second mesh model are shown in table 3. The statistics for the second mesh is 253, 615 nodes and 1, 338, 868 elements.

Three primary factors to evaluate mesh quality analyzed here which are skewness, aspect ratio and orthogonal quality (figures 4-6). This second mesh model is used in the next stage of the simulation which is setting up the boundary conditions. The preliminary results of these are shared in the results section of this paper.

Table 3. Second mesh setting.

| Details of "Mesh"       | Description                                      |
|-------------------------|--------------------------------------------------|
| **Sizing**              |                                                  |
| Use Advanced Size Function | On: Curvature                                   |
| Relevance Center        | Coarse                                          |
| Initial Size Seed       | Active Assembly                                  |
| Smoothing               | Medium                                           |
| Transition              | Slow                                             |
| Span Angle Center       | Fine                                             |
| □ Curvature Normal Angle | Default (18.0 °)                                |
| □ Min Size              | Default (6.1304e-003 m)                          |
| □ Max Face Size         | Default (0.613040 m)                            |
| □ Max Size              | Default (1.22610 m)                             |
| □ Growth Rate           | Default (1.20)                                  |
| Minimum Edge Length     | 3.1919e-002 m                                  |
| **Inflation**           |                                                  |
| Use Automatic Inflation | None                                             |
| Inflation Option        | Smooth Transition                                |
| □ Transition Ratio      | 0.272                                           |
| □ Maximum Layers        | 5                                               |
| □ Growth Rate           | 1.2                                             |
| Inflation Algorithm     | Pre                                             |
| View Advanced Option    | No                                              |
5. **Skewness**
Skewness is one of the primary quality measures for the cells in a mesh. Skewness determines how close to ideal (i.e., equilateral or equiangular) a face or cell is. Skewness is valued from 0 to 1 and is computed directly based on the mesh built. 0 being the best and 1 being the worst quality of mesh elements. The skewness distribution of the present model is shown in figure 4. The average skewness of this initial model is 0.24, which falls under excellent category. Hence this model is acceptable.

![Figure 4. Mesh statistics – skewness.](image)

6. **Orthogonal quality**
Orthogonal quality helps determine the quality of the mesh itself. The worst cells will have an orthogonal quality closer to 0 and the best cells will have an orthogonal quality closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher. The average orthogonal quality calculated for the second mesh for the present model is 0.85, as in figure 5; hence it is very good.

![Figure 5. Mesh statistics – orthogonal quality.](image)

7. **Orthogonal quality**
The aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids, and the distances between the cell centroid and nodes. Generally, it is best to avoid sudden and large changes in cell aspect ratios in areas where the flow field exhibit large
changes or strong gradients. Analysis of the mesh in the present work in terms of aspect ratio is shown in figure 6.

![Figure 6. Mesh statistics – aspect ratio.](image)

8. **Boundary conditions**
The boundary conditions for the setup is taken from the Bayan field in Miri. Table 4 shows the input into the numerical solution.

| Parameter                  | Value                          |
|---------------------------|--------------------------------|
| **Flow Properties**       |                                |
| Inlet flowrate            | $3.036 \times 10^4$ m$^3$/s   |
| Inlet pressure            | 12,776 kPa                     |
| Outlet relative pressure  | 620.53 kPa                     |
| Turbulence intensity at inlet | 10%                           |
| Gravity                   | $(0, -9.81, 0)$ m/s$^2$        |
| **Fluid Input**           |                                |
| Density Water             | 1000 kg/m$^3$                  |
| Viscosity Water           | 0.00097 kg/m$\cdot$s           |
| Density Hydrocarbon       | 840 kg/m$^3$                   |
| Viscosity Hydrocarbon    | 0.00045 kg/m$\cdot$s           |
| Variation of Oil Content  | 10%, 30%, 50%, 70%, and 90%    |

9. **Results and discussion**
The solution is computed for 5 different cases, the variation being the oil content in the flow (10%, 30%, 50%, 70%, and 90%). Figure 8 shows the density distribution at the middle section of the simulation zone i.e. at the perforated holes and at the bottom of the simulation zone, near the inlets to the separator. We verify the simulation results by comparing the densities of the mixture in the flow obtained from the simulation against the mathematical formula. Equation (1) is used to calculate the mixture density $\rho$ where $c_{oil}$ is the volume fraction of oil in the mixture, $\rho_{oil}$ is the density of oil and $\rho_{water}$ is the density of water. The error from this calculation is shown in table 5.
\[ \rho = c_{oil}\rho_{oil} + (1 - c_{oil})\rho_{water} \]  

(1)

Table 5. Verification of density.

| Oil Content | Calculated Density | Average Density from Simulation | Error  |
|-------------|--------------------|---------------------------------|--------|
| 10%         | 984.00             | 985.88                          | 0.2%   |
| 30%         | 952.00             | 961.58                          | 1.0%   |
| 50%         | 920.00             | 936.69                          | 1.8%   |
| 70%         | 888.00             | 900.64                          | 1.4%   |
| 90%         | 856.00             | 876.40                          | 2.4%   |

Figure 7. The predicted density distribution of oil inside the wellbore at different oil/water percentage in the production zone.

Visualization of the oil/water mixture, as predicted by the simulation, is shown in figure 7 in terms of density distribution. Dark red color means 100% oil. Two locations in the upper zone of the well,
which is the production zone, identified as the middle and bottom. At 10%/90% oil/water production, the simulation in the upper location of the upper zone reveals that there is no spots with pure oil or pure water. But the mixture structure is changing at the bottom of the upper zone, where it is clear that lower density portions are produced. Same behavior could be noticed for the case of 90%/10% oil/water mixture. At the middle, the mixture is more homogenous compared to the bottom. This is due to the high turbulence and jet interaction from the perforating holes in the middle region.

![Density distribution at the middle and bottom sections.](image)

Figure 8 shows the predicted volume fraction of oil at cases of 10%/90% oil/water and 90%/10% oil/water mixtures. In the case of 10%/90% oil/water, significant difference of oil distribution between the middle section where flow enters the wellbore and the bottom section before the hydrocyclone separator inlets. The even color in the bottom section depicts that the flow is more settled. In the case of 90%/10% oil/water, it is obvious that the distribution of oil in entire wellbore length is even.

10. **Conclusions**
A wellbore with production in the upper zone is designed and modeled by Computational Fluid Dynamics using ANSYS-Fluent 14. The model can become an essential tool to better predict the behavior of oil/water mixture flow inside the wellbores, and to serve in designing downhole separators. The simulation results revealed that the mixture is more even in the upper portion of the production zone where the jet interaction from the perforation holes induce high turbulence which enhance the even distribution of the fluids droplets.

**Acknowledgements**
The authors acknowledge Universiti Teknologi Petronas for the financial and technical support to produce this paper under the research grant YUTP – FRG 015-3AA-A73.
References
[1] Bowers B, Brownlee R, Schrenkel P, Development of a Downhole Oilwater Separation and Reinjection System for Offshore Application. Offshore Technology Conference; 1998: Offshore Technology Conference.
[2] Matthews C, Chachula R, Peachey B, Solanki S, Application of downhole oil/water separation systems in the alliance field. SPE Health, Safety and Environment in Oil and Gas Exploration and Production Conference; 1996: Society of Petroleum Engineers.
[3] Aziz K 2001 SPE Reservoir Evaluation & Engineering 4 327-35
[4] Flores J G, Chen X T, Brill J P, Characterization of oil-water flow patterns in vertical and deviated wells. SPE Annual Technical Conference and Exhibition; 1997: Society of Petroleum Engineers.
[5] Ahmed M M, Ayoub M A 2013 Journal of Applied Sciences 14 3162-71
[6] Amini S, Mowla D, Golkar M, Esmaeilzadeh F 2012 Chemical Engineering Research and Design 90 2186-95