*Transactions of the Nigerian Association of Mathematical Physics*
***Volume 7, (March, 2018), pp47 – 54***


# A NEW MODEL FOR THE LAMINAR FLOW OF TWO IMMISCIBLE FLUIDS THROUGH A HORIZONTAL PIPE

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

**Department of Mathematics, Michael Okpara University of Agriculture, Umudike, Abia State.**

***Abstract***

***We model the flow of two immiscible fluids through a horizontal pipe with low Reynolds number using computational fluid dynamics. A light oil and water with constant density and viscosity at $20°C$ were used as the system fluids for the investigation, with surface tension coefficient $0.024N/m$.The numerical simulation were carried out in $0.025m$ diameter pipe using a finite element (FE) based software solver, COMSOL Multiphysics 4.2a with level set (LS) method. Theinflow velocity of both fluids were varied and varying the velocities, the simulation successfully predicted slug flow pattern. 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 to the experimental results in literature and the numerical results showed a good approximation to the experimental results.***

**Keywords:** CFD, Oil – water flow, Horizontal pipe, Reynolds number, Flow pattern, Level set method

## 1.0 Introduction

Two-phase flow is a phenomenon which occurs in many industries such as chemical plants, nuclear power generation and transportations of petroleum products in petroleum industries. The simultaneous flow of immiscible liquid-liquid in a pipeline is encountered in transportation of petroleum products since water and oil are generally produced together from oil wells. There is a significant effect due to water presence during transportation of oil from the well to the processing facility. Transportation is very important in offshore oil fields by using pipelines to transfer the oil to an onshore or to an existing facility where they can be processed. These pipes lie on the seabed in a horizontal or near horizontal orientation. However, understanding the process is still difficult because of the complex phenomena underlying its behaviour.

The design parameters such as liquid volume fraction, pressure drop and flow patterns in a single phase flow in a pipe can be modelled easily. However, the existence of a secondary phase such as water can lead to increase in the complexity of the hydrodynamics and creates different challenges in modelling liquid-liquid (oil-water) two-phase flow. The introduction of water into oil transportation pipelines can have several effects such as the complex interfacial structure between oil and water which complicates the hydrodynamic prediction of the fluid flow. Knowledge of the hydrodynamics of such two-phase liquid-liquid flow is essential for the design of extractor, mixture-settlers, pipeline network, transportation pipeline, downstream separators, etc.

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 as reported in [1].In the 1990s interest increased again with the need to improve the predictive models of pressure drop and hold-up in multiphase pipelines. These models required that because 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.

In recent years, Computational Fluid Dynamics (CFD) has become an industrial simulation tool for engineering systems investigation which includes fluid flow, design, performance determination and analysis. Due to easy accessibility 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 2007, the research of oil-water pipe flow in the past decade was reviewed in [2]. He discussed flow pattern identification and its transition, phase inversion modelling and pressure drop prediction. He argued that the main difficulties in modelling

Correspondence Author: Vincent E.A., Email: vincent.asor@mouau.edu.ng, Tel: +2348102450388, +2348132242834 (KOA)

oil-water flows arise from the existence of interfaces and discontinuities associated with them. He concluded from his result that phase inversion point (PIP) prediction and pressure drop are key parameters in design of oil-water flow systems.In 2013, a study of the fluid dynamics of two-phase flow (oil-water) in pipe with leakages was reported in [3]. 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. Also, in 2013, a study of the transition boundaries of different flow patterns for moderately viscous oil-water two-phase flow through a horizontal pipeline using CFD simulation was reported in [4]. 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. Furthermore, 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) method was simulated and reported in [5]. 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. The propagation of gravity waves along the common surface of two superposed liquids was studied using a selection of Cartesian axes to analyze the effect of a small perturbation for fluid at low Reynold's number [1]. He used three dimensional steady-state mathematical model for oil-in-water dispersion flow to study the effect of lift force on dispersed phase behaviour in vertical pipe by adopting an Euler/Lagrange multiphase method in CFD code ANSYS FLUENT to investigate the interaction between the oil droplets and water (dispersion and continuous phase respectively) and solved the flow field of the continuous phase by using Reynolds-average Navier-Stokes conservation equation with k-$\epsilon$ turbulence model and standard wall function. It was suggested that the shear-lift force plays an important role in the overall force balance acting on the droplets and influences the radial distribution of the oil droplets, which means that the oil droplets tends to concentrate at the core of the pipe rather than near the wall. Furthermore, it was also suggested that the oil droplets diameters depend on the mixture velocity for the case in his study.
The literature review above shows that many researchers have used finite volume based computational fluid dynamics (CFD) code for their studies. We now attempt to use CFD simulations to investigate slug flow pattern of viscous oil-water two-phase flow through a horizontal pipe by varying the inflow superficial velocities of both fluids, using a finite element based CFD code, COMSOL Multiphysics 4.2a with level set (LS) method. Volume fraction profile of the two fluids, area average pressure across a cross section and velocity profile along the radius of the flow pattern were investigated. The simulated results were validated with experimental results of [4] and the simulated slug flow exhibited a good approximation with the experimental results of the slug flow pattern.

## 2.0 Model Development

Two-phase flow of oil and water are simulated using COMSOL Multiphysics 4.2a. Laminar two-phase flow, Level set method was adopted in other 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 have four sections: the horizontal test section (the 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 and the outlet of the pipe. The simulations were done in a 2D model with an unsteady state study chosen for the computation.

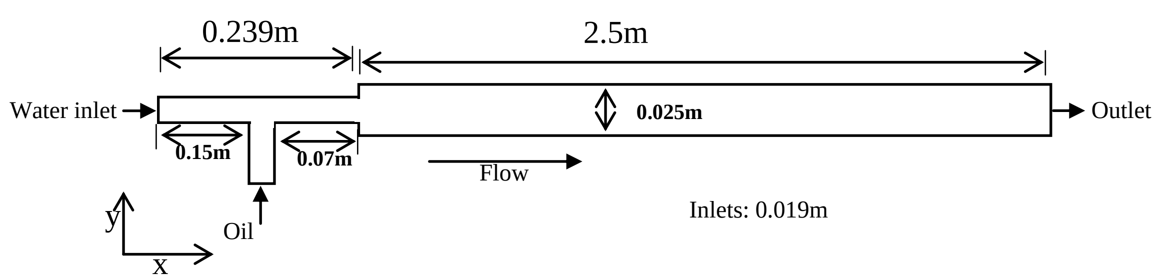


***Figure 2.1:*** *Geometry and detailed dimensions of the horizontal pipe.*

The properties of the two fluids are as stated in table 2.1.
***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 Mesh

In fluid flow, the partial differential equations (in this case, the Navier-Stokes equations and the level set equation) 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 were generated (see figure 2.2).
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.

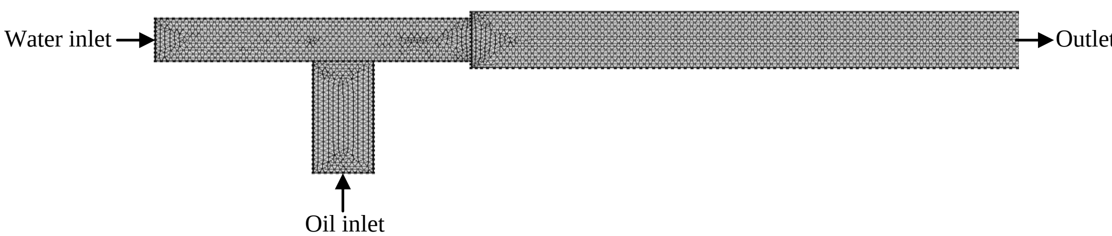


***Figure 2.2:*** *Mesh geometry of the entire domain of the pipe.*

## 2.2 Governing Equations and their Specifications

The governing equations solved in COMSOL Multiphysics are Navier-Stoke equations (2.1 and 2.2) for incompressible fluid flows and interface tracking algorithm which in this work is level set equation (2.3) following [1] and using surface tension coefficient $\sigma$ 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.

$$\rho(\phi)\left(\frac{\partial\bar{u}}{\partial t}+(\bar{u}\cdot\nabla)\bar{u}\right)=\nabla\cdot\left[-pI+\left(\mu(\phi)(\nabla\bar{u}+\nabla\bar{u}^T)\right)\right]+\rho(\phi)g+f_{st} \qquad (2.1)$$

$$\nabla\cdot\bar{u}=0 \qquad (2.2)$$

$$\frac{\partial\phi}{\partial t}+\bar{u}\cdot\nabla\phi=\gamma\nabla\cdot\left[\epsilon\nabla\phi-\phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right] \qquad (2.3)$$

$\rho$ is the density of the fluid, $\mu$ is the dynamic viscosity, $T$ is the temperature, $I$ is a diagonal unit matrix, $g$ is gravitational force, $\phi$ is a level set variable which denote the volume fraction of the fluids, $\epsilon$ isthe thickness of the interface, $\gamma$ is a re-initialization or stabilization parameter.
The force $f_{st}$ at the right side of the fully coupled Navier-Stoke equation with level setequation, is the internal force boundary condition between the two fluids (the surface tension force) and can be re-written as a volumetric force, which is derived in[6] as

$$f_{st}=\sigma\kappa\hat{n}\delta_\epsilon(\Gamma,x)(2.4)$$

where $\delta_\epsilon(\Gamma,x)$ is the Dirac delta function concentrating the surface tension force to the interface between the two fluids, $\Gamma$ is the interface, $\sigma$ is the surface tension coefficient, $\kappa$ is the interface curvature and $\hat{n}$ is the unit normal to the interface.

## 2.3 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 $\phi$ 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,$\phi_o$, is zero, which means that volume fraction of water,$\phi_w$, 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,$\phi_o$, 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 order to account for the interfaces of the two fluids and the solid wall of the pipe, wetted wall boundary condition was used. In level set method, wetted wall boundary condition sets the velocity component normal to the wall to zero, which is suitable for solid walls in contact with a fluid interface.
$\bar{u} \cdot \hat{n}_{wall} = 0$
and introduces a wall frictional boundary force of the form
$F_{fr} = -\frac{\mu}{\beta}\bar{u}$
where $\beta$ 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 2.3).
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

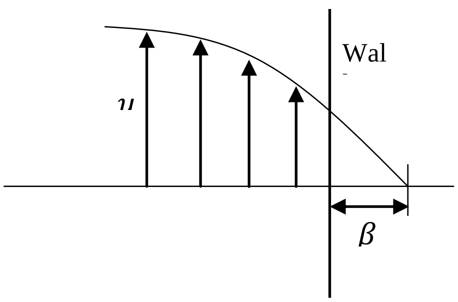


***Figure 2.3:*** *Description of slip length $\beta$.*

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

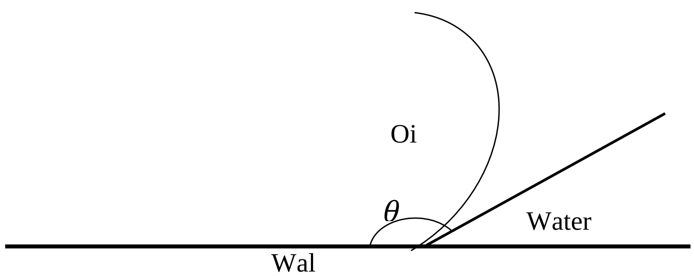


***Figure 2.4:*** *Description of contact angle $\theta$.*

where $\theta$ 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 contact angle $\theta$in radian and slip length $\beta$ in meters are needed. The best value of $\beta$ is the mesh element size$h$, and the value of $\theta$ used is 2.99406215 radian (171.55 degree).

### 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 3.1.

***Table 3.1: 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 |
| 0.13 | 0.023 | 9.81 | Slug flow (S) |

## 4.0 Results and Discussion

Simulations was successfully performed in two dimensional (2D) model, using COMSOL Multiphysics. The laminar two-phase flow, level set method used, successfully predicted slug flow pattern. In the simulation, inlet velocities of water and oil were selected from the velocities used in [4] in their experimental work, in which at low phase velocities (0.13m/s and 0.023m/s for water and oil respectively), as the flow develops, oil slugs appear with a unique liquid breakup between two consecutive slugs in the continuous water phase. This pattern is called slug flow as shown in figure 4.1

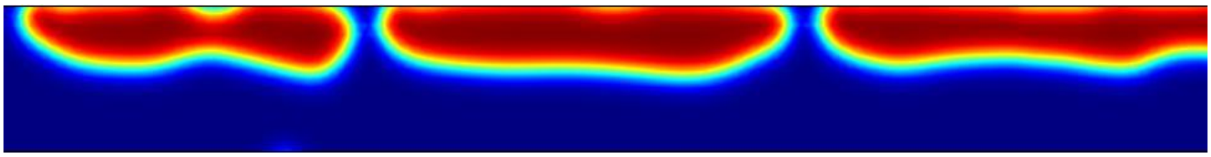

***Figure 4.1:*** *Slug flow.*

## 4.1 Volume Fraction

Volume fraction is one of the most important parameters used in classifying two-phase flow. It is the physical value that is used in determining other important parameter, such as the two-phase density and viscosity. In turn, the two-phase density and viscosity helps in obtaining the average velocity of the two-phases, which is very important 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 buoyance effect with respect to gravitational force, in agreement of oil being less dense than water.

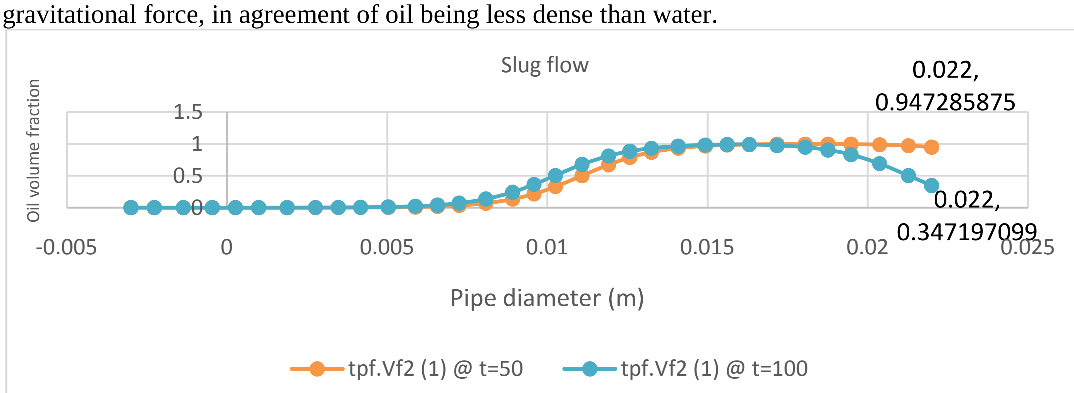


***Figure 4.2:*** *Diameter distribution of oil volume fraction.*

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. The diameter distribution of the volume fraction strictly depends on the area it is plotted, because the flow breaks into slugs.

## 4.2 Pressure Profile

In two-phase flow, pressure is an important parameter, 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 order to balance the force acting on the fluids. Pressure decreases upward the pipe diameter as gravitational force act downward the pipe diameter.

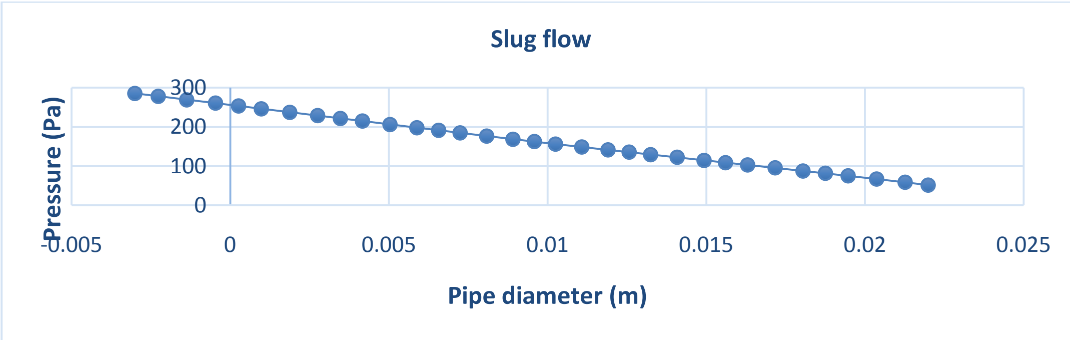


***Figure 4.3:*** *Pressure distribution.*

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 this type of flow pattern, velocity profile is distinguished into two, one at the point where there is slug and the other, between two slugs. Taking the profile as the flow develops (50 and 100 seconds), figure 4.4 shows that the velocity profile increases upward to the middle of the pipe and started decreasing slowly toward the upper side of the pipe, but as the flow develops to 100 seconds, velocity increased at the center of the pipe and had a sharp decrease at the interface between oil and water.

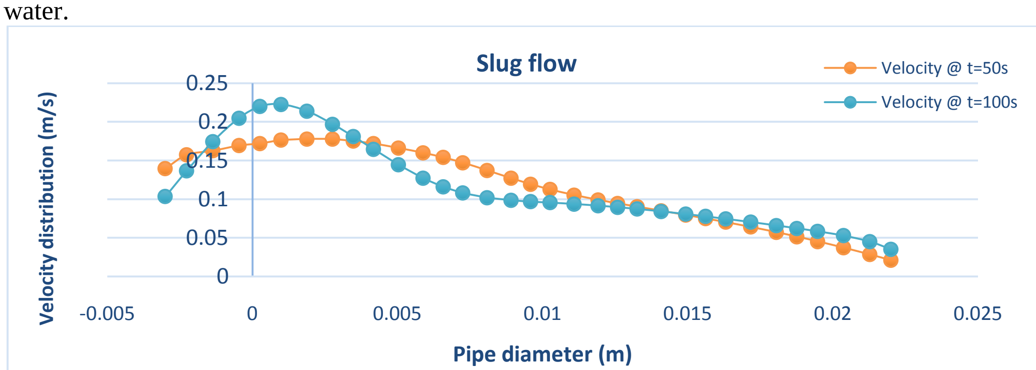


***Figure 4.4:*** *Velocity distribution.*

Figure 4.4 shows clearly the nature of slug flow. At the cut line of the pipe diameter, where the data used to plot the profile were collected, the profile shows that at 50 seconds, there were no slug passing across the line and at 100 seconds, a slug passed the line which produced the sharp decrease of velocity at the interface of oil and water.

## 4.4 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 was obtained for slug flow at 50 and 100 seconds in the upper and lower part of the pipe.

***Table 4.1:*** *Maximum values for Oil volume fraction, Pressure and Mixture velocity at upper and lower part of the pipe.*

**Slug flow**

| | Upper part of the Pipe | | Lower part of the Pipe | |
|---|---|---|---|---|
| Time (s) | 50 | 100 | 50 | 100 |
| Oil volume fraction | 0.99406 | 0.9875 | 0.01992 | 0.021 |
| Pressure (Pa) | 89.007 | 102.95557 | 276.19728 | 291.15753 |
| Mixture velocity (m/s) | 0.08211 | 0.0934 | 0.22395 | 0.22429 |

## 4.5 Validation

The simulated result (slug flow) were compared with slug flow of experimental results of [4]. Figure 4.5a and b shows the experimental result of slug flow and simulated result of slug fow respectively.

***Figure 4.5a:*** *Slug flow experimental work* [4].

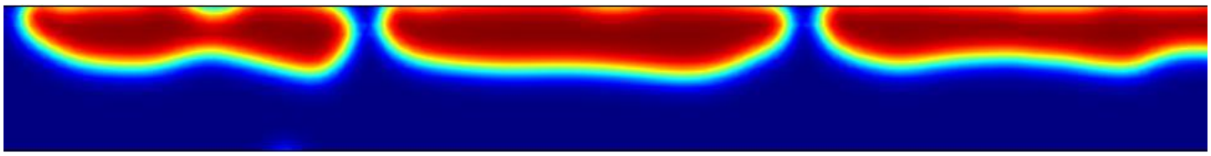

***Figure 4.5b:*** *Simulated results of slug flow with LS.*

The figures show that the simulated slug flow are in line with the experimental result of the slug flow pattern. The simulated result showed that the length of the oil slug increases with an increase in oil velocity and this is in agreement with [4] which was predicted in their experimental result.

In general, these comparisons gave an indication that laminar tow-phase flow, level set (LS) method is a good approach for simulating slug flow of oil and water in COMSOL Multiphysics.

## 5. Conclusion

Computational fluid dynamics (CFD) study has been conducted to predict flow pattern of light viscous oil-water flow through a horizontal pipe in a low velocity. 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 flow pattern. The flow pattern has been successfully predicted, using CFD package in FEM based software COMSOL Multiphysics 4.2a, in 2-dimensional geometry. 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 4.1).

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 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 two dense, which in turn helped in reducing the simulation time.

Finally, the flow pattern predicted was validated with experimental results of [5]. The validation showed a good agreement with the experimental results, which implied that, we have successfully shown that laminar two-phase flow level set method can be used to simulate 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] Asor, VE (2014): Effect of Small Perturbation on Propagation of Gravity Waves.*J. Nig. Ass. Math. Phys*,**28**, No. 1, pp 89 – 94

[2] Xiao-Xuan, X. (2007). Study on Oil-Water Two-Phase Flow in Horizontal Pipelines. *Journal of Petroleum Science and Engineering,* **59**, 43-58

[3] 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.

[4] 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.

[5] 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.

[6] Smolianski, A. (2001). *Numerical Modeling of Two-Fluid Interfacial Flows.* University of Jyvaskyla.