

MODELLING OF OIL-WATER SEPARATOR USING COMPUTATIONAL FLUID DYNAMICS (CFD)

Abdulkadir M.^{1*}, Hernandez-Perez V.¹ and Hossain M.²

*Author for correspondence

¹Process and Environmental Engineering Research Division, Faculty of Engineering, University of Nottingham, United Kingdom

²Faculty of Engineering, Robert Gordon University, Aberdeen, Scotland, United Kingdom
E-mail: enxma17@nottingham.ac.uk

ABSTRACT

The API design code is the existing design method for separators. It is based on rules of thumb; it does not look at complex phenomena that happen inside the separator. Building prototypes is both time consuming and expensive. Even if the design task is accomplished, the prototypes provide limited information as to why a particular design did or did not work. As a consequence of this, results may be obtained that are not exact and often lead to overdesign of the separator. Also, separation which is one component of a production phase poses a distinctive challenge on a floating platform because of the unavoidable wave motion to be expected at sea. These wave motions, i.e. pitch, heave, yaw, sway, surge and roll are present even in calm weather conditions. They tend to have a natural mixing effect on the oil, water and gas, thereby resulting in an increase in the time it takes to separate the mixture. The API design code has no answer to such a challenge. Computational fluid dynamics (CFD) can provide solutions to the aforementioned problems.

The characteristics of fluid flow and phase separation were numerically analysed as part of the work presented herein. The effects of parameters like velocity and droplet diameter on the separator geometry were solved using the software package Fluent 6.2, which is designed for numerical simulation of fluid flow and mass transfer.

Simulations were performed for different velocities and bubble diameters. The results showed that there is a strong dependency of phase separation on mixture velocity and droplet diameter. An increase in mixture velocity will bring about a slow down in phase separation and as a consequence will require a weir of greater height. An increase in droplet diameter will produce a better phase separation. The simulations are in agreement with results reported in literature and show that CFD can be a useful tool in studying a horizontal oil-water separator.

INTRODUCTION

Predicting and optimisation of the flow of oil, water and gas within the separator is crucial to economic and safe petroleum production. Experiments and Computational fluid dynamics simulations have been used to determine parameters such as

velocity, pressure, bubble diameter and so on. These parameters are important in predicting how they influence the separator geometry.

Sayda and Taylor [10] developed a dynamic mathematical model for an oil production facility. The hydrodynamics of liquid-liquid separation were modelled based on the API design criteria. The simulation model consisted of a two-phase separator followed by a three-phase model. The simulation results proved the sophistication of the model in spite of its simplicity. Furthermore, there study demonstrated the challenging task of modelling and controlling an oil and gas production facilities, and that more work has to be done to develop higher fidelity models.

According to [7], knowledge of the fluid flow dispersion characteristics in a separator is needed to improve the distributed fluid flow simulation. Also more research about the break up/ coalescence processes inside the multi-phase flow field in all zones in a separator are needed, in order to increase the separation efficiency in the bulk flow zones.

According to the work by [6], there is a strong dependency of drop diameters on entrainment. This is based on the fact that separation velocity is approximately proportional to the square of the particle diameter. The mixture model for the water drops could also be used on liquid drops in the gas phase. The entrainment of the liquid in gas would however be influenced most by the efficiency of the demister.

Powers [9] demonstrated that the effect of residence time can vary considerably, depending on vessel proportions and orientation and on the liquid level of horizontal vessels.

API SPEC 12J [2], presents the standard basis for sizing oil and gas separators. While that standard applies to vertical-separator gas capacities, it does not adequately describe horizontal-vessel performance.

The development of more efficient computers has generated the interest in CFD and, in turn, this has produced a dramatic improvement in the efficiency of the computational techniques. Consequently CFD is now the preferred means of testing alternative designs in many branches of the aircraft, flow

machinery, separators and, to a lesser extent, automobile industries [3].

The study presented herein will attempt to determine the effect different parameters like velocity and bubble diameter will have on the separator geometry using computational fluid dynamics

NUMERICAL METHOD

Computational Fluid Dynamics (CFD) is a numerical modelling technique that solves the Navier-Stokes equations on a discretised domain of the geometry of interest with the appropriate flow boundary conditions supplied. The Navier-Stokes equations are a complex non-linear set of partial differential equations that describe the mass and momentum conservation of a fluid. In this study, the Navier-Stokes equations for two phases are used via the Control volume method in FLUENT. An Eulerian model was used for the multiphase flow model. Additional physics can also be resolved by solving additional conservation equations, e.g., heat transfer, multi-phase flow and combustion.

A CFD analysis consists of the following steps [5]:

- Problem identification and Pre-Processing
 1. define your modelling goals
 2. identify the domain you will model
 3. design and create the grid
- Solver Extraction
 1. Set up the numerical model
 2. Compute and monitor the solution
- Post-Processing
 1. Examine the results
 2. Consider revisions to the model

Some of these steps will now be discussed in more detail. For details of these see [11] and [4].

Grid Generation

The grid generation process deals with the division of the domain under consideration into small control volumes on which the discretised governing equations will be solved.

The grid generation forms a large part in terms of person-hour time of the CFD analysis. A large amount of time has gone into and is currently going into the development of commercial automated grid generators. One such product is GAMBIT (Geometry and Mesh Building Intelligent Toolkit) [4]. GAMBIT uses solid modelling techniques to create a virtual model of the geometry under consideration. Various grid generation tools are available to create hexahedron, tetrahedron, prism and pyramid cells.

Geometry creation

The cylindrical separator used in this project is 25m long, with a radius of 1.25m. A weir of non porous medium is located at 17.5m from the inlet. Two outlets are positioned either side of

the weir at the bottom of the separator at 16.5m and 19m from the inlet to separate water and oil respectively. The third outlet is positioned at the top of the separator at 17.5m from the inlet. Due to symmetry only half the separator is modelled.

For complex geometries, quad/ hex meshes show no numerical advantage, and meshing effort can be saved by using a Tri/ Tetrahedral mesh [4]. Since the separator has a complex geometry, a Tri/ Tetrahedral mesh was used with an interval size of 0.12m. Below are the details of the steps followed in generating the separator geometry (model):

- 1) A 3D circular cylinder was created with a radius of 1.25m and height of 20m;
- 2) A sphere was then created on both sides of the cylinder with a radius of 1.25m;
- 3) There are now three volumes; volume of sphere at the extreme left hand side of the cylinder, volume of cylinder and volume of sphere at the extreme right hand side of the cylinder.

For the volume of sphere at the extreme left hand side of the cylinder, the volume of the sphere was split with the volume of the cylinder. A half portion of the sphere was deleted to form a hemisphere.

The same procedure was repeated for the volume of the sphere at the extreme right hand side of the cylinder. Details can be found in [1].

Conservation Equations

The governing equations that describe the flow field are a set of non-linear partial differential equations. The equations are derived from mass, momentum and energy conservation.

To effectively model the separator, the following assumptions are prescribed in the numerical computation:

- Incompressible flow;
- Steady state;
- Turbulent flow;
- No heat transfer;
- No heat radiation

Under these conditions, the governing equations for continuity, momentum and turbulent equations can be found in [1]:

Solving these sets of equations is very difficult; it is based on this reason that it will be done using a software package Fluent 6.2.

PROCESSING RESULT

Solution Algorithm

The control volume approach as implemented in the commercial CFD solver, FLUENT [4] is used in this study. In this method, the governing equations are first integrated on the individual control volumes that were created in the grid generation phase, to construct algebraic equations for the discrete dependent variables such as velocities, pressure, temperature, and conserved scalars. Secondly, the discretised equations are linearised and the resulting linear equation system is solved to yield updated values of the dependent variables.

In this study, the segregated solution method of FLUENT is used, as it is suitable for incompressible flows. In this approach, the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained.

TRENDS AND RESULTS

This section will aim to compare the contours of volume fraction of oil for different mixture velocities and for those with the same mixture velocity but different droplet diameters. It will also compare the velocity vectors for simulations with the same droplet diameter but with different mixture velocities.

Figures 1, 2 and 3 compare the predicted oil volume fraction of simulations with the same droplet diameter of 1mm with three different mixture velocities of 0.5, 0.75 and 1.0m/ s respectively. Figures 4 and 5 compare the predicted oil volume fraction for simulations with the same mixture velocity of 0.5m/ s with droplet diameters of 0.5 and 0.25mm respectively. Figures 6, 7 and 8 compares the velocity vector patterns for three mixture velocities of 0.5, 0.75 and 1.0m/ s with the same oil volume fraction of 0.5 and same droplet diameter of 1mm. The weir height for all the cases is 1.5m.

It can be observed that the flow of fluid from the inlet is not uniform in Figures 1 to 8 inclusive. A perforated plate will be required close to the inlet to assist in the development of uniform flow across the entire liquid section.

Over the range of conditions, a range of flow patterns were observed. For efficient separation, it was observed that the density difference between the phases must be high so that they can separate out under the influence of gravity. The present study shows the possible use of CFD for separator design. Parameters such as volume fraction, droplet size, inlet and outlet location and size and weir height can be easily changed in CFD design study. Although these studies are useful for initial design, there are parameters that have not been considered at present such as gas and sand effects. Emulsification and coagulation effects, which are transient effects, were also not undertaken in this steady state study.

Table 1: Details of the cases investigated

Case	Inlet Velocity (m/s)	Particles (mm)	Notes: Inlet water volume fraction = 50% Phase 1 (primary phase) = water Phase 2 (secondary phase) = oil Weir height = 1.5m
1	0.5	1	droplet diameter = 1mm
2	0.75	1	droplet diameter = 1mm
3	1.0	1	droplet diameter = 1mm
4	0.5	0.5	droplet diameter = 0.5mm
5	0.5	0.25	Droplet diameter = 0.25mm

Comparing the contours of volume fraction of oil for a velocity of 0.5, 0.75 and 1.0 m/ s with a droplet diameter of 1mm

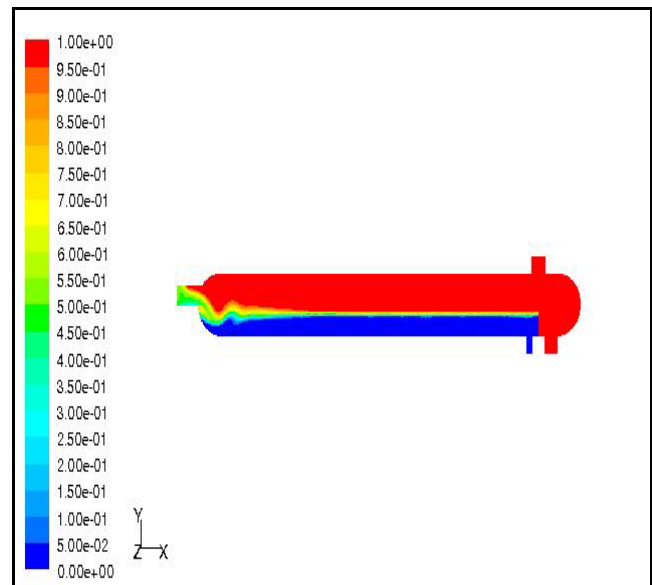


Figure 1: contours of volume fraction of oil (1mm diameter) for a velocity of 0.5m/ s

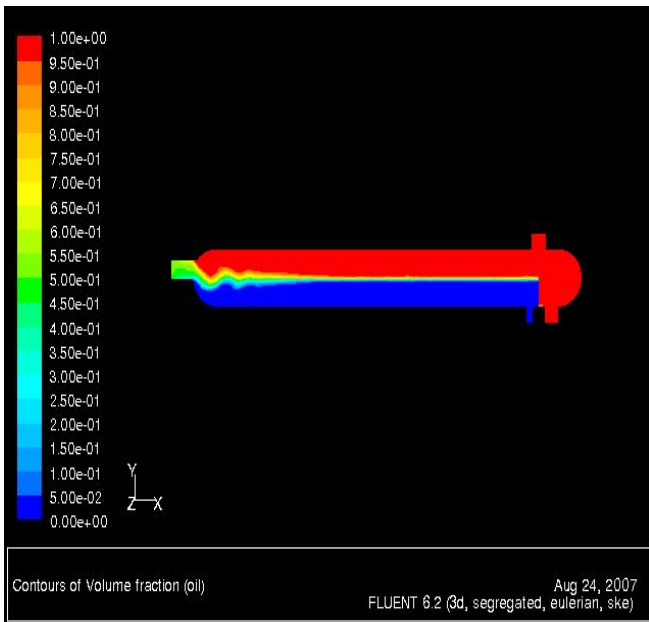


Figure 2: contours of volume fraction of oil (1mm diameter) for a velocity of 0.75m/ s. The black background colour used is an indication of the use of an intermediate value.

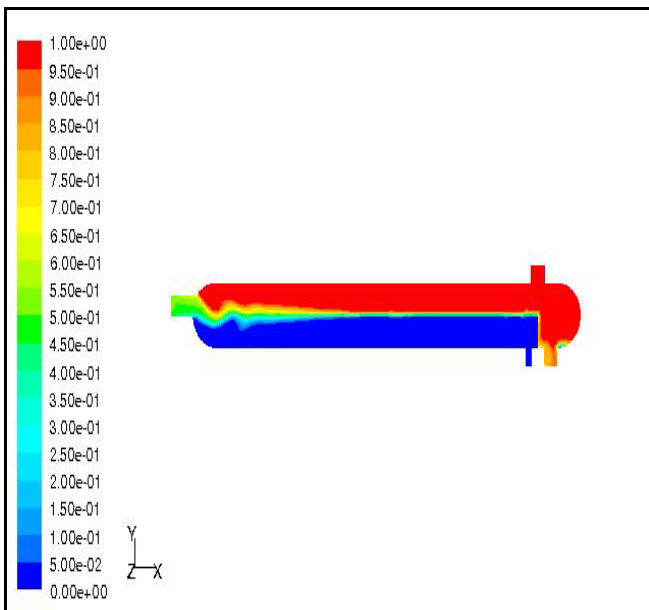


Figure 3: contours of volume fraction of oil (1mm diameter) for a velocity of 1.0m/ s

Comparing the contours of oil volume fraction of Cases1, 2 and 3 (respectively Figures 1, 2 and 3), it can be observed that the lighter oil layer floats on top of the water layer and spills over the weir into the oil chamber, where it is withdrawn, due to differences in density in all the three cases. The heavier water layer remains on the inlet side of the weir and is withdrawn through a separate interface. The weir plate allows the top layer of the oil/ water section to cascade over and accumulate in the oil only section of the separator thus

facilitating separate oil and water draw offs from the vessel. The red colour according to the contour line indicates maximum oil while blue indicates maximum water. A change in colour signifies mixed flow.

Figure 1 showed that at a mixture velocity of 0.5m/ s, the level of water in the separator is below the height of the weir, indicating that there is no overflow of water into the oil section. The weir forms a dam, which creates a section where the water can separate out of the oil. The water falls to the bottom of the separator with the oil on top, which flows over the weir and into the oil section.

An increase in mixture velocity to 0.75m/ s as depicted in Figure 2 show that the level of water in the separator tends to increase to the same height as that of the weir.

At a mixture velocity of 1.0m/ s, the level of water in the separator increased as shown in Figure 3, resulting in water spill over the weir into the oil section of the separator.

In summary, the aforementioned results show that an increase in mixture velocity results in an increase in water level to a point where it begins to overflow into the oil section of the separator. These results obtained are in agreement with those obtained by [1] and [8]. The design of the separator should take cognisance of the mixture velocity such that when the mixture velocity is high, the geometry of the separator can be changed by increasing the weir height.

Comparing the contours of volume fraction of oil for a velocity of 0.5m/ s with differing droplet diameters of 1, 0.5 and 0.25mm

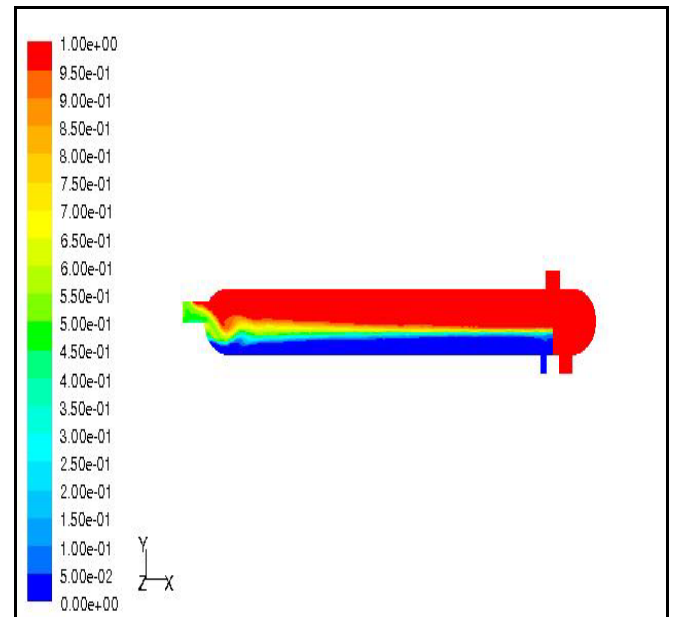


Figure 4: contours of volume fraction of oil (0.5mm diameter) for a velocity of 0.5m/ s

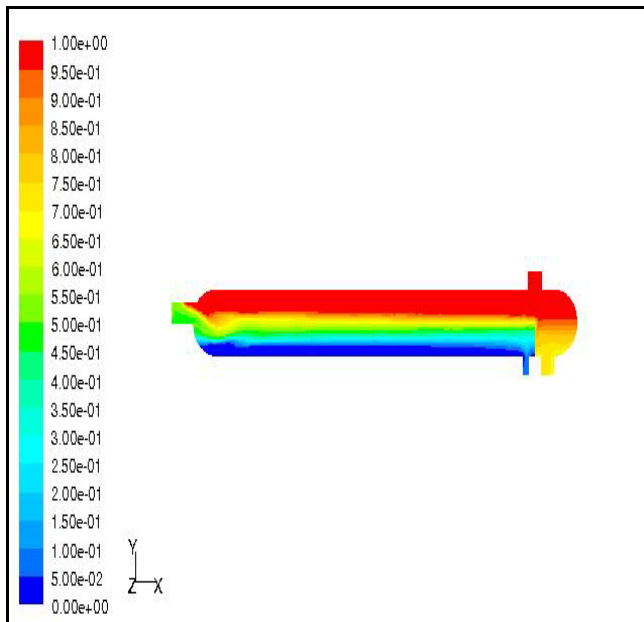


Figure 5: contours of volume fraction of oil (0.25mm diameter) for a velocity of 0.5m/ s

Comparing the contours of oil volume fraction of Cases 1, 4 and 5 (respectively Figures 1, 4 and 5), it can be observed that there are increases in mixed flow patterns in the separator when accompanied by a decrease in droplet diameter. The mixed flow pattern showed that there is no clear change from water to oil indicating that an emulsion layer exists at the interface. Formation of emulsions occurs when oil and water is agitated inside the separator. This tends to bring about a decrease in accumulation level of the liquid required for separation.

Figure 4 showed that with a droplet diameter of 0.5mm, there is an increase in mixed flow pattern. By decreasing the droplet diameter to 0.25 mm as shown in Figure 5, the degree of mixing pattern tends to be higher than that of a droplet diameter of 0.5mm. At a droplet diameter of 1mm as shown in Figure 1, the least degree of mixing occurs in comparison to those of 0.25 and 0.5mm. This is not surprising, since the velocity of separation is proportional to the square of the bubble diameter.

In summary, the aforementioned results show that with the same volume of oil, there are significant differences in separation of oil and water when different droplet diameters are selected. Indicating that the smaller droplets will result in more mixed flow than the larger ones where separation of two-phases is more prominent. These results obtained are in agreement with the works of [1], [6] and [8]. The design of the separator should take cognisance of the size of the droplet diameter so as to avoid mixed flow patterns.

Comparing the contours of velocity vector of oil for a velocity of 0.5, 0.75 and 1.0 m/ s with a droplet diameter of 1mm

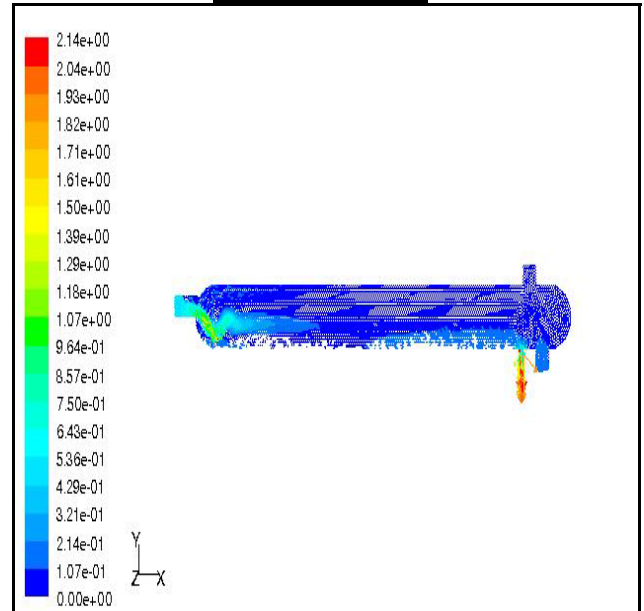


Figure 6: contours of velocity vector of oil (1.0mm diameter) for a velocity of 0.5m/ s

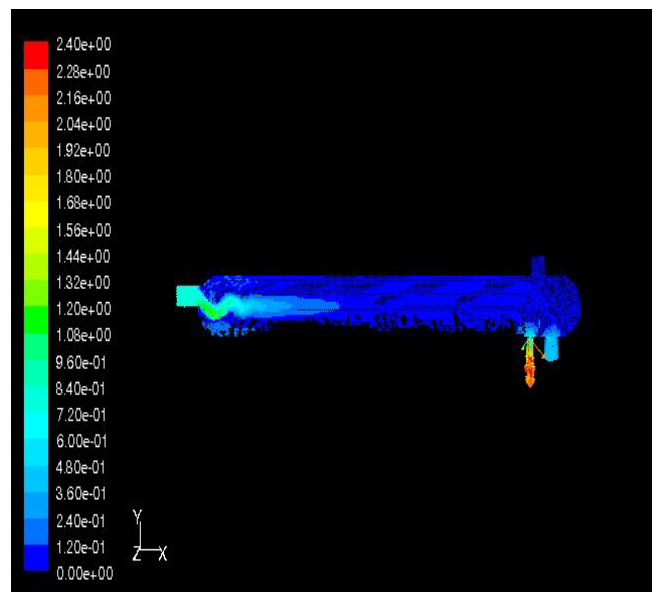


Figure 7: contours of velocity vector of oil (1.0mm diameter) for a velocity of 0.75m/ s. The black background colour used is an indication of the use of an intermediate value.

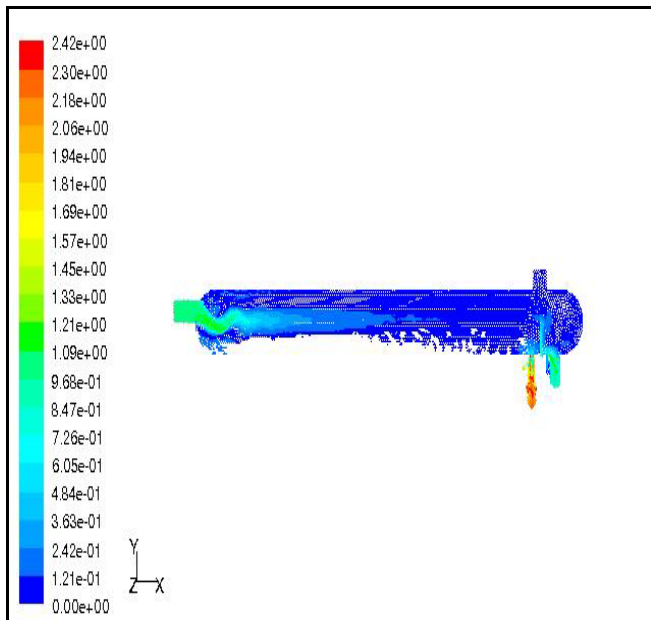


Figure 8: contours of velocity vector of oil (1.0mm diameter) for a velocity of 1.0m/ s

The intensity of velocity vector is depicted by different colours as shown. The maximum, medium and least intensity are represented by red, green and blue respectively. The velocity vector can be used to explain fluid movement with the red colour indicating maximum movement while blue colour indicating no movement of flow. And the green colour indicates average fluid movement. The velocity vector can also be used to explain phase separation depending on velocity. A change in colour indicates there is a phase separation going on. It can be observed that Figures 6, 7 and 8 shows the same colour red at the water outlet, indicating that there is a maximum intensity there. It shows that there is maximum movement of fluid. It also signifies that only one fluid is present therein. It can also be observed that there is significant turbulence from the inlet into the separator; the fluid flow is not uniform. The presence of a perforated plate at the inlet is necessary to straighten flow. Figure 6 can also be observed to show a darker colouration than Figures 7 and 8. A high turbulence level at the inlet can be observed in Figure 6 together with a much darker blue colour than Figures 7 and 8 at the oil outlet. This shows that it has the least intensity therein. It also indicates that there is no fluid movement (stagnant). Figure 7 show a higher turbulence compared to Figure 6 at the oil outlet and inlet. This shows that it has a lesser turbulence compared to Figure 8. It also indicates that there is a low fluid movement. The degree of turbulence is higher in Figure 8 compared to Figures 6 and 7. It also shows that there is more movement of fluid compared to Figure 6 and 7. In summary, the velocity vector shows that the movement of fluid and turbulence (at inlet and outlet) increases with mixture velocity at constant droplet diameter.

CONCLUSION

The modelling of oil and water separator has been successfully carried out. The results of the analysis show that there is a strong dependency of phase separation on mixture velocity and droplet diameter. Simulations using the same volume fraction of oil with different mixture velocities and droplet diameters gave different results. The results show that a mixture velocity of 0.5m/ s produced the best result when compared with those of 0.75m/ s and 1.0m/ s. Mixture velocities of 0.75m/ s and 1.0m/ s require the weir height to be increased, so as to prevent water over flowing into the oil section of the separator. The result of 1.0mm droplet diameter produced the best result when compared with those of 0.5mm and 0.25mm. The 0.5mm and 0.25mm showed a low phase separation, requiring a control mechanism to prevent mixed flow pattern. This is in positive agreement with the works of [1], [6] and [8]. The Eulerian model for a multiphase flow found in software Fluent6.2 was used for simulation, while GAMBIT2.2 was used for generating the separator geometry.

This concludes that the mixture velocity and droplet diameter are important parameters that influence/ affect the separator geometry. In consequence, computational fluid dynamics (CFD) techniques can prove useful in understanding important flow mechanisms in the separator and thus predict its performance. Furthermore, this study demonstrated the challenging task of modelling oil and water separator, and that more work has to be done to make the work more complete.

REFERENCES

- [1] Abdulkadir, M. And Hernandez-Perez, the effect of mixture velocity and droplet diameter on an oil-water using computational fluid dynamics (CFD), Accepted for the International conference on fluid flow heat transfer and thermodynamics, Cape Town, South Africa, 29-31 January 2010
- [2] API Specification 12J (SPEC 12J), 7ed, 1 October 1989
- [3] Fletcher, C.A.J., Computational techniques for fluid dynamics, Vol.1. 2nd ed, Berlin: Springer, 1991
- [4] FLUENT INC., Gambit version 2 manuals, Centerra Resource Park, 10 Cavendish Court, Lebanon, New Hampshire, USA. Available from: <http://www.fluent.com>, 2001
- [5] FLUENT INC., Fluent 6 introductory training notes, Available from: <http://www.fluent.com>, 2003
- [6] Hallanger, A., Soenstaboe, F., and Knutsen, T., A simulation model for three-phase gravity separators. Proceedings of SPE Annual Technical Conference and Exhibition. Denver, Colorado: SPE, 6 – 9 October 1996, pp. 5 – 7
- [7] Hansen, E.M.W., Heitmann, H., Laksa, B., Ellingsen, A., Stby, O., Morrow, T.B. and Dodge, F.T., Fluid Flow Modelling of Gravity Separators, London: Elsevier Science Publishers, 1991
- [8] Holdo, A.E. and Calay, R.K., Two-phase flow modelling for industrial applications, Fluid Mechanics Research Group, UK: University of Hertfordshire Hatfield Herts, 2006.
- [9] Powers, M.L., New perspective on oil and gas separator performance, SPE 20699, Annual Technical Conference and Exhibition, New Orleans. SPE, 23 – 26 September 1993
- [10] Sayda, A.F. and Taylor, J.H., Modelling and control of three – phase gravity separators in oil production facilities, 1992
- [11] White, F.M., Viscous Fluid Flow. 2nd ed, New York: McGraw – Hill, 1991