ABUAD Journal of Engineering Research and Development (AJERD)
ISSN: 2645-2685

Volume 4, Issue 1, 84-97

Numerical Investigation of the Phase Distribution of Dispersed Water in Oil Flow in a Pipe with Valve

Bildad DakwaL LENGS[1], Nicholas Sylvester GUKOP[1], Adesanmi ADEGOKE[1]

[1]*Mechanical Engineering Department, University of Jos, Plateau State, Nigeria*
lengsb@unijos.edu.ng/gukopn@unijos.edu.ng/adesanmia@unijos.edu.ng

Corresponding Author: lengsb@unijos.edu.ng, +2347034231355

Date Submitted: 25/03/2021
Date Accepted: 04/05/2021
*Date Published:*30/06/2021

Abstract: *In this work, a multiphase flow of water in oil in a pipeline fitted with a valve was studied using numerical analysis method. Grid was generated in ICEM software for a pipe of 0.1 m interior diameter and length 12 m and simulation was done in Ansys FLUENT 12.1 computational environment. The standard k-epsilon Reynolds Averaged Navier-Stokes model was used to solve the turbulence in the flow. A mixture of oil of density 825 kg/m^3 and viscosity 2 mPa s; and water of density 998.2 kg/m^3 and viscosity 1.003 mPa s was studied for mixture velocities of 0.6 -3.0 m/s and water input volume fraction 20%. The results revealed that water wet the lower surface of the pipe and oil wet the upper surface of the pipe at mixture velocity of 0.6 m/s at positions 3 m from the valve centre pivot. As the mixture velocity is increased, the amount of water in contact with the pipe surface decreases. Around the valve the water hold-up increases as the flow approaches the valve at position 0.05 m before the valve, but drastically reduced at position 0.05 m after the valve centre. The results led to the conclusion that the presence of a valve in an oil-water flow affect the phase distribution of the water and the oil in a way that increases the pipe susceptibility to corrosion; just before the valve as a result of water in contact with the pipe, and after the pipe as a result of erosion from high velocity.*

Keywords: *Multiphase flow, Turbulence, Valve, Water cut, Water-Oil flow.*

1. INTRODUCTION

In the process industries, pipelines are used for the various fluid transfers, an example is the petroleum industry where oil and gas are extracted from sedimentary rocks and refined into various finished products. Pipes are used extensively in the petroleum industry from the exploration of oil and gas through the conveying of the same to the refinery to the selling of the final product at the fuel station. Petroleum exploitation involves drilling through sedimentary rocks to reach oil and gas reservoir, after which oil and/or gas are conveyed to the outside through a network of pipes (Jürgen, 2015; Dake, 1998). The petroleum conveyed from oil wells contain some fraction of water in it, a new oil well produces negligible amount of water which is transported as a dispersed phase in the oil, but as the well get older the volume fraction of water increases reaching as much as 90% in some wells at their economic limits when the wells are abandoned (Bratland, 2010; Angeli and Hewitt, 2000). Pipeline transportation is also recommended for conveying the produced offshore oil-water petroleum from the production site to the place of separation and further processing through, mostly, horizontal and slightly inclined pipes in areas with gentle slope.

The major challenge of pipeline transportation is the structural integrity of the pipe material, the main pipe material used is steel which has the strength and resilience needed for the conveying of the bulk petroleum product at high pressures. As strong as steel is, it is an active metal that is easily affected by environmental conditions; it gets corroded in the present of air and moisture. Although oil does not react easily with steel, any contact with water should be avoided, if possible, because it enhances the rusting of the steel material thereby reducing its strength. The water content of the oil is also a very good solvent which carries some inorganic gases like carbon dioxide (CO_2) and hydrogen disulphide (H_2S) which have been proved to promote the corrosion of steel (Chong et al., 2006; Francois et al, 2008), a good pipeline design will be to keep the water emulsify and entrained in the oil and avoid any condition that will promote the separation of the water and the oil, which allow the water to wet the pipe. Bratland (2010) stated that increases in temperature, pressure and velocity lead to erosion, which is a very good starting point for corrosion.

Fluid flow transportation is made complete by the installation of flow rate and pressure control devices along the pipe networks to maintain the desired flow condition and amount planned for. Valves are mostly used as the flow rate and

 ISSN: 2645-2685

pressure control devices in the pipelines fluid transportation; these are devices which are placed along flow path which obstruct the flow of fluid to a varying extent depending on the degree of their closure or opening. They tend to change the flow Physics at the region where they are installed thus generating a different flow behaviour around their zone of influence (Spedding et al, 2004), this may include separation of the water from the oil which should not have being, in the absent of any obstacle in the flow path. The opening and closing of valves can produce pressure surge which lead to cavitations as bubbles or drops collapsed, this causes small pieces of the pipe to be removed which reduces the strength of the pipe, and pipe failure if not check. The study of fluid flow around valves will provide additional information about the flow; this is imperative for the proper design of the fluid flow in the pipe Yuan and Li (2003).

This research work seeks to study the flow of oil-water in a pipe with a valve installed to understand the changes in the flow Physics especially flow separations, as a result of the presence of the valve, and how to use the information obtained for a better pipeline design for the petroleum transportation of oil-water multiphase fluid.

2. GOVERNING EQUATIONS

2.1 Basic Terminology used in Multiphase flow

The definitions of the fundamental terms associated with multiphase flows that will be used in this research are made in this section. Since we will be dealing with two phase flow of oil-water, with water as the dispersed phase and oil as the carrier/continuous phase our terminologies will be limited to such flow.

Volume fraction is the proportion of space occupied by a given phase (Brennen, 2005), the volume fractions for the dispersed (α_d) and carrier phases (α_c) are given as follows:

$$\alpha_d = \lim_{\delta V \to V^0} \frac{\delta V_d}{\delta V} \tag{1}$$

$$\alpha_c = \lim_{\delta V \to V^0} \frac{\delta V_c}{\delta V} \tag{2}$$

Where δV_d and δV_c are the volumes of the dispersed and the continuous/carrier phases respectively in volume δV, the limiting volume to ensures stationary average is δV^0 (Crowe et al., 1998). The volume fraction is also referred to as void fraction and holdup in some literatures, especially for the dispersed phase (Bratland, 2010). These definitions implied that the sum of the volume fractions is unity.

$$\alpha_d + \alpha_c = 1 \tag{3}$$

The bulk/apparent/effective density defined as the mass of a phase per unit volume of mixture is given as

$$\hat{\rho}_d = \lim_{\delta V \to V^0} \frac{\delta M_d}{\delta V} \tag{4}$$

This is related to the material/physical density as

$$\hat{\rho}_d = \alpha_d \rho_d \tag{5}$$

where δM_d and ρ_d are the mass and material density of the dispersed phase, the same goes for the continuous phase. The mixture density can be obtained as the sum of the bulk densities of the phases.

$$\hat{\rho}_d + \hat{\rho}_c = \rho_m \tag{6}$$

The superficial velocity is defined as the mass flow rate of a phase divided by the phase material density and the cross section area of the pipe; this is the velocity of a phase assuming it occupied the whole pipe area. For the dispersed phase, the superficial velocity ($\hat{u}_d$) is

$$\hat{u}_d = \frac{\dot{M}_d}{\rho_d A} \tag{7}$$

where A is the pipe area. This can be related to the actual velocity of a phase, called the phase velocity (u_d) by the volume fraction.

$$\hat{u}_d = \alpha_d u_d \tag{8}$$

The same equations hold for the continuous phase.

The mixture velocity is obtained as the summation of the superficial velocities of the various phases as given below:

$$\hat{u}_m = \alpha_d u_d + \alpha_c u_c \tag{9}$$

The time taken for a particle or droplet to respond to a change in the flow velocity or temperature determined the relevance of the coupling parameters used in the flow. For a spherical particle or droplet of diameter, d; the momentum (τ_v) and the thermal (τ_T) response times, which correspond to the times required for the particle to respond to change in velocity and temperature respectively, are given as

$$\tau_v = \frac{\rho_d d^2}{18\mu_c} \tag{10}$$

$$\tau_T = \frac{\rho_d c_d d^2}{12\kappa_c} \tag{11}$$

where κ, μ and c are the thermal conductivity, dynamic viscosity and specific heat capacity of the phases respectively.
Multiphase flow can be defined as dispersed flow if the particle/droplet motion is control by fluid drag and lift, or dense flow for particle/droplets motion controlled by collision (FLUENT, 2009). If the ratio of the momentum time response to the time between collisions is greater than one, it is a dispersed flow; otherwise it is a dense flow. Coupling is achieved through the exchange of mass, momentum and energy between the phases involved in the flow.

2.2Eulerian Multiphase Model

The Eulerian multiphase model is used for the modelling of interacting multiphase flows (FLUENT, 2009).Two or more secondary phases can be model with this method with each treated as an interpenetrating continuum. This model solved a momentum and continuity equation for each of the phase with all of the phases sharing a common pressure. The different k-ϵ models can be used with this model in FLUENT. Ansys Fluent 12 was used for the simulation in this study, and there is not mass transfer between the oil-water phases, also the oil is the primary phase (c) and the water the secondary phase (d). The equations solved by Fluent for this case are as follows:

- Mass conservation for the phase d is obtained in the continuity equation as(FLUENT, 2009)

$$\frac{1}{\rho_d}\frac{\partial(\alpha_d \rho_d)}{\partial t} + \nabla \cdot (\alpha_d \rho_d u_d) = 0 \tag{12}$$

This equation was solved for the secondary phase to obtain the volume fraction at any position, after which equation 1.3 was solved to obtain the volume fraction of the primary phase.

- Momentum conservation for the phase d is obtained as(FLUENT, 2009)

$$\frac{\partial(\alpha_d \rho_d u_d)}{\partial t} + \nabla \cdot (\alpha_d \rho_d u_d u_d) = -\alpha_d \nabla P + \nabla \cdot \bar{\tau}_d + \alpha_d \rho_d g + K_{cd}(u_c - u_d) + F_d + F_{lift,d} + F_{vm,d} \tag{13}$$

The strain-stress tensor for the phase d is given as(FLUENT, 2009)

$$\bar{\tau}_d = \alpha_d \mu_d \left(\nabla u_d + \nabla u_d{}^T\right) + \alpha_d \left(\lambda_d - \frac{2}{3}\mu_d\right)\nabla \cdot u_d \bar{I} \tag{14}$$

The virtual mass force for phase d is given by(FLUENT, 2009)

$$F_{vm,d} = 0.5\alpha_c \rho_d \left(\frac{d_d u_d}{dt} - \frac{d_c u_c}{dt}\right) \tag{15}$$

The lift force is given by

$$F_{lift,d} = 0.5\alpha_d \rho_c (u_d - u_c) \times (\nabla \times u_d) \tag{16}$$

The interphase exchange coefficient is model by

$$K_{cd} = \frac{\alpha_d \alpha_c \rho_c f}{\tau_v} \tag{17}$$

The drag function can be obtained as follows; as

$$f = \frac{C_D \,\mathrm{Re}}{24} \tag{18}$$

Where C_D, the drag coefficient depends on the relative Reynolds number Re between the phases defined as

$$\mathrm{Re} = \frac{\rho_c \left| u_d - u_c \right| d_d}{\mu_c} \tag{19}$$

There are three different models in Fluent used to obtain the drag coefficient for fluid-fluid flow; these include: the Schiller and Naumann model, the default in FLUENT, it is recommended for general fluid-fluid flows because it is stable and produced results that are accurate enough for any engineering application for such flows; the Morsi and Alexander model, the most complete model but less stable than the others, and the symmetric model which is recommended for flows in which the secondary (dispersed) phase can become continuous in some parts of the flow. The model of Morsi and Alexander was used to calculate the drag coefficient.

2.3K-€ Multiphase Turbulence Modelling

The standard k-ε turbulence model was used for the simulation as stated earlier. ANSYS FLUENT have three options for this model; these include: the mixture turbulence model, the dispersed turbulence model and the turbulence modelling per phase.

The mixture turbulence model determines turbulence by the used of the mixture properties and parameters of the different phases; this model is suitable for stratified or nearly stratified flows with density ratio close to 1.

Dispersed turbulence model solved transport equations of the continuous phase for the turbulence kinetic energy (k) and the dissipation rate (ε) as in the standard k-ε model with additional terms to define the momentum transfer between the phases. The mean characteristics of the continuous phase and the ratio of the momentum time response to the interacting time between turbulent drops are used to determine the turbulence in the dispersed phase. This model is used for multiphase problem where collisions between the drops of the secondary phase is negligible, that is the flow is not dense.

Turbulence modelling per phase solves for each phase a transport equation for k and ε. This is the most accurate of the three models and is used for flows where turbulence exchange is dominant between the phases. The turbulence model for each phase will be used for this study since we are interested in the flow around the valve; much instability is expected in that region, this require the consideration of turbulence in each of the phases to predict the flow accurately. Transport equations (20 - 21) for the turbulent kinetic energy (k) and the turbulent dissipation rate (ε) are solved for each of the phase using phase weighted velocity U, as given below: The turbulent kinetic energy transport equation for the dispersed phase d is

$$\frac{\partial(\alpha_d \rho_d k_d)}{\partial t} + \nabla\cdot(\alpha_d \rho_d U_d k_d) = \nabla\cdot\left(\alpha_d\left(\mu_d + \frac{\mu_{t,d}}{\sigma_k}\right)\nabla k_d\right) + (\alpha_d G_{k,d} - \alpha_d \rho_d \varepsilon_d) + K_{cd}(C_{cd}k_c - C_{dc}k_d)$$
$$+ K_{cd}(U_d - U_c)\cdot\frac{\mu_{t,c}}{\alpha_c \sigma_c}\nabla\alpha_c - K_{cd}(U_d - U_c)\cdot\frac{\mu_{t,d}}{\alpha_d \sigma_d}\nabla\alpha_d \tag{20}$$

The turbulent rate of dissipation transport equation for the dispersed phase is

$$\frac{\partial(\alpha_d \rho_d \varepsilon_d)}{\partial t} + \nabla\cdot(\alpha_d \rho_d U_d \varepsilon_d) = \nabla\cdot\left(\alpha_d\left(\frac{\mu_{t,d}}{\sigma_\varepsilon}\right)\nabla\varepsilon_d\right) + \frac{\varepsilon_d}{k_d}\Big[C_{1\varepsilon}\alpha_d G_{k,d} - C_{2\varepsilon}\alpha_d \rho_d \varepsilon_d +$$
$$C_{3\varepsilon}\left(K_{cd}(C_{dc}k_c - C_{cd}k_d) - K_{cd}(U_c - U_d)\cdot\frac{\mu_{t,c}}{\alpha_c\sigma_c}\nabla\alpha_c + K_{cd}(U_c - U_d)\cdot\frac{\mu_{t,d}}{\alpha_d\sigma_d}\nabla\alpha_d\right)\Big] \tag{21}$$

The turbulent kinetic energy generation for the phase d is given as

$$G_{k,d} = \mu_{t,d}\left(\nabla U_d + \nabla U_d^{\,T}\right)\cdot u_d \tag{22}$$

The Reynolds stress tensor is given as

$$\bar{\tau}'_d = \frac{3}{2}\left(\rho_d k_d + \rho_d \nu_{t,d} \nabla \cdot U_d\right)\bar{I} + \rho_d \nu_{t,d}\left(\nabla U_d + \nabla U_d{}^T\right) \tag{22}$$

The turbulent viscosity is determined as

$$\mu_{t,d} = \frac{C_\mu \rho_d k_d^2}{\varepsilon_d} \tag{24}$$

The turbulent drag term is determined as

$$K_{cd}\left(u_d - u_c\right) = K_{cd}\left(U_d - U_c\right) - K_{cd} u_{dr} \tag{25}$$

The drift velocity which results from volume fraction fluctuation is given as

$$u_{dr} = \left(\frac{\overline{\alpha_d u'_d}}{\alpha_d} - \frac{\overline{\alpha_c u'_c}}{\alpha_c}\right) = \left(\frac{D_d}{\alpha_d \sigma_{dc}} \nabla \alpha_d - \frac{D_c}{\alpha_c \sigma_{dc}} \nabla \alpha_c\right) \tag{26}$$

The eddies energy characteristics time is defined as

$$\tau_{t,d} = \frac{3 k_d C_\mu}{2 \varepsilon_d} \tag{27}$$

The turbulent characteristics length is given as

$$L_{t,c} = \sqrt{\frac{3}{2}} \frac{k_d^{3/2} C_\mu}{\varepsilon_d} \tag{28}$$

The characteristics drops momentum relaxation time is defined as

$$\tau_{F,cd} = \alpha_d \rho_c K_{cd}^{-1}\left(\frac{\rho_d}{\rho_c} + C_V\right) \tag{29}$$

The Lagrangian integral time scale is defined as

$$\tau_{t,cd} = \frac{\tau_{t,d}}{\sqrt{1 + C_\beta \xi^2}} \tag{30}$$

$$\xi = \frac{\left|u_{cd}\right| \tau_{t,d}}{L_{t,c}}$$

$$C_\beta = 1.8 - 1.35 \cos^2 \theta \tag{31}$$

Where Θ is the angle between the phase velocity and the relative velocity
The ratio of the time scales is given as

The term C_{cd} is obtained as

 ISSN: 2645-2685

$$C_{cd} = 2\left(\frac{\eta_{dc}}{1+\eta_{dc}}\right) \tag{32}$$

3. VALIDATION AND GRID CONVERGENCE

3.1 Water Flow in a Pipe with a Valve

A simplified geometry(fig.1) consisting of a pipe of diameter 400 mm and length 3500 mm with upstream of 900 mm and downstream of 2600 mm, valve was model as a dummy cylinder of diameter 400 mm and thickness 2 mm. An unstructured mesh was generated in ICEM and adjusted appropriately according to the valve lift used as shown in fig.1, the characteristics of the grids used are summaries as shown in Table 1 below. Four valves lift positions were used, these are 16.67% (15°), 33.33% (30°), 50 (45°) and 100% (90°); from nearly closed position to completely open position. A fix differential pressure of 1 psi (6895 Pa) was used between the upstream and downstream; SIMPLEC for the pressure correction, and second order for solving the equations.

Table 1: Grid characteristics for the validation case

Valve Lift (%)	No. of Cells	First Height(mm)	Y-Plus
16.67	572917	1.50	6.52
33.33	753058	0.50	10.96
50.00	759573	0.50	17.10
100.00	771296	0.50	22.28

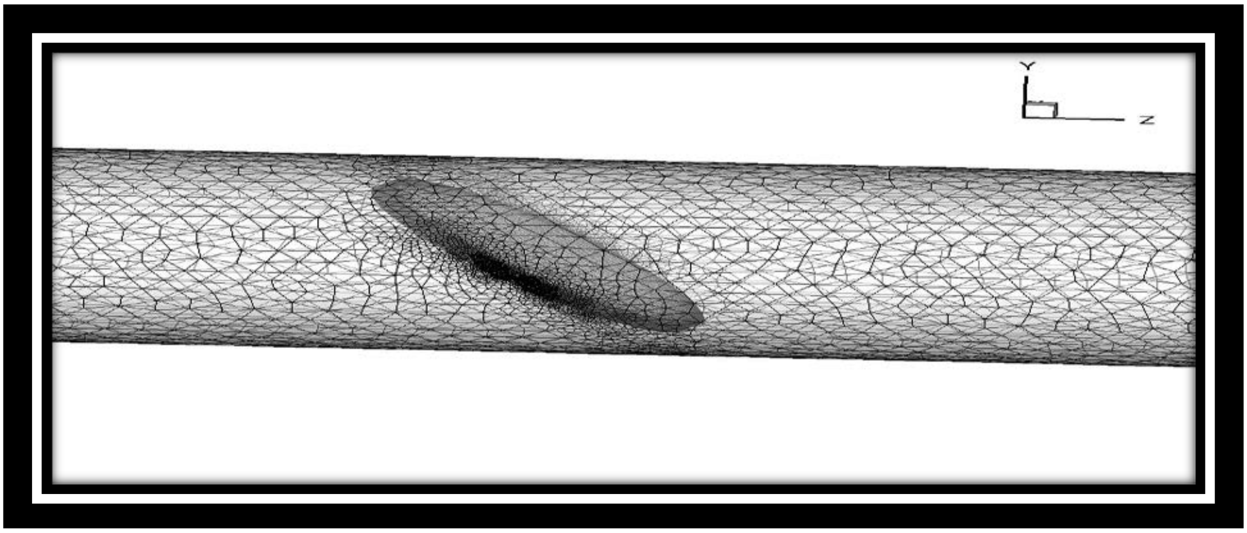


Figure 1: Grid generated for valve lift of 50%

A differential pressure of 6895 KPa was used, the valve flow coefficient given as a percentage ratio of the maximum valve flow coefficient (Valve flow coefficient at valve lift of 100%), and the valve loss coefficients calculated using equation 33. The valve flow coefficient was observed to increase with the valve lift, this show that the more the opening of the valve is, the more it allow flow to pass without much interruption. The valve loss coefficients are found to decrease with increased in the valve opening angle.

This pattern shows that closing a valve causes much pressure drop in a fluid flow in a multiple ratio, as it is observed not to be linear with the valve lift. Figure 2 shows the plot of the valve flow and loss coefficients against the valve lift. Figure 2a shows valve flow coefficients increasing with valve lift and fig. 2b shows valve loss coefficients decreasing with increased in valve lift.

(a) (b)

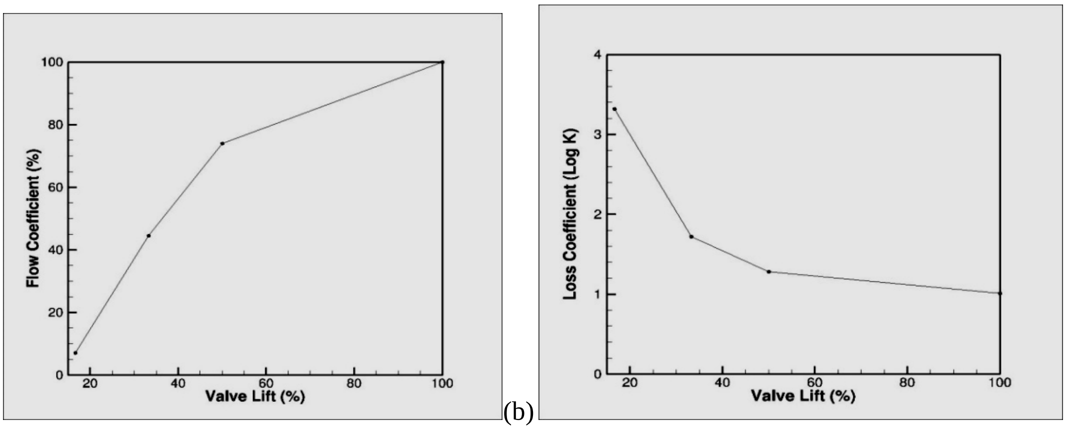


Figure 2: Flow characteristic (a) valve flow coefficient (b) valve loss coefficient

3.2 Grid Convergence Study

Numerical solution is an approximation of an exact equation in which an infinite space step is made finite to solve a given problem. It is then expected that the smaller the space step used the more accurate the solution should be, taking smaller step size mean using more cells for the computation. In this section a grid convergence study was done for the pipe to be used for the oil-water multiphase problem later, using single water flow.

A pipe of diameter 100 mm and length 12000 mm with a dummy cylinder of diameter 100 mm and thickness 5 mm at 6000 mm to serve as the valve. For the convergence study, a valve lift of 50% will be used with three different grids level, each higher level of approximately twice the previous grid as shown in table 2 below. Water of density (ρ) 998.2 Kg/m^3 and dynamic viscosity (μ) of 1.003mPa swith mean velocity (u) of 1.2 m/s was used for the three cases to be considered. The Reynolds number (Re) for the flow is 119426. The skin-friction/wall shear velocity (u_τ) and y-plus (y^+) are obtained from the following relationships:

$$K = \frac{\Delta P}{0.5\rho u^2} \tag{33}$$

$$u_\tau = u\sqrt{\frac{C_f}{2}} \tag{34}$$

$$C_f \approx 0.078Re^{-0.25} \tag{35}$$

$$y = \frac{\mu y^+}{\rho u_\tau} \tag{36}$$

where C_f is the fanning friction coefficient.

The grids characteristics used for the convergence study are as shown in Table 2, which include the number of elements, initial height used to capture the boundary layer flow around the wall and the resulting y-plus produced.

Table 2: Grid characteristics for convergence study

Grid	No. Of Cells	First Height(mm)	Y-Plus
Coarse	931543	0.1	5.47
Fine	2288783	0.1	5.47
Finest	3665272	0.1	5.47

The differential pressure between the inlet and the outlet of the pipe for the three grids were obtained and tabulated as shown in table 3, the mass flow rate for the three grids computed to three decimal places were the same. The differences in differential pressures were evaluated and tabulated as deviation (%), it revealed an increase of 0.198% in differential pressure from the coarse grid to the fine grid and an increase of 0.144% from fine grid to finest grid.

Table 3:Differential pressure and mass flux obtained for various grid refinements

Grid	**Mass Flux/ (Kg/s)**	**Diff. Press./ (Pa)**	**Deviation/ (%)**
Coarse	9.402	14569	-
Fine	9.402	14598	0.20
Finest	9.402	14619	0.14

4. OIL-WATER FLOW IN A PIPE WITH VALVE

4.1 Numerical Experiment Setup and Methodology

The study involves the numerical investigation of the two-phase flow of oil and water in a pipeline fitted with a valve. The following procedure was followed:

- The pipe geometry (fig.1) was constructed in GAMBIT with the valve (simple cylindrical plate) inside.
- Simulation was done in Fluent 12.1 for different valve openings, mixture velocities and input water volume fractions.
- The effect of the valve on the flow phase distributions (the water holdup) was investigated in the positions before and after the valve in the pipeline.
- Separations and recirculation around the valve was studied to understand how they affect the water volume fraction in the flow.

The fluids (oil and water) to be used; the flow parameter to be considered and the pipe dimensions for the study are as follows:

- The internal diameter and length of the pipe are 0.1 m and 12 m respectively.
- The density and dynamic viscosity of the oil used are 825 Kg/m^3 and 2 mPa s respectively.
- Water of density 998.2 Kg/m^3 and viscosity 1 mPa s will be used.
- The valve is a cylindrical disc of diameter 0.1 m and thickness 0.005 m, located at 6 m from the inlet side of the pipe.
- Valve opening position of 45° was considered.
- The input water volume fraction used is 20%
- Different mixture velocities from 0.5 m/s -3 m/s were used
- Mixture Reynolds number considered was in the range of 22166-154747
- The convergence criteria were set as $1x10^{-3}$ for continuity and $1x10^{-4}$ for the other parameters in line with Okhuahesogie (2010), and also ensuring a mass balance of less than 0.2%between the inlet and the outlet.

4.2 Numerical Results and Discussions

Three different cases were considered for the valve lift of 50% to enable us investigates the effect of the valve on the flow characteristics. Three different mixture velocities were considered for water input volume fractions of 0.2 as shown in Table 4. To investigate the changes introduced into the flow as a result of the valve condition, water hold up were compared at distances of 0.05 m and 3.0 m along the flow direction. The boundary conditions were set with the inlet (plane z=-6) as velocity inlet; the outlet (plane z=6) as pressure outlet; the pipe and valve surfaces as walls with gravity acting in the negative y direction.

Table 4: The different cases considered for valve lift of 50%

	Case 1	Case 2	Case 3
u_m	0.6	1.8	3.0
μ_m	1.8006	1.8006	1.8006
ρ_m	859.6	859.6	859.6
Re_m	28645	85935	143226

The water input volume fraction α_w is defined by equation 1.1 and the oil input volume fraction can be obtained by using equation 3, the actual and superficial phase velocities (u_w^s, u_o^s) can be obtained by using equation 8 and equation 9. The mixture density ρ_m was obtained from the water and oil densities by using equation 5 and equation 6. The mixture dynamic viscosity μ_m (mPa s) was obtained from the water and oil dynamic viscosities by using equation 37.

$$\mu_m = \alpha_w \mu_w + \alpha_o \mu_o \quad (37)$$

$$Re_m = \frac{\rho_w u_w^s D}{\mu_w} \quad (38)$$

$$Re_m = \frac{\rho_o u_o^s D}{\mu_o} \quad (39)$$

$$Re_m = \frac{\rho_m u_m D}{\mu_m} \quad (40)$$

Equations38, 39 and 40 were used to obtain the water, oil and mixture Reynolds numbers of the flow, D is the pipe diameter. Angeli and Hewitt (2000b) found that the drop size distribution in a steel pipe has a maximum diameter which is generally less than 0.1 x diameter of the pipe,also Chong et al (2006) conducted an experimental study on water wetting and CO_2 corrosion in oil-water flow and found the droplet diameter to be between 5 mm and 10 mm for water input volume fraction below 10% at a mixture velocity of 0.5 m/s. They reported that the droplet diameter decreases with increased in the mixture velocity, thus a drop diameter of 4 mm was used in this study.

At the mixture velocity of 0.6 m/s, the water hold-up profile at line (0 y -3) on the left and line (0 y 3) on the right- 3 m before and after the valve respectively shows water volume fraction of 0.0% at the uppermost part of the pipe and 100% at the lowermost part of the pipe. A plateau corresponding to the water input volume fraction seem to cover most part of the profile between the extremes at the top and the bottom as shown in fig. 3 and fig 4. This is in agreement with the experimental results of Chong et al (2006) and Francois et al (2008) obtained for a plain pipe. The effect of the gravitational force in the y-direction draw the water which is denser than the oil to the lower part of the pipe, this is possible because of the mixture velocity (0.6 m/s) which is low allowing gravity to develop to some good extent in the flow resulting in this sedimentation. The results also showed that the flow is not much affected at these positions by the valve- the profile are almost the same 3 m before and after the valve.As the mixture velocity increases, the gravitational force had lesser time to act on the flow, hence both oil and water are found around the pipe surface, though in different proportion.

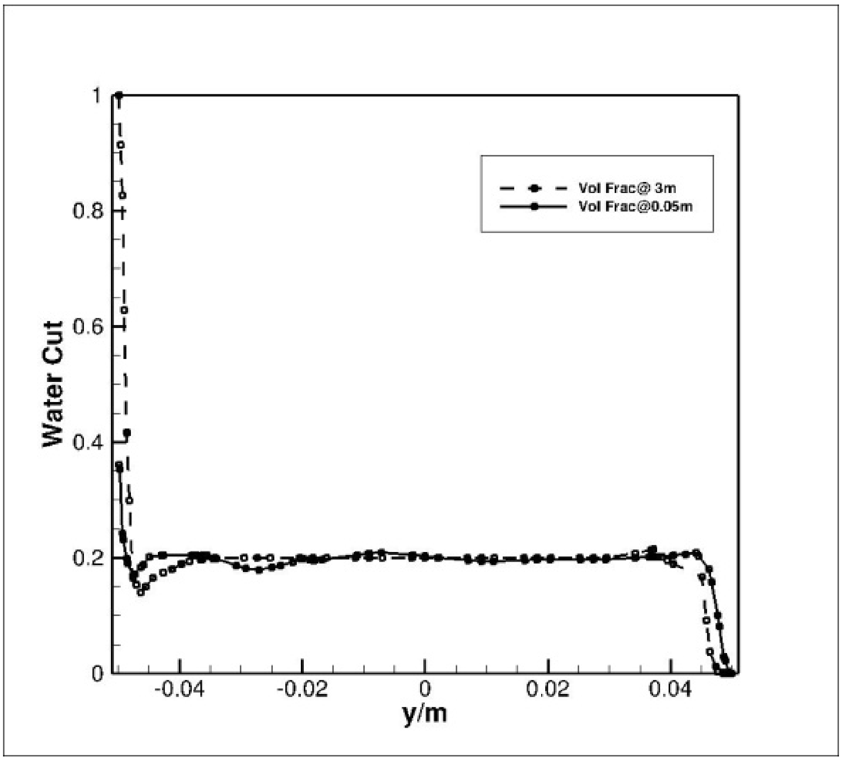


Figure 3: Comparison of water cut at distances of 0.05 m and 3.0 m along flow direction for mixture velocity of 0.6 m/s

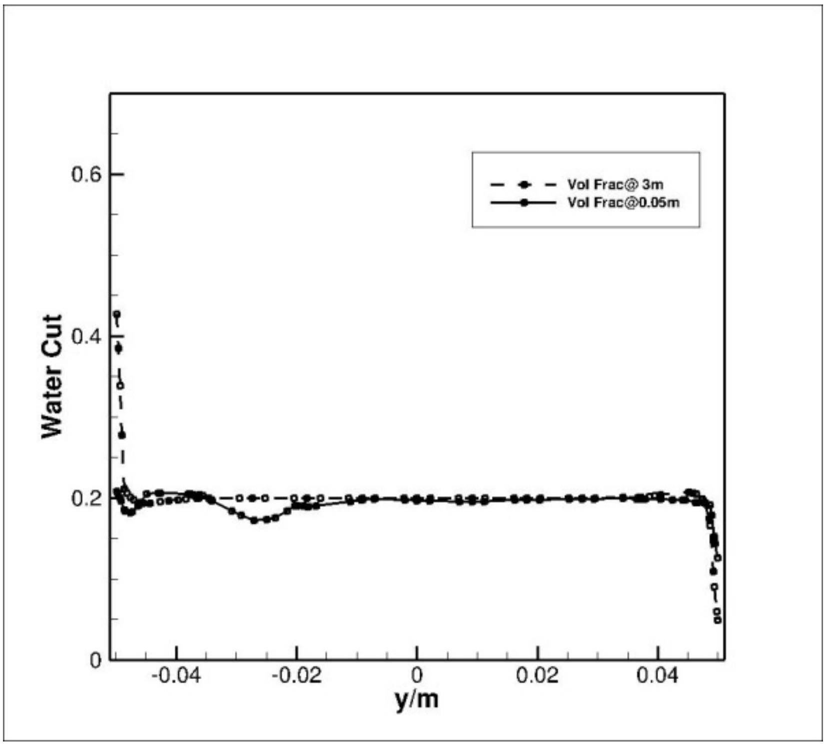


Figure 4: Comparison of water cut at distances of 0.05 m and 3.0 m along flow direction for mixture velocity of 1.8 m/s

The gravity effect was still evident in the flow at mixture velocity of 1.8 m/s. At mixture velocity of 3.0 m/s, the gravitational effect was also observed, with more water volume fraction at the bottom part of the pipe- about 50%, and less than 25% at the top of the pipe.

The action of the valve in the flow was investigated by comparing the water hold up at positions around the valve with the water hold up at positions far from the valve. The water hold-up for the three mixture velocities of 0.6 m/s, 1.8 m/s and 3.0 m/s were plotted as shown in fig 4, fig 5and fig 6 for the water input volume fractions in the pipe with the valve. The effect of gravity seem to follow the same trend for both positions considered, except that the water hold up at the distance of 0.05 m after the valve were observed to have considerately decreased at the lower part of the pipe compared to the water hold up at the distances of 3 m after the valve. For the mixture velocity of 0.6 m/s and water input volume fraction of 20% at the distances of 3 m and 0.05 m after the valve were 100% and 36% respectively. As the mixture velocity increases, lower maximum water hold up were observed for the two positions obtained as shown in fig 4and fig 5, these shows intermittent wetting of the pipe. The appearance of 100% water hold up at the bottom of the 50% valve lift pipe for the mixture velocity of 0.6 m/s is a very dangerous working condition for the pipe. Cai et al (2004), Cai et al (2005), Chong et al (2006) Tang et al (2007) and Francois et al (2008); all concluded that corrosion occurred in the pipe only when water is in contact with the pipe (that is, water wetting and intermittent wetting conditions), with the worst case coming from water wetting. If corrosion is to be avoided, oil-water should be transported in a pipe with any obstructive device in it, at higher mixture velocities to avoid the water wetting the pipe which enhances corrosion. The effect of the valve as a function of mixture velocity on the water input volume fraction was investigated. Profiles of the water hold up were obtained at distances of 0.05 m before and after the valve centre in the flow direction. Figure 6 revealed a slight increased in water volume fraction as the fluid approaches the valve at distance of 0.05 m in the flow direction before the valve and a drastic reduction in the water volume fraction as the fluid flow leaves the valve at a distance of 0.05 m in the flow direction after the valve at the bottom of the pipe. This was expected as the obstructive actions of the valve tend to bring fluid particles around the middle part of the valve to rest and accelerate those that pass through the constricted area between the valve and the pipe at the upper and lower part of the pipe.

The present of the valve in the flow reduces the water hold up at the bottom of the pipe at positions immediately after the valve. This can be explained as a result of the increased velocity at this region, as these velocities allow little time for the gravitational force to act at these regions to separate the water and the oil as happened in region away from the valve. This seem to be a good news for those looking for a way of stopping water wetting of the pipe, which means reduced corrosion, the other news is that the increase velocity is capable of increasing erosion in the pipe which is also a contributing factor to corrosion of the pipe material, Bratland (2009, 2010).

ISSN: 2645-2685

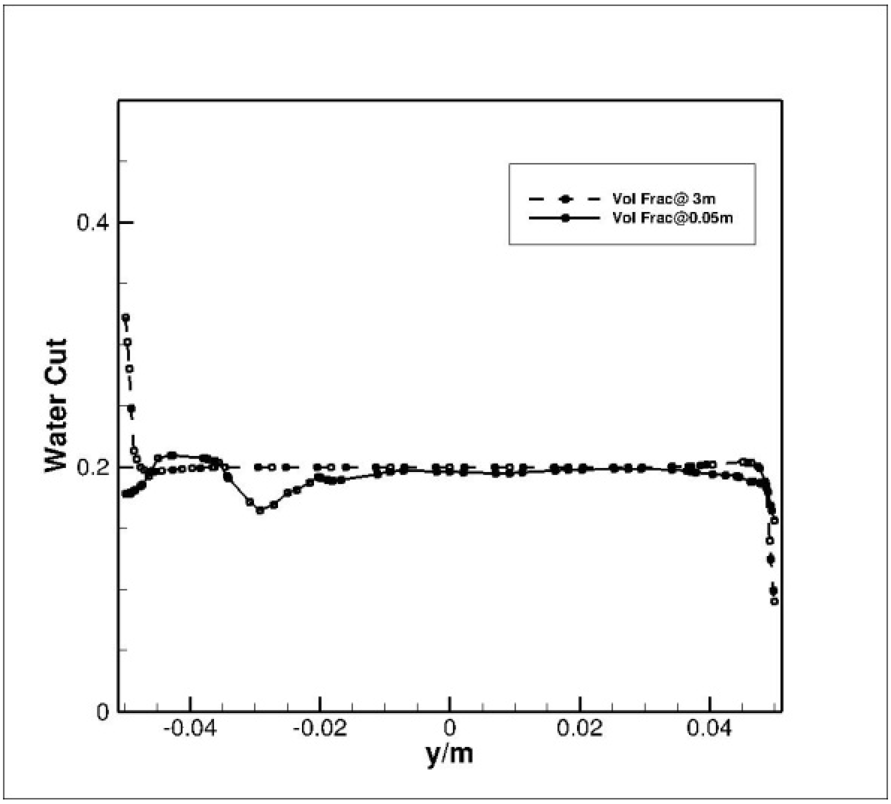
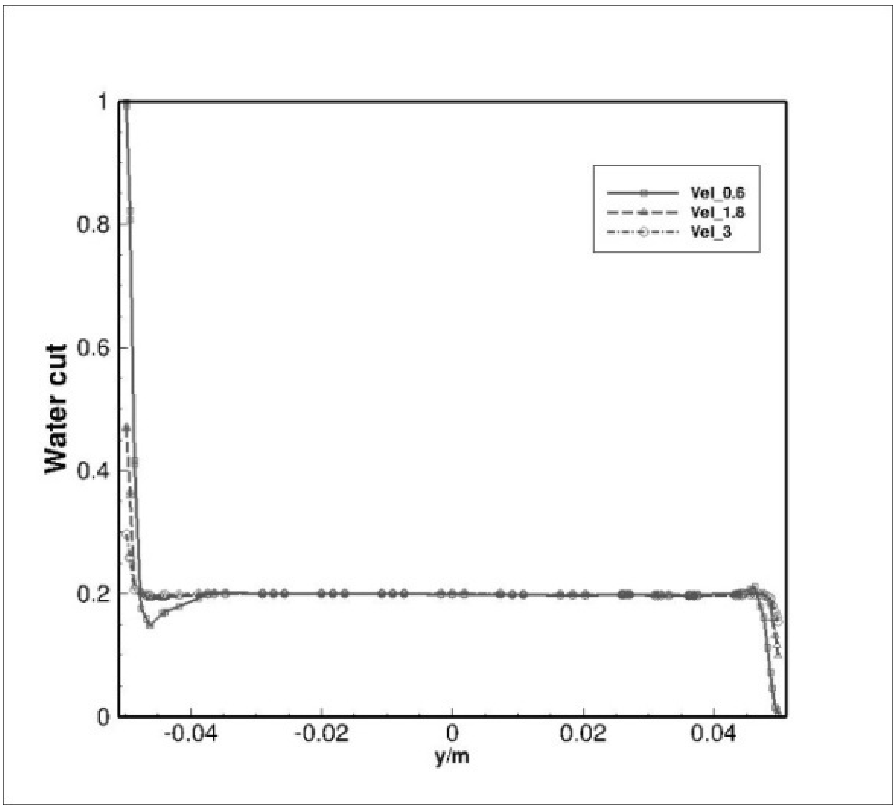
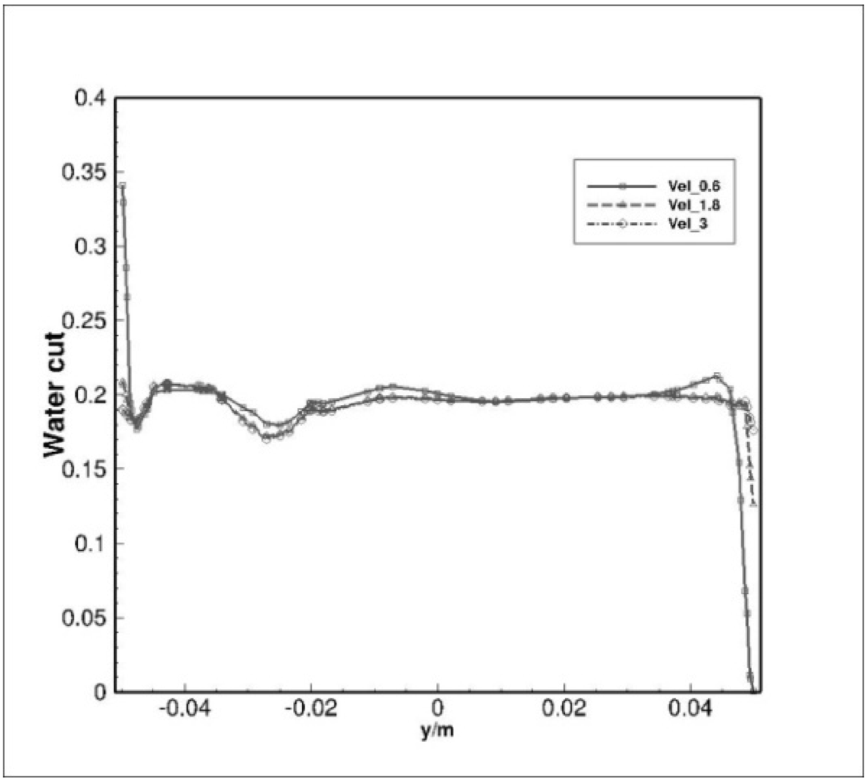


Figure 5: Comparison of water cut at distances of 0.05 m and 3.0 m along flow direction for mixture velocity of 3.0 m/s

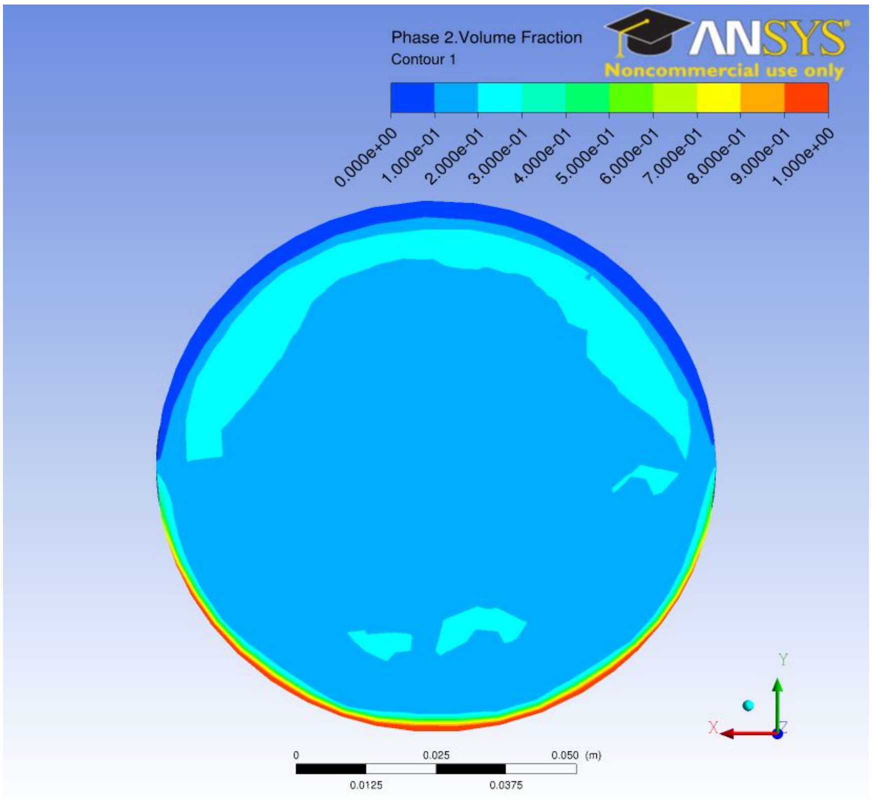


(a)

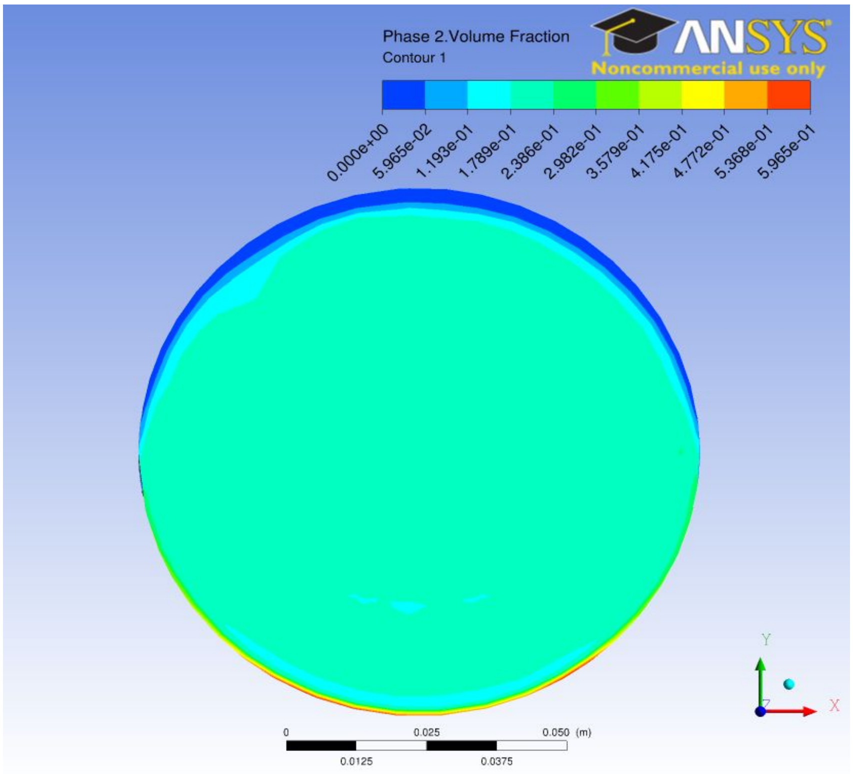


(b)

Figure 6: Volume fraction profiles for water input fraction of 0.2 at positions (a) 0.05 m before valve (0 0 0) (b) 0.05 m after valve (0 0 0)



(a)



(b)

Figure 7: Volume fractions contours for mixture velocity of 0.6 m/s at position 0.05m along flow direction (a) before valve centre (b) aftervalve centre.

5. CONCLUSION

Oil-Water flow in a pipe with a valve was investigated in this work in an attempt to understand the changes that occurred in the flow as a result of the obstructive action of the valve. A simple pipeline model of a cylindrical pipe of diameter 0.1 m and length 12 m and a dummy cylinder of diameter 0.1 m and thickness 5 mm were used for the pipe and the valve respectively. The computational method employed was validated using a single phase flow of water in a pipe with valve at different angles, flow characteristics parameters- valve loss coefficient and valve flow coefficient were calculated and plot against the valve lift.

The oil-water multiphase flow was solved for a pipe with a valve of 50% valve lift, nine cases were considered for water input volume fractions of 20% and mixture velocity of 0.6- 3.0 m/s. Around the pipe, at position (0 0 0.05) after the valve centre, a very low water volume fraction was observed in the 50% valve lift pipe compare to the volume fraction at a distance 3 m away from the valve centre in the flow direction. The general trend was that more water was observed to be in contact with the pipe lower surface at lower mixture velocities and higher water input volume fractions and vice versa.

Though valves are everywhere in pipelines system, there seem be little research work done on them. The facts obtained from this work revealed that the present of a valve in the flow is capable of not only changing the flow Physics, but also post danger on the pipe material- as it allow more water in contact with the pipe at region far from the valve and also caused acceleration of the flow around the valve which can increase erosion of the pipe surface, all these contribute to corrosion in the pipe.

REFERENCES

[1] Angeli, P. & Hewitt, G. F. (2000b). Drop Size Distributions in Horizontal Oil-water Dispersed Flows. *Chemical Engineering Science, Vol. 55, No. 16* , 3133-3143.
[2] Angeli, P. & Hewitt, G. F. (2000a). Flow Structure in Horizontal Oil-water Flow. *International Journal of Multiphase Flow* , 1183-1203.
[3] Bratland, O. (2010). Pipe Flow 2 : *Multi-Phase Flow Assurance. Oslo, Norway:System Pte. Ltd.*
[4] Bratland, O. (2009). Pipe Flow 1: *Single-Phase Flow Assurance.Oslo, Norway: System Pte. Ltd.*
[5] Brauner, N. (2003). *Modelling and control of two phase phenomena: Liquid-liquid two-phase flow systems.* Tel-Aviv: Tel-Aviv University.
[6] Brennen, C. E. (2005). *Fundamentals of Multiphase Flow.* California, United States: Cambridge University Press.
[7] Crowe, C. T. & et al. (2005). *Handbook of Multiphase Flow.*Bosa Roca, United States: CRC Press.
[8]Dake, L. (1998). *Fundamentals of Reservoir Engineering* . Amsterdam, The Netherlands : Elsevier .
[9] FLUENT. (2009). *ANSYS FLUENT 12.1 Theory Guide.*

[10] Francois, A., Li, C., Tang, X., Cai, J. & al, S. N. (2008). Determination of Phase Wetting in Oil-Water Pipe Flow. *CORROSION* (p. 08566). NACE International.

[11] Jürgen, S. (2015). *Basic Well Logging and Formation Evaluation.* Leoben Austria : Bookboon.com.

[12] Kumara, W. A., Halvorsen, B. M. & Melaaen, M. C. (2007). *Pressure Drop, Flow Pattern and Local Water Volume Fraction Measurements of Oil-Water Flow in Pipes.* Porsgrum: Telemark University College.

[13] Okhuahesogie, O. F. (2010). *Phase Separation of Oil-Water Flow in a Pipe Bend.* MSc. Thesis, Cranfield University.

[14] Rodriquez, O., & Oliemans, R. V. (2006). Experimental Study of Oil-water Flow in Horizontal and Inclined Pipes. *International Journal of Multiphase Flow 32* , 323-343.

[15] Soleiman, A., Lawrence, C. J. & Hewitt, G. F. (1999). Spatial Distribution of Oil and Water in Horizontal Pipe Flow. *Department of Chemical Engineering, Imperial College of Science, Technology and Medicine, London* .

[16] Spedding, P. L., Benard, E. & McNally, G. M. (2004). Fluid Flow Through 90 Degree Bends. *Dev. Chem. Eng. Mineral Process* , 107-128.

[17] Tang, X., Li, C., Ayello, F., Cai, J., Nesic, S. & al, e. (2007). Effect of Oil Type on Phase Wetting Transition and Corrosion in Oil-Water Flow. *CORROSION* (p. 07170). Houston: NACE International.

[18] Wang, L., Song, X. & Park, Y. (2009). The Improvement of Large Butterfly Valve by Using Numerical Analysis Method. *FLUID's 09.*

[19] Xu, X.-X. (2007). Study on Oil-water Two-Phase in Horizontal Pipelines. *Journal of Petroleum Science and Engineering, Vol. 59, No. 1-2* , 43-58.

[20] Yang, L., & Azzopardi, B. J. (2007). Phase Split of Liquid-Liquid Two-Phase Flow at a Horizontal T-Junction. *International Journal of Multiphase Flow, 33* (2), 207-216.

[21] Yuan, Q. & Li, P. Y. (2003). Modelling and Experimental Study of Flow Forces for Unstable Valve Design. *IMECE* (p. 42924). Washington D. C.: ASM