Journal of the Nigerian Association of Mathematical Physics
Volume 47 (July, 2018 Issue), pp55 – 62
© J. of NAMP

A NEW MODEL FOR STRATIFIED WAVY FLOW DYNAMICS OF A TWO PHASE INCOMPRESSIBLE FLUID FLOW

[1]Kenneth O. Apeh and [2]Vincent E. Asor

Department of Mathematics, Michael Okpara University of Agriculture, Umudike, Abia State, Nigeria

Abstract

A Computational Fluid Dynamic (CFD) model is presented to predict stratified wavy flow dynamics of a two phase incompressible fluid flow in a horizontal pipe. The simulation was done using a finite element method (FEM) based solver, COMSOL Multiphysics 4.2a, with conservative level set (CLS) method. Light viscous oil and water at 20^0C were used as the system fluids for the study, with surface tension coefficient as 0.024 N/m. The oil-water two-phase flow simulation were performed in 0.025m diameter pipe. The inflow velocity of both fluids were varied in each simulation, and the effect of the variations were analysed. Varying the inflow velocities of the fluids, the simulation successfully predicted stratified wavy flow. The simulation results show that the flow stratified with an increase in inlet velocities of both fluids and the oil volume fraction gave a maximum value at the upper part of the pipe as the flow develops. The volume fraction, pressure difference and velocity profile of the two fluids along the diameter of the pipe at the flow pattern were investigated and compared with experimental results in the literature and the results are in a good agreement.

Keywords: CFD Simulation, Horizontal Pipeline, Flow Pattern Prediction, Laminar Two-Phase Flow, CLS Method.

1.0 Introduction

Two-phase flows are found widely in nature and in a whole range of industrial application such as chemical plants, nuclear power generation and transportation of petroleum products.The simultaneous flow of immiscible liquids in a pipeline is encountered in transportation of petroleum products since water and oil are generally produced together from oil wells. In a single phase flow, the design parameters such as liquid volume fraction, pressure drop and flow patterns can be modelled easily but the presence of a secondary phase increases the complexity of the fluid flow model. Hence, understanding the process of two-phase flow of oil and water is difficult because of the complex phenomena underlying its behaviour.

In previous years much attention was on the gas-liquid flow, mainly driven by the nuclear industry where steam-water flow occurs in cooling systems. Some of the early liquid-liquid flow research was carried out in the early 1960s when it was hoped that the addition of water to single phase oil would help reduce pressure drop[1].In the 1990s, interest increased again with the need to improve the predictive models of pressure drop and traffic in multiphase pipelines. These models required that because of the difference in density and viscosity of the two fluids (oil and water), the flow cannot be assumed as one homogeneous flow and that the details of the flow pattern should be considered in determining the flow behaviour. The density difference between the two liquids has a substantial effect on the flow pattern. Two immiscible liquids of different densities tend to stratify when flowing in a pipeline. Hence, at high density difference, it is difficult to produce a dispersed flow pattern. For viscosity, the same flow patterns are observed when different oil viscosities are used, but transition from one flow regime to another may appear at different superficial velocities [1] and [2]. In two-phase flow, viscosity has a dual effect: an increase in viscosity can increase the instability of the flow due to the difference in velocity profiles at the interface of the two layers, and at the same time it helps to dissolve the energy that causes instability.

In recent years, CFD has become an industrial simulation technique for an engineering system investigation which includes fluid flow, design, performance determination and analysis. Due to easy availability and enormous increase in computer memory capacity and speed, this improvement has been achieved, resulting in a reduction in costs of simulation compared to experimental work.

In 2012, core annular flow through sudden contraction and expansion of pipe, using CFD was simulated in [3]. In their simulation, they used a volume of fluid (VOF) method to simulate core annular flow of lubricated oil and water and monitored profiles of

Corresponding Author**:** Kenneth O., Email: okwudilikenneth@gmail.com, Tel: +2348132242834

velocity, pressure and volume fraction over a wide range of oil and water superficial velocities for an abrupt expansion and contraction. They validated their simulated result with experimental results and observed a satisfactory match.The fluid dynamics of two-phase flow (oil-water) in pipe with leakages was studied in [4]. They solved the governing equation of the flow, using ANSYS CFX commercial code with the aid of a structured mesh of a horizontal pipe with three holes of leaks. They used an Eulerian-Eulerian model, using water as the dispersed phase and oil as continous phase with constant fluid properties. They solved the *influence of leakages* in single-phase (oil) and two-phase (oil and water) by monitoring the pressure profile and volume fraction along the pipe. They found that the volume fraction of oil in the two-phase mixture injected into the duct affected the amount of oil leaked and that the presence of second leak on downstream of the initial leakages on the duct affected the oil flow of the first one, which becomes established with a lower flow rate compared to the situation where the first leak occurred alone. The transition boundaries of different flow patterns for moderately viscous oil-water two-phase flow through a horizontal pipeline using CFD simulation was studied in [5]. They used a volume of fluid (VOF) method with effect of surface tension to predict the flow pattern and found that the simulation results predict transition boundaries of wavy stratified-to-stratified mixture flow. Also, in 2014, the hydrodynamics of slug, stratified wavy, stratified mixture and annular flow using the CFD software, ANSYS FLUENT 6.2 with a volume of fluid (VOF) was studied and simulated in [6]. They found that in the annular flow, total pressure of the mixture decreases with increase in oil velocity due to the fact that pipe cross section is completely wetted with water, and the simulated oil volume fraction shows maximum at the centre in core annular flow, while in stratified flow, oil volume fraction shows maximum at the upper side of the pipe. Furthermore, in 2014, rig facility and CFD code FLUENT 6.3 was used in [7] for their numerical and experimental study. In their experimental study, they found that the pressure drop in oil-water flow is dependent on flow pattern condition, pressure drop increases with increasing mixture velocity and that pressure decreases in fluctuation manner along the pipe due to heat transfer effect, gradually until it reached a minimum value at the end of the pipe. In their numerical study, they found that an annular flow regime exists over a wider range of superficial velocities, which shows that flow pattern in gas-liquid flow cannot be used to predict the flow pattern of liquid-liquid flow. They also found that in numerical simulation, the pressure drop depend on the changes in oil-water superficial velocities. Oil-water stratified flow in 0.0254meter internal diameter horizontal pipe using computational fluid dynamics (CFD) software FLUENT 6.2 was studied in [8]. They predicted pressure gradient based on different flow velocities and found that as velocity increases the pressure gradient also increases and also argued that extending the pressure gradient prediction can help to examine the effect from different water volume fractions.

We have shown that many researchers have used finite volume based computational fluid dynamics (CFD) code for their studies. In this work, we use CFD simulations to investigate stratified wavy flow pattern of light viscous oil-water two-phase flow through a horizontal pipe by varying the inflow velocities of both fluids, using a finite element (FE) based CFD code, COMSOL Multiphysics 4.2a with laminar two-phase flow conservative level set method. Volume fraction profile of the two fluids, pressure difference and velocity profile along the diameter of the pipe atthe flow pattern were investigated. The simulated results were compared with experimental results of [5]and their simulated results in [6] where the finite volume (FV) based CFD code, ANSYS FLUENT 6.2 with a volume of liquid method including effect of surface tension and gravitational force was employed.

2.0 Model Development

Two-phase flow of oil and water are simulated using COMSOL Multiphysics 4.2a. Laminar two-phase flow, conservative Level set method was adopted in order to properly solve for interface of the two fluids, with an introduction of contact angle on the wall to take care of wettability of the fluid. The geometry and its detailed dimensions used in this work is shown in figure 2.1. Oil and water which are the fluids used for the work were introduced into the test section of the pipe through a T-junction at the entry section, oil and water entered into the pipe in a vertical and horizontal direction, respectively. The geometry of the system has three sections, the horizontal test section which have internal diameter of 0.025 meter, the horizontal water inlet with internal diameter of 0.019 meter, the vertical oil inlet with internal diameter of 0.019 meter. The simulations were done in a 2D model with an unsteady state study chosen for the computation. The properties of the two fluids are as stated in table 2.1.

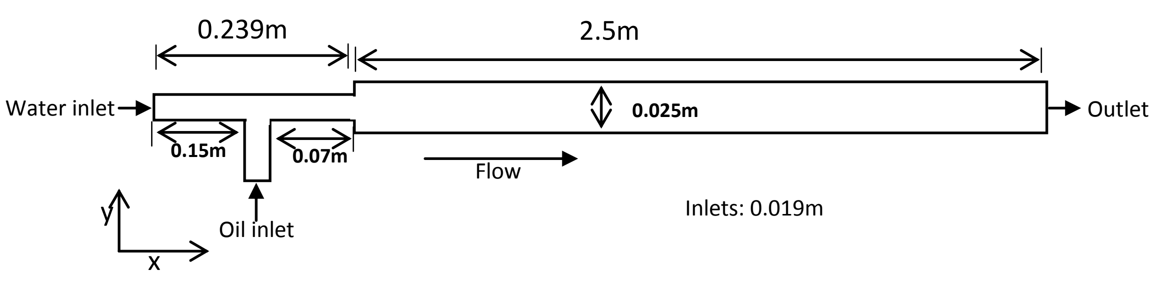

Figure 2.1: *Geometry and detailed dimensions of the horizontal pipe.*

Table 2.1: Fluid properties for oil and water.

	Density	Viscosity
Water	$1000kg/m^3$	$0.001Pa*s$
Oil	$890kg/m^3$	$0.107Pa*s$

2.1 Governing Equations

The governing equations as employed in COMSOL Multiphysics will be the Navier-Stoke equations for incompressible fluid flows and interface tracking algorithm which in this work is CLS equation, using surface tension coefficient σ as 0.024N/m, re-initialization parameter as 1m/s and default interface thickness, which is $h/2$, where h is the characteristic mesh size in the region passed by the interface.

In the level set (LS) method, there is no built in volume conservation which could lead to incorrect loss or gain of mass occurring mainly in regions where the evolving surface has high curvature. With time the error will accumulate as simulation continues. To overcome this deficiency, a conservative level set method was developed in [9] and [10], which is implemented in the finite element based solver COMSOL Multiphysics, to simulate incompressible two-phase flow, where gravity and surface tension effects are included.

The level set function $\phi(x,t)$ is represented by distance function but, in the conservative level set method the level set function is represented by a smeared Heaviside function $H_{\in}(\phi)$, so that the level set function will change smoothly across the interface from 0 to 1, where the interface is defined by 0.5 iso-contour of the level set function.

$$H_\epsilon(\phi)=\begin{cases}0, & if\ \phi<-\epsilon\\ \frac{\phi+\epsilon}{2\epsilon}+\frac{1}{2\pi}\sin\left(\frac{\pi\phi}{\epsilon}\right), & if\ |\phi|\le\epsilon\\ 1, & if\ \phi>\epsilon\end{cases} \qquad (2.1)$$

where

$$|\phi(x)|=d(x)=\min_{x_\Gamma\in\Gamma}(|x-x_\Gamma|)$$

andϵ is half the thickness of the interface, which depend on the grid size in the mesh, such that it will be sufficiently determined.

In the numerical simulation, an intermediate step has to be taken in order to maintain the thickness of the interface. This step which are implemented by solving the conservation law adds an artificial compression on the system

$$\frac{\partial\phi}{\partial\tau}+\nabla\cdot f(\phi)=0 \qquad (2.2)$$

where τ is an artificial time and f is the artificial flux. Equation (2.2) works in the regions where $\phi\in(0,1)$ and in the direction of the interface, the artificial flux f is taken as $f=\phi(1-\phi)\hat{n}$, which using the normal unit vector equation becomes

$$f=\phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}. \qquad (2.3)$$

If equation (2.2) is used in simulation, there will be discontinuities at the interface. To avoid the discontinuities, we introduce a small perturbation (diffusion term) to the equation to get

$$\frac{\partial\phi}{\partial\tau}+\nabla\cdot f(\phi)=\epsilon\nabla^2\phi$$

which can be rearranged as

$$\frac{\partial\phi}{\partial\tau}=\epsilon\nabla^2\phi-\nabla\cdot f(\phi). \qquad (2.4)$$

In COMSOL, the LS equation $\frac{\partial\phi}{\partial t}+\nabla\cdot(\bar{u}\phi)=0$ and (2.4) can be implemented by setting the original level set method with real time to be equal to the artificial time, which is

$$\frac{\partial\phi}{\partial t}+\nabla\cdot(\bar{u}\phi)=\epsilon\nabla^2\phi-\nabla\cdot f(\phi).$$

Substituting the artificial flux (2.3), the equation in conservative form becomes

$$\frac{\partial\phi}{\partial t}+\nabla\cdot(\bar{u}\phi)=\nabla\cdot\left[\epsilon\nabla\phi-\phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right]. \qquad (2.5)$$

The right hand side of equation (2.5) has a diffusion term which can enlarge the width of the interface during simulation. To control this diffusion, so that the thickness of the interface ϵ remain constant, a re-initialization or stabilization parameter γ is introduced at the right hand side of equation (2.5) as

$$\frac{\partial\phi}{\partial t}+\nabla\cdot(\bar{u}\phi)=\gamma\nabla\cdot\left[\epsilon\nabla\phi-\phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right]. \qquad (2.6)$$

The re-initialization parameter γ is carefully selected for each specific problem, because if the value is too small, the thickness of the interface might not remain constant, and oscillation in the level set function will occur because of numerical instabilities, and if it is too large, the interface moves incorrectly. Using the conservative form (equation 2.6), exact numerical conservation of the integral of ϕ will be obtained.

This conservative level set equation (2.6) coupled with the Navier-Stokes equations

$$\rho\left(\frac{\partial u}{\partial t}+(\bar{u}\cdot\nabla)\bar{u}\right)=\nabla\cdot\left[-pI+\mu(\nabla\bar{u}+\nabla\bar{u}^T)\right]+\rho g+f_{st} \qquad (2.7)$$

$$\nabla\cdot\bar{u}=0 \qquad (2.8)$$

are what COMSOL uses to solve incompressible two-phase flows, conservative level set method, where g is the acceleration due to gravity, σ is the surface tension, ρ and μ are the density and viscosity, respectively, which varies smoothly over the interface. The internal force boundary condition between the two fluids f_{st} (the surface tension force) is calculated as

$$f_{st}=\sigma\kappa\hat{n}\delta_\epsilon(\Gamma,x) \qquad (2.8a)$$

where $\hat{n}$ and κ are unit vector and curvature field respectively and the Dirac delta function $\delta_\epsilon(\phi)$ is approximated as

$$\delta_\epsilon(\phi) = 6|\phi(1-\phi)||\nabla\phi| \qquad (2.8b)$$

2.2 Initial and Boundary Conditions

Initially, in all the simulations, the pipe is filled with water, while oil is introduced in the pipe through the oil inlet with no initial interface. At the initial stage the velocities of both oil and water are zeros, with default reciprocal initial interface distance.

In COMSOL, the level set variable ϕ is used to denote volume fraction of fluid in a fluid flow. The total volume fraction of both fluids are equal to one, that is $\phi_w + \phi_o = 1$. At the water inlet, the volume fraction of oil ϕ_o is zero, which means that volume fraction of water ϕ_w is one. Hence, only water enters through the water inlet with its superficial velocity which are varied in each simulation. At the oil inlet, the volume fraction of oil ϕ_o is one, which also shows that it is only oil that enters through the oil inlet. At the outlet, the boundary is set as "pressure, no viscous stress", and pressure equal to zero. At this boundary, volume fraction is not set because it is an outflow condition.In other to account for the interfaces of the two fluids and the solid wall of the pipe, wetted

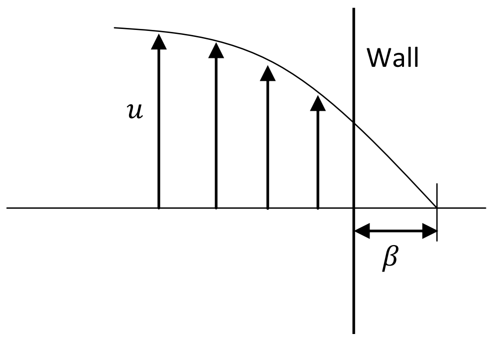

Figure 2.2: *Description of slip lengthβ.*

wall boundary condition was used. In level set method, wetted wall boundary condition sets the velocity component normal to the wall to zero,$\bar{u}\hat{n}_{wall} = 0$, which is suitable for solid walls in contact with a fluid interface and introduces a wall frictional boundary force of the form

$$F_{fr} = -\frac{\mu}{\beta}\bar{u} \qquad (2.9)$$

β is the slip length.

The boundary does not set the tangential velocity component to zero as a result of setting the velocity component normal to the wall as zero, nevertheless, the extrapolated tangential velocity component is zero at the distance outside the wall which is the slip length (figure 2.2).

Using finite element method to solve the Navier-Stokes equations, the equations are multiplied by a test functions and then integrated over the computational domain. This approach is used in laminar two-phase flow interface and surface tension effect. As a result of the partial integration of the surface tension force in the Navier-Stokes equations, the boundary condition introduces a boundary integral over the computational domain as

$$\int_{\partial\Omega} test(\bar{u}) \cdot \left[\sigma\left(\hat{n}_{wall} - (\hat{n}\cos\theta)\right)\delta\right] dS$$

where θ is the contact angle, which is the angle between the fluid interface and the wall.

In COMSOL Multiphysics, to use wetted wall boundary condition, the values of the contact angle θ, in radian and slip length β, in meters are needed. The best value of β is the mesh element size h, and the value of θ used is 2.99406215 radian (171.55 degree).

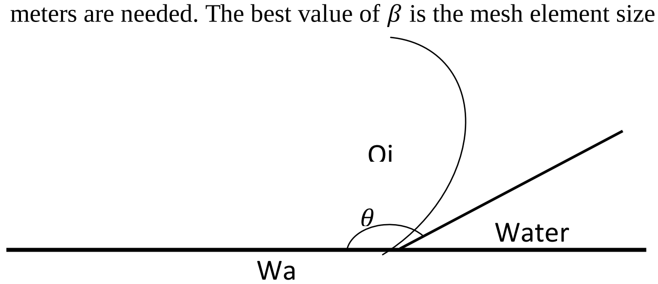

Figure 2.3: *Description of contact angle θ.*

2.3 Mesh

In fluid flow, the partial differential equations are not always amenable to analytical solutions, especially in complex cases. Hence, in order to simulate a fluid flow and get a good approximated result, flow domains are divided into smaller sub domains called elements or cells. The collection of these elements is known as mesh. In this work, the meshing of the 2D geometry has been done using sequence type called physics-controlled mesh. In the entire domain, 33,634 triangular elements mesh generated (see figure 2.4).

The partial differential equations are solved inside each of these elements in the domain. With proper simulation settings, good continuity of solutions across the interfaces between two elements are carefully done, so that the approximate solutions inside each elements can be collected to give a good approximation of fluid flow in the entire domain.

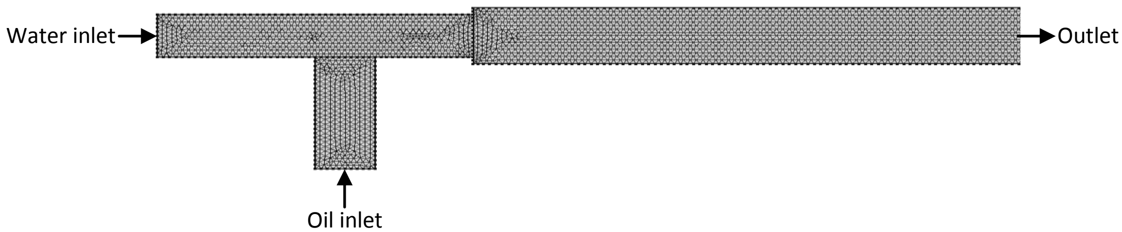

Figure 2.4: *Mesh geometry of the entire domain of the pipe.*

3 Simulation Settings

In this work, the fully coupled equations are solved using time stepping method, Backward Differentiation Formula (BDF) as the Time-Dependent Solver for the simulations with free solver time stepping. Using global method, the absolute tolerance is set to scale with absolute tolerance value of 0.001 for all the variables. We used Backward Euler for the consistent initialization and exclude algebraic for the error estimation with no complex number. To allow the flow to fully develop, the study time range were set as 0, 0.5, and 100 with relative tolerance as 0.01. In the advance setting, the matrix symmetry and null-space function were set to automatic, to allow the solver look for zero-filled rows or columns in the mass matrix as a means of detecting a differential-algebraic equation.

We used automatic damping method, so that the solver will automatically determine a damping factor in each iteration, with initial damping factor 1, minimum damping factor 0.0001, restriction for step-size update 10 and recovery damping factor 0.75. The terminating technique used is tolerance, with maximum number of iterations 1000 and tolerance factor of 1. The PERDISO Direct solver was used as the linear system solver. Nested dissection multithreaded preordering algorithm with row preordering and pivoting perturbation value of 1e-8. The error estimate check is set to automatic with factor of 400. The velocities and gravitational force for the simulation are in table 3.2.

Table 3.2: *Simulation velocities.*

Velocities used for simulation

Water velocity, U_w(m/s)	Oil velocity, U_o(m/s)	Gravity force, g(m/s^2)	Simulation results
0.23	0.1	9.81	Stratified wavy flow (SW)

4 Results

Simulations have been successfully performed in two dimension (2D) geometry, with COMSOL settings as described under simulation settings. The laminar two-phase flow, conservative level set method used, successfully predicted the stratified wavy flow pattern. The pattern predicted were simulated by varying the inflow velocities of the two fluids (oil and water). The inlet velocities of water and oil are 0.23m/s and 0.1m/s respectively as stated in table 3.2 above. These velocities were selected from the velocities used in[5] in their experimental work.When the velocities of the water and oil were increased to 0.23m/s and 0.1m/s respectively, as the flow develops, oil and water stratified with a wavy oil and water interface where observed. This pattern which is represented in figure 4.1, is called stratified wavy flow.In the flow pattern, the flow features; volume fraction, pressure and velocity distributions were examined.

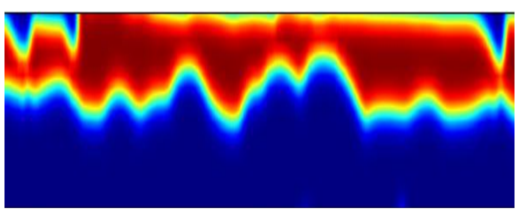

Figure 4.1: *Stratified wavy flow.*

4.1 Volume Fraction

Volume fraction is one of the most significant parameters used in classifying two-phase flow. It is the physical value that is used in determining other significant parameter, such as the two-phase density and viscosity. In turn, the two-phase density and viscosity, helps in finding the average velocity of the two-phases, which is very significant in the model for predicting flow patterns and pressure drop (or pressure gradient).

The volume fraction of oil shows maximum value at the upper part of the pipe, which shows buoyancy effect with respect to gravitational force, in agreement of oil being less dense than water.

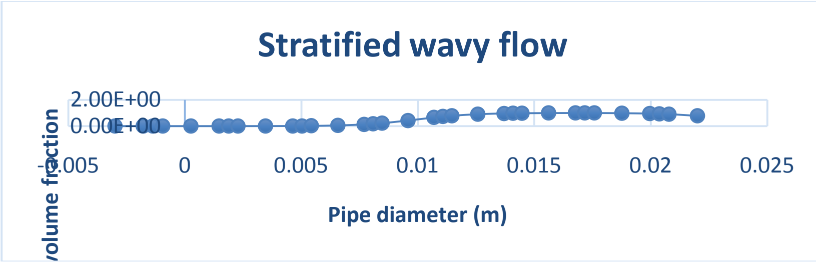

Figure 4.2: *Diameter distribution of oil volume fraction.*

Figure 4.2 shows the distribution of oil volume fraction which is higher at the upper part of the pipe, but because the pipe is wetted with water, the oil volume fraction reduces as it approaches the pipe wall.

4.2 Pressure Profile

Pressure is asignificant parameter in two-phase flow, which governs the pumping power needed to transport two-phase fluids. In the horizontal pipe flow with effect of gravitational force, pressure act in opposite direction to the force of gravity, in other to balance the force acting on the fluids. Pressure decreases upward the pipe diameter as gravitational force act downward the pipe diameter.

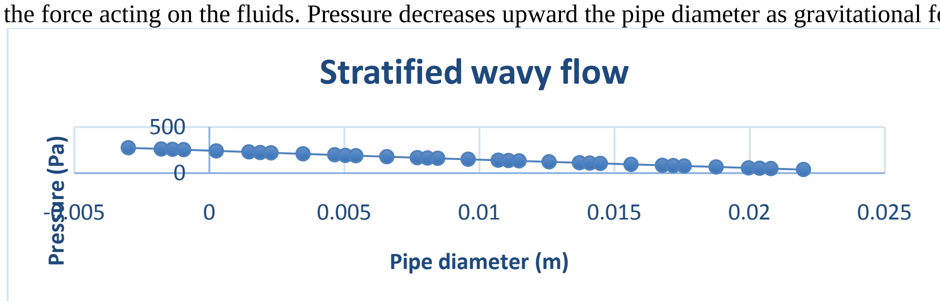

Figure 4.3: *Pressure distribution across the pipe diameter.*

In figure 4.3, it is clear that the pressure decreases smoothly upward across the pipe diameter in a linear form, because both the oil and water phase are in laminar flow.

4.3 Velocity Profile

In the flow pattern, because the oil does not break into slugs, the velocity profile at 50 and 100 seconds looks slightly similar, but with different values at each point across the pipe diameter due to the wavy nature of the flow (see figure 4.4). As the flow develops, the velocity increases at the center of the pipe close to the interface, but at the interface, velocity shows sharper decrease when the flow have reached 100 seconds. This shows that the more the flow develops, the better the approximation for the velocity distribution.

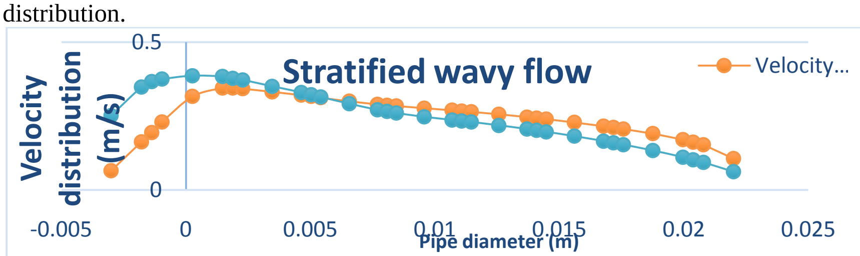

Figure 4.4: *Velocity distribution for stratified wavy flow.*

4.4 Parameter Relationship

In the system, there is a gravitational effect which act on the fluid vertically downward. Because of this effect, as oil enters the pipe, there is a buoyance effect which pushes oil up to the upper part of the pipe. Figure 4.5 shows clearly this buoyance effect. This figure shows that at the lower side of the pipe, volume fraction of water is one ($\phi_w = 1$), which automatically implies that volume fraction of oil is zero ($\phi_o = 0$).

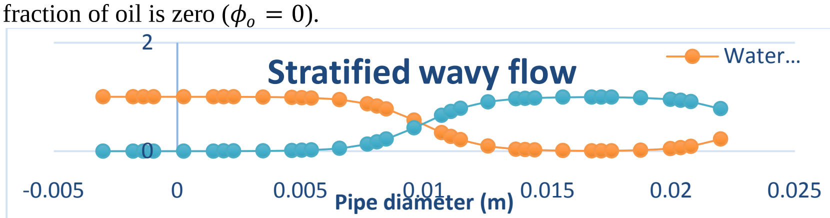

Figure 4.5: *Distribution of volume fraction of oil and water across pipe diameter.*

The figure also shows that at the center of the pipe, the volume fractions of oil and water are equal (0.5), and this indicate the interface between the two fluids.
Using volume fraction of oil, the relationship between velocity and viscosity can be predicted. Figure 4.6 pictured how the velocity reduces with respect to volume fraction of oil, because in the system, oil is more viscous than water.

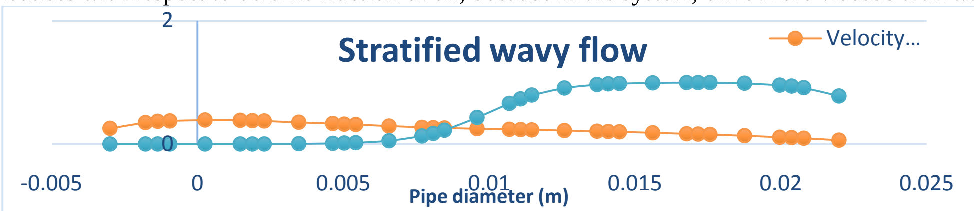

Figure 4.6: *Distribution of oil volume fraction and velocity across the pipe diameter.*
The figure shows that as oil volume fraction increases, the velocity decreases upward the pipe diameter because of the viscosity of the oil.

4.5 Derived Values

Using cut line 2D in data set, and line maximum in derived values which are features of COMSOL Multiphysics results analysis, the following values in table 4.1 and 4.2 are obtained for slug and stratified wavy flow respectively at 50 and 100 seconds.
Table 4.1: *Maximum values for Oil volume fraction, Pressure and Mixture velocity at upper and lower part of the pipe.*

Stratified wavy flow

	Upper part of the Pipe		Lower part of the Pipe	
Time (*seconds*)	50	100	50	100
Oil volume fraction	0.95221	0.9892	0.02426	0.01205
Pressure (Pa)	68.5152	66.43456	253.68418	252.15973
Mixture velocity (m/s)	0.17893	0.16104	0.41891	0.42397

4.6 Validation

The simulated result (stratified wavy flow) were compared with experimental results of [5] and their simulated results in which they used volume of fluid (VOF) method to study in 2014,[6]. Figure 4.7 shows the experimental result of stratified wavy fow, figure 4.8 shows their simulated result with VOF method, and figure 4.11 represent the simulated stratified wavy flow of this work with conservative level set (CLS) method.
Comparing the simulated result with the experimental result, the CLS method is more similar to the experimental result than the VOF method in terms of clear interfaces between the two fluids, but in wetting performance, VOF method is better than CLS method.
The stratified wavy flow of simulated result has been compared with the stratified wavy flow of their experimental and simulated result at the same buoyance conditions.

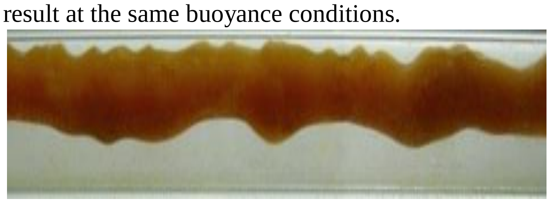

Figure 4.7: *Stratified wavy flow, experimental result*(Anand, et al., 2013, p. 77)

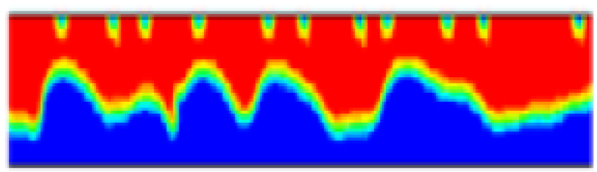

Figure 4.8: *Stratified wavy flow with VOF method* [6].

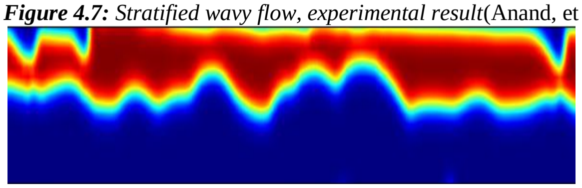

Figure 4.9: *Stratified wavy flow with CLS method (this work).*
The result shows that CLS method successfully approximated the stratified wavy flow pattern with clear interface with its waviness. The simulated result shows that at the interface the amplitude of the wave increases with an increase in mixture velocity which suggest that CLS method is capable of predicting the sensitive wavy nature of stratified wavy flow at the interface. Validating with their simulated result, in stratified wavy flow the two simulated results looks more similar. VOF method have more oil in the pipe than CLS method, but both have the oil stick to the upper wall of the pipe more than the experimental result, which shows that both method lost some degree of wetting performance in stratified wavy flow.
Figure 4.10 and 4.11show the simulated mixture velocity distribution across pipe diameter and the simulation mixture velocity distribution across radial distance of [6] for stratified wavy flow respectively. These two figures shows a good approximation

between the simulated result and the experimental result. In the simulated result the maximum velocity is close to 0.4m/s, while experimental result (black line) have maximum velocity approximately 0.4m/s. The small difference being that in the experiment, inlet velocity of oil is 0.2m/s and that of simulation is 0.1m/s.

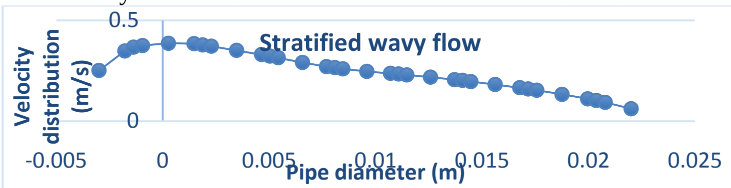

Figure 4.10: *Velocity distribution for stratified wavy flow.*

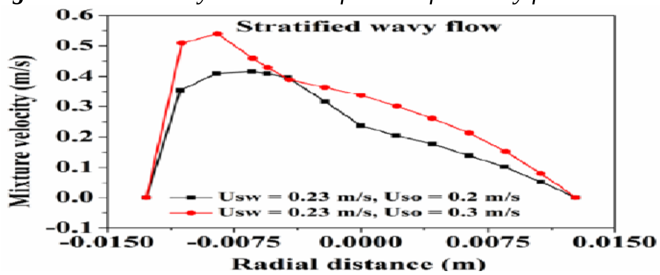

Figure 4.11: *Velocity distribution of stratified wavy flow across radial distance*[6].

In general, these comparisons gave an indication that laminar two-phase flow, CLS method is a good approach for simulating stratified wavy flow of oil and water in COMSOL Multiphysics.

5 Conclusion

Computational fluid dynamics (CFD) study has been conducted to predict flow patterns of light viscous oil-water flow through a horizontal pipe.Oil and water were introduced into the test section of the pipe through a bifurcation. Water was introduced into the pipe through horizontal inlet and oil through vertical inlet with both inlet diameter as 0.019m. Laminar two-phase flow, level set method was adopted for the simulation with time-dependent study.The study has been simulated in 33,634 elements mesh which was generated, using sequence type Physics-controlled mesh. Simulations were performed using laminar two-phase flow, level set method for slug and stratified wavy flow patterns. The two flow patterns have been successfully predicted, using CFD package in finite element based software COMSOL Multiphysics 4.2a, in 2 dimensional geometries. The geometry of the system was divided into three parts, the horizontal part which contain the water inlet, the vertical part which contain the oil inlet, and the horizontal test section where the two fluids flow together (see figure 2.1).

In modelling oil-water two-phase flow, difficulties always arise from the existence of interfaces and discontinuities associated with the model. In COMSOL, we solved this problem by using conservative level set interface which contain re-initialization techniques that takes care of deterioration of the level set function at the interface.Similarly, using two stability control in COMSOL, which are consistence and inconsistence stability techniques also helped in predicting a good numerical approximation to the experimental results. These techniques also helped in reducing the mesh so that it will not be too dense, which in turn helped in reducing the simulation time.

Finally, the flow pattern predicted, were validated with experimental result of [5] and their simulated results,[6].The validation showed a good agreement with the experimental result, which implied that, we have successfully predicted that laminar level set can be used to predict stratified wavy flow in two-phase flow of oil and water if the Reynolds number is at laminar level or at the transition level, using COMSOL Multiphysics.

References

[1] Rusell, T., Hodgson, H. G., & Govier, H. G. (1959). Horizontal Pipline Flow of Mixtures of Oil and Water. *The Canadian Journal of Chemical Engineering., 37*, 9-17.

[2] Charles, M., Govier, H. G., & Hodgson, H. G. (1961). Horizontal Pipline Flow of Equal Density Oil-Water Mixture. *The Canadian Journal of Chemical Engineering., 39*, 27-36.

[3] Kaushik, V., Sumana, G., Gargi, D., & Prasanta, K. D. (2012). CFD Simulation of Core Annular Flow Through Sudden Contraction and Expansion. *Journal of Petroleum Science and Engineering, 86, 87*, 153-164.

[4] Morgana, d. A., Severino, R. N., & Antonio, G. d. (2013). Theoretical Evaluation of Two-Phase Flow in a Horizontal Duct With Leaks. *Advances in Chemical Engineering and Science, 3*, 6-14.

[5] Anand, B. D., Anjali, D., Vinayak, V., Bharath, K. G., Ashok, K. D., & Tapas, K. M. (2013). CFD Simulation and Validation of Flow Pattern Transition Boundaries During Moderately Viscous Oil-Water Two-Phase Flow Through Horizontal Pipeline. *International Journal of Chemical, Molecular, Nuclear, Materials and Metallurgical Engineering, 7*(1), 72-77.

[6] Anand, B. D., Ashok, K. D., & Tapas, K. M. (2014). Oil-Water Two-Phase Flow Characteristics in Horizontal Pipeline -A Comprehensive. *International journal of Chemical, Nuclear, Materials and Metallurgical Engineering, 8*(4), 336-340.

[7] Esam, M. A., & Zahra'a, A. A. (2014). Simulation and Experimental of Oil-Water Flow With Effect of Heat Transfer in Horizontal Pipe. *International Journal of Mechanical Engineering and Applications, 2*(6), 117-127

[8] Adib , Z. A., Jaswar, K., Yasser, M. A., & Abd, K. J. (2014). Stratified Oil-Water Two-Phase Flow of Subsea Pipeline. *Journal of occean, Mechanical and Aerospace, 14*, 19-24.

[9] Olsson, E., & Kreiss, G. (2005). A Conservative Level Set Method for Two-Phase Flow. *Journal of Computational Physics, 210*, 225-246.

[10] Olsson, E., Kreiss, G., & Zahedi, S. (2007). A Conservative Level Set Method for Two-Phase Flow II. *Journal of Computational Physics, 225*, 785-807.