ISSN 2229-5518

MULTIPHASE FLOW SIMULATION OF OIL AND GAS IN VERTICAL FLOW USING CFX-PRE AND FLUENT.

Ajoko, Tolumoye John[1] and Ebughni, Nangi Petro[2]
[1]Department of Mechanical/Marine Engineering, Faculty of Engineering.
Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria.
Email: johntolumoye@yahoo.co.uk
[2]Department of Mechanical Engineering, Faculty of Engineering.
Rivers State University of Science and Technology, Nkpolu – Oroworukwo, Port-Harcourt Rivers State, Nigeria.
Email: petrosnangi@gmail.com

Abstract – The complexity to analyze corrective mathematical principles to predict the characteristics of multiphase flow in pipes in the petroleum industry is a key target due to its significance. Hence, the current research on multiphase flow simulation of oil and gas in vertical flow using CFX-PRE and Ansys Fluent is focused on the identification and study of the flow-regime map; mostly on the pressure distribution across the pipe and the effect of each phase on the wall of the pipe. The methodology is based on the use of simulation tools such as Solidworks which was used to model the pipe and to do initial flow analysis and Ansys CFX-pre for preparation of the model in the computational domain. Reported results for the characteristics of multiphase like pressure drop across vertical pipes were proffered solution. Also results confirmed change in phase either by heat addition or exchange of heat between phases as the prime cause of wavy motions in fluid transportation of difference in velocities of gas and liquid bubbles along pipes. Therefore, the simulation tools employed for the research study is considered as an effective and reliable technology.

Index Terms— Ansys Fluent, Iteration, Meshing, Modeling, Multiphase flow, Pressure drop, Solidworks

1. INTRODUCTION

Multiphase flow in pipes for the petroleum engineering industry is of great importance due to its relevance for the separation and transportation of mixture of fluids. Fluid mixtures such as light and heavy oil components, solid particles, hydrates, wax, etc are produced and transported through different cylindrical vessels. Thus, in multiphase flow process along pipes; it is most predictable to have sudden changes in phase of relative volumetric fraction which is usually as an effect of heat addition or exchange of heat between the phases or flashing due to depressurization [1].

Obviously in recent times, studies have been carried out to investigate multiphase flow in pipes. A reviewed literature reported that flow regime like the slug flow is applicable of transporting hydrocarbon fluids in the oil and gas industry through pipes allows gas bubbles flowing alternatively with liquid slugs at randomly fluctuating frequency is undesirable because it is capable of causing severe adverse effects [2]. Hence, at this note the flow rate

- *Ajoko, Tolumoye John is a lecturer in the department of Mechanical/Marine Engineering, Niger Delta University, Bayelsa State – Nigeria.*
- *Ebughni, Nangi Petro is a lecturer/Master student at Rivers State University of Science and Technology, Nkpolu – Oroworukwo, Port-Harcourt Rivers State, Nigeria*

ISSN 2229-5518

of the fluid in the equipment with highly – unsteady loading on the piping system is liable of widely fluctuation and processing equipment causing catastrophic failure such as pressure – drop, liquid hold – up in pipes, etc; due to metal fatigue and also with respect to the complexity of the two phases (oil and gas), the separated flow, dispersed flow and intermittent flow will depend on the pipe diameter, inclination angle, etc [3], [4].

Another contribution from a researcher affirmed that flow regimes (flow pattern) exist when multiphase flow topology co-currently acquire varieties of characteristic distributions [5]. Meanwhile, the optimization of the design and successful operation of multiphase well systems requires a substantial knowledge of the behavior characteristics of such flows. Various studies have confirmed the difficulties of corrective prediction on the characteristics of multiphase flow in wells where gas-liquid flow are involved because at present there is no satisfactorily mathematical principles to validate the concept based on the fact that the distribution of multiphase is normally unknown and difficult to specify quantitatively [6]. Nevertheless, an alternative numerical model used for the estimation of flow characteristics along vertical pipes in multiphase flow was established by some researchers. According to their work, simple numerical methods were proposed in order to prescribe flow variables along the piping for vertical upward flow; thereby providing correlation for the slip velocity between liquid and gaseous phases and calculating the hold-up from conservation equations with the help of a codified Fortran Code they named *GOWFLOW* [7].

Research reveals the attainable benefits in terms of well testing, reservoir management, production allocation and monitoring, capital and operational expenditure if multiphase flow assurance control and design strategy is fully implemented [8]. However, the knowledge of flow characteristics in multiphase wells enables the predictions of fluid dynamic behaviour and defines the design parameters and ensures maximum production over the life of the well at minimum cost [9].

Therefore, the purpose of the multiphase flow modeling and simulation of oil and gas on a vertical pipe arrangement in this research is focused on the identification and study of the flow-regime map and mostly on the pressure distribution across the pipe and the effect of each phase on the wall of the pipe. The necessity of this investigation is paramount because it is believed that effective multiphase flow will aid efficient design of satellite wells and reservoir engineering. Thus, Solidworks was used to model the pipe and to do initial flow simulation using flow-works now Solidworks Flow Simulation. Using Solidworks, the flow was a continuum flow and so no adaptation was made for the different phases of the fluid. However, the modelled pipe was imported to Ansys CFX-pre for preparation of the model and the computational domain. The pipe was separated and the mesh file saved in a format that can be accessed and used by Fluent. The mesh was then imported into Fluent for 3D simulation and the result converged after 5000 iterations.

2. MODELING AND DISCRETISATION

The modelled vertical pipe is imported to Ansys fluent for meshing. The pipe does not have complex geometries and the CFD process to be modelled does not involve discrete meshing of special regions however, a generic volumetric mesh was adequate for this problem. This is on the assumption that there are no suspended solid particles on the flow that will require particle study. The two phases are modelled to come from the same reservoir and that means they are assumed to enter the pipe at the opening simultaneously. Fig. 1 below shows the mesh model.

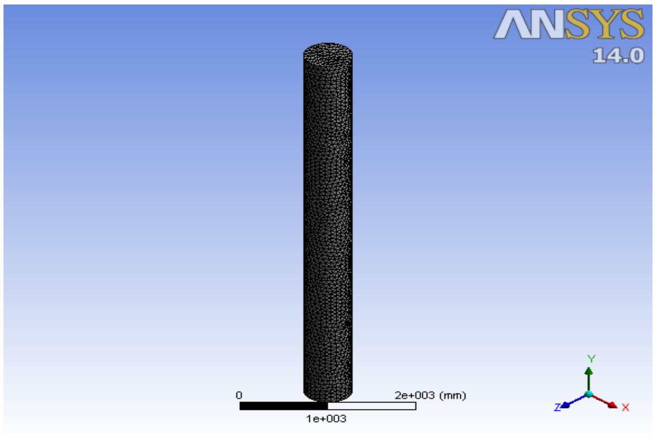


Fig. 1: Mesh Model

The discretization was carried out using tetrahedron meshing method of a smooth transition inflation option with a growth of 1.5 and transition ratio of 0.272. The mesh detail shows a total of 8816 nodes and 43535 elements.

3. MODEL EQUATIONS

Studies have shown that a lot of gas wells produce liquid such as water. Thus, in order to prevent fluid collection in well, Velocity of small droplets are often used as flow criterion [10]. The idea is that if liquid largely exist by itself as a film along the well wall, it will still mainly be transported as droplets. If the gas velocity is greater than sinking velocity for the largest liquid drops, the liquid droplets will be transported out of the production pipe. Small bubbles that rise in the continuous liquid are subjected to similar forces as liquid droplets in the gas. Stability consideration and the formula for maximum velocity for the rising gas bubbles are similar.

$$v_b^* = K_b \left[\frac{g\sigma(\rho_l - \rho_g)}{\rho_l^2} \right]^{0.25} \qquad (1)$$

Flow conditions around a gas bubble that rises in viscous liquid will be different from a liquid drop that falls in small viscous gas, so the friction factor will be different. For gas bubbles, the rise in the stagnant liquid is found by [11] as Kb = 1.53 in (1). However, under stationary (steady-state) conditions, the mass flow is constant through any cross section along the pipe.

Knowing production rates at standard conditions: q_g, q_o, q_w, we can express volume streams down in the well by using the black oil model. It is convenient to represent the volume streams by superficial velocities and fraction in (2) – (6); where flow velocity in the pipe defines the flow rate per cross-sectional area.

$$v_{sg} = \frac{Q_g}{A} = \frac{q_o(R_t - R_s)B_g}{\pi d^2/4} \qquad (2)$$

$$v_{sl} = \frac{Q_l}{A} = \frac{q_o B_o + q_w B_w}{\pi d^2/4} \qquad (3)$$

$$v_m = \frac{Q_l + Q_g}{A} = v_{sg} + v_{sl} \qquad (4)$$

$$v_g = \frac{Q_g}{A_g} = \frac{Q_g/A}{A_g/A} = \frac{v_{sg}}{y_g} \qquad (5)$$

$$v_l = \frac{Q_l}{A_l} = \frac{Q_l/A}{A_l/A} = \frac{v_{sl}}{y_l} \qquad (6)$$

Where,
y_g : gas fraction
y_l : liquid fraction

The Average density of fluid mixture in a pipe segment can be linked to fluid densities and fractions as seen in (7) and (8) which illustrates that gas has less density and viscosity respectively than the liquid and will usually flow faster [12].

$$\rho_{TP} = \frac{\rho_g A_g + \rho_l A_l}{A} = \rho_g\, y_g + \rho_l\, y_l \qquad (7)$$

$$v_g = C_o v_m + v_d \qquad (8)$$

C_o: distribution parameter for bubbles in flow, usually: 1.0 < C_o < 1.2

v_o : Buoyancy velocity of the gas bubbles, or sink velocity for droplets

Equation 9 is a summary of combining (1) and (7).

ISSN 2229-5518

$$y_l = 1 - \frac{v_{sg}}{C_o\left(v_{sg}+v_{sl}\right)+v_d} \qquad (9)$$

Similar equation to that of expression (7) and (8) by [12] is presented in (10) from an open literature according to [13] which relates gas velocity directly to the velocity surrounding liquids

$$v_g = C_o v_l + v_o \qquad (10)$$

By combining the relationship between velocity and superficial velocity to the drop relationship (10), the liquid fraction is expressed as

$$y_l = \pm\frac{1}{2}\sqrt{\left(\frac{v_{sg}}{v_o}+C_o\frac{v_{sl}}{v_o}-1\right)^2+4C_o\frac{v_{sl}}{v_o}} - \frac{1}{2}\left(\frac{v_{sg}}{v_o}+C_o\frac{v_{sl}}{v_o}-1\right) \qquad (11)$$

Relationship (11) is more complicated than (9), but it avoids most shortcomings of the [12] model. If the gas flows up and the liquid down; (11) is capable of giving one, two, or no solutions, then one solution predicts stable counter current flow. No solutions imply that counter-flow at the given rates is impossible whereas two solutions imply transition between continuous liquid and continuous gas. One of these solutions will then usually be unstable, such that the flow regimes will change.

3.1 Pressure and Flow

Assuming the flow is stratified, the following can be deduced:

To start, we may assume that the phase flows in stratified channels, as outlined in Fig. 2 below. The gas flow equation becomes

$$A_g\,dp + A_g\,\rho_g\,g_x\,dx + A_g\,\rho_g\,v_g\,dv_g + \tau_{gw}\,S_{gw}\,dx + \tau_i\,S_i dx = 0 \qquad (12)$$

Where,

A_g: gas-filled cross-sectional area

S_{gw}: contact length (perimeter), gas against pipe wall

τ_{gw}: shear stress, between gas and pipe wall

S_i: contact length between gas and liquid

τ_i: shear stress, between gas and liquid

The flow equation for the liquid channel becomes

$$A_l\,dp + A_l\,\rho_l\,g_x\,dx + A_l\,\rho_l\,v_l\,dv_l + \tau_{lw}\,S_{lw}\,dx - \tau_i\,S_i dx = 0 \qquad (13)$$

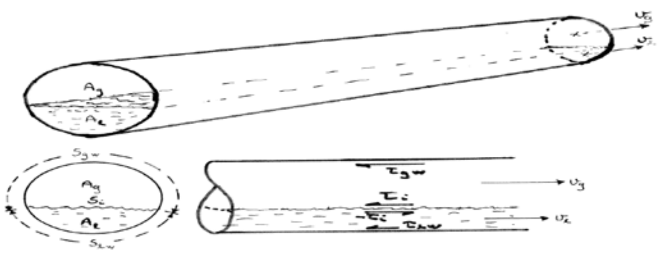


Fig. 2: Gas and liquid flow with different rates

For horizontal pipes, flow with small velocity will be stratified [14] predicted velocity and fractions of stratified flow are used as stability analysis to assess whether the stratified solutions were physically realistic, or whether the gas and liquid would be divided in any way in the pipe. If gas and liquid flow together as in the case with this work, we need a flow equation for the mixture; it is determined by putting: $A_g/A = y_g$, $A_l/A = y_l$ into (12) and (13), and by its addition: it gives the elimination of the inter-phasial shear. The mixed flow equation becomes:

$$dp + \left(\rho_g y_g + \rho_l y_l\right) g_x dx + \rho_g v_g y_g dv_g + \rho_l v_l y_l dv_l + \frac{\tau_g S_{gw} + \tau_{lw} S_{lw}}{A} dx = 0 \qquad (14)$$

It observed that the last part in (14) contains shear stresses and wetted perimeters, if we presume that perimeter is proportional with fractions ($S_{gw}=y_g S = y_g \pi d$), and represent the velocity by (1) and (2); the shear contribution can be developed as:

$$\frac{\tau_g S_{gw} + \tau_l S_{lw}}{A} = \frac{1}{\pi d^2/4}\left(\frac{1}{8} f_g \rho_g v_{sg}\left|\frac{v_{sg}}{y_g}\right|\pi d + \frac{1}{8} f_l \rho_l v_{sl}\left|\frac{v_{sl}}{y_l}\right|\pi d\right) = \frac{1}{2d}\left(f_g \rho_g v_{sg}\left|\frac{v_{sg}}{y_g}\right| + f_l \rho_l v_{sl}\left|\frac{v_{sl}}{y_l}\right|\right)$$

When gas and liquid flow in the same direction, absolute values will be ignored and by assumption of equal friction factors for liquid and gas : $f_g = f_l = f^o$, we have:

$$\frac{\tau_g S_{gw} + \tau_l S_{lw}}{A} = \frac{1}{2} f^o \frac{1}{d}\left(\rho_g \frac{\lambda_g^2}{y_g} + \rho_l \frac{\lambda_l^2}{y_l}\right) v_m^2 \quad (15)$$

Often the shear contribution in (15) can be expressed as flow of homogeneous mixture in (16)

$$\frac{\tau_g S_{gw} + \tau_l S_{lw}}{A} = \frac{1}{2} f_{TP} \frac{1}{d} \rho_m v_m^2 \quad (16)$$

Where f_{TP} is the two phase friction factor, estimated from the correlation for single-phase flow, with a correction factor for the drop:

$$f_{TP} = f^o c_{TP} \quad (17)$$

The comparable single phase friction factor f^o is estimated by standard single-phase correlation (for example: $f^o = 0.16/Re_m^{0.172}$) with Reynolds Number for the homogeneous mixture, is usually defined as

$$\mathrm{Re}_m = \frac{\rho_m v_m d}{\mu_g \lambda_g + \mu_l \lambda_l} \quad (18)$$

From equation (16) and (18); a slip correction factor will be derived as:

$$c_{TP} = \frac{\rho_g}{\rho_m}\frac{\lambda_g^2}{y_g} + \frac{\rho_l}{\rho_m}\frac{\lambda_l^2}{y_l} = \frac{\rho_g y_l (1-\lambda_l)^2 + \rho_l y_g \lambda_l^2}{\rho_m y_l (1-y_l)} \quad (19)$$

With this theoretical basis, the pressure gradient can be calculated using (20)

$$\frac{dp}{dx} + \rho_{TP} g_x + \rho_g v_{sg} \frac{dv_g}{dx} + \rho_l v_{sl} \frac{dv_l}{dx} + \frac{1}{2} f_{TP} \frac{\rho_m}{d} v_m^2 = 0 \quad (20)$$

The theory above involved many assumptions and approaches. Published models for steady-state two phase flows may have several deviations somewhat from the basis outlined above.

3.1.2 Pressure and flow conditions along the pipe:

Relations above are applied to steady-state flow at a given pressure, temperature and flow rate. Along the well pipe we will then have constant mass flow, while pressure and temperature will change. Equation (20) enables calculation of pressure changes along the pipe, and thus also phases relationships, viscosity, volume and velocity. A common task is that knowing the expected well pressure: p_w, for a given reservoir pressure and rate; thus, to estimate the tubing head pressure: p_{th}. The tubing head pressure is connected with expression below:

$$p_{th} = p_w + \left.\frac{dp}{dx}\right|_{\bar{p}}^{\bar{T}} \cdot L_w \quad (21)$$

where:

L_w = length along well pipe

$\left.\frac{dp}{dx}\right|_{\bar{p}}^{\bar{T}}$ = Average pressure gradient, estimated from

Well pressure: $\bar{p} = \frac{p_w + p_{th}}{2}$, And temperature:

$$\bar{T} = \frac{T_w + T_{th}}{2}$$

Since we do not know the average pressure in the well before we have estimated the pressure, iterations are required: we may estimate the pressure gradient at the bottom, solve (21), and then use the solutions to estimate the average pressure. In many cases it provides pretty good estimate of the pressure. Such step will be called a Single-Step Runge-Kutta Solution. If the pressure gradient change much, it would be desirable to assume the pressure and flow conditions at intervals along the pipe. The algorithm can then formally be expressed as:

$$p_{i+1} = p_i + \left.\frac{dp}{dx}\right|_{p_{i+0.5}}^{T_{i+0.5}} \cdot (L_{i+1} - L_i) \quad (22)$$

The multiphase equation; energy flow equation in fluent were used to solved the model. The boundary condition at the inlet is a velocity flow of 5m/s for the mixture and the outlet was set to standard atmospheric pressure.

4 RESULTS PRESENTATION AND DISCUSSION

The solution was solve with different method, first the volume of fluid method was used with two different phases, the discretisation scheme for volume fraction was solve with geo construct and then pressure to PRESTO.

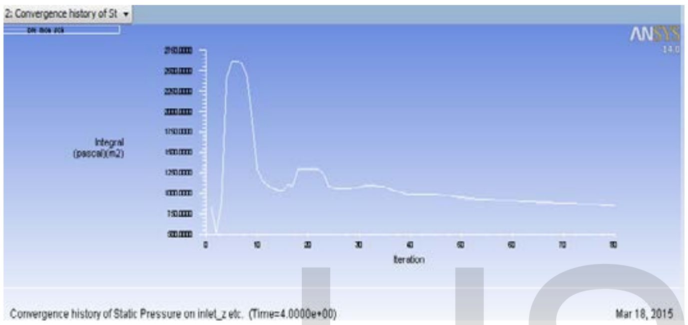


Fig. 3: Convergence history of Static pressure.

Pressure based solver is used and the time dependency of the solution is transient. For the transient, pressure base solver in Fluent, the flow time was set to 50s meaning a time step of 100 and a cycle time of 20 iteration per time step, and updating the result after every time step. The pressure was found to converge after 340 – iteration. The plot in fig. 3 above shows the static pressure converging after a few iterations. The result shows a very high pressure drop across the vertical pipe as is shown in fig. 4 below. At initial set-up, the pressure at the outlet was set to atmospheric pressure but due to the anticipated pressure drop across the vertical pipe with the calculation from Fluent shows the pressure at when the fluid finally leave the pipe at the outlet to have drop significantly. Considering (22), the pressure drop is a function of length as also in (9). As the length of the pipe increases, the pressure drop also increases.

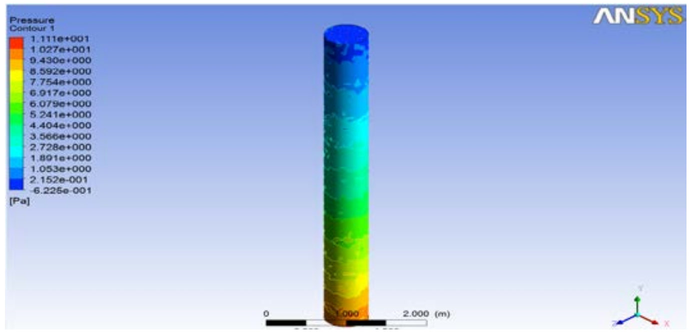


Fig. 4: Pressure Plot across the Pipe.

A very important observation from the plot is the wavy nature of the flow across the pipe. The velocity of the gas bubbles is higher than that of the liquid and as such it forms a wavelike motion as it travels across the length of the pipe. This was shown to correlate with the result gotten from the calculation in Fluent as the velocity of the gas which in this case is the primary phase is 10.65129 m/s while that of the liquid (oil) is at 5m/s. Fig. 5 and 6 below shows the velocity plot of phase 1 across the pipe and the stratified mixture of oil and gas due to gas bubbles respectively.

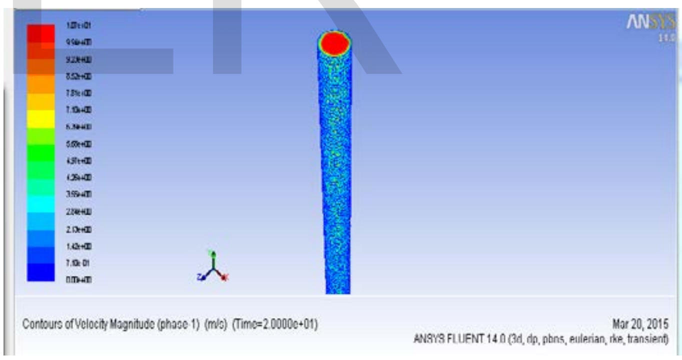


Fig. 5: Velocity Contour Plot of Phase 1 (Gas)

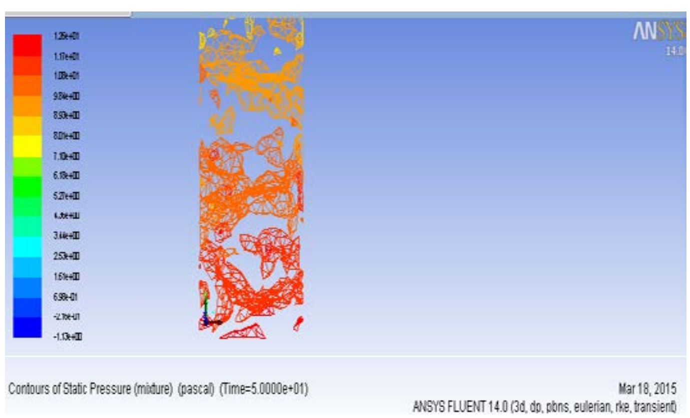


Fig. 6: Contour of Static Pressure showing the stratified nature of the mixture of oil and gas

Other simulation results such as Convergence History of Velocity inlet, Contour Plot of Dynamic pressure of gas bubbles along the pipe, Initial result from Solidworks Flow on friction forces are anaysed and shown in appendix – 1; while appendix – 2 is analysis of results as the CFD – Ansys Fluent tool is used to simulate the multiphase flow of oil and gas.

5 CONCLUSION

The use of CFX-Pre and Ansys Fluent to analyze the multiphase flow simulation of oil and gas in vertical flow was successful from the reported results. Interestingly, pressure drop across the vertical pipes were confirmed high to validate the concepts from the open literature which attributes to the fluctuation of the processing equipment causing catastrophic failure. Also results attests the difference in velocities of gas and liquid bubbles which causes wavy motion as the flow travels along the pipes, this is clear evidence that sudden changes in phase is an effect of heat addition or exchange of heat between the phases also know as flashing due to depressurization as confirmed in the research.

At a close watch of the simulation, the pressure was found to converge just after 340 – iterations. This is to correct the impression that there are satisfactorily mathematical principles to enhance the prediction of multiphase flows such as improving pressure drop in pipes. Thus, mathematical algorithm of pressure and flow conditions along pipes at an interval expressed in (22) and (9) are highly recommended since, the pressure drop is a function of length of pipe; and as the length of the pipe increases, the pressure drop also increases. Therefore, pressure drop can be controlled by subsequent reduction of pipe length.

AKNOWLEDMENT

Special appreciation to God for His divine knowledge of understanding and other well wishers.

REFERENCES

[1] L. Djamel, “Multiphase flow simulation in pipes& risers”, *TransAT for Oil & Gas*, 2014.

[2] R. Issa, “Simulation of intermittent flow in multiphase oil and gas pipelines**”,** *proc. Seventh International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia, pp. 9-11*, 2009.

[3] E. Duret, I. Faille, M. Gainville, V. Henriot, H. Tran, and F. Willien, “Mathematical and Numerical Aspects of Low Mach Number Flows”, *Division Technologie Informatique et Math’ematiques Appliquées*, 2004.

[4] R. Belt, B. Djoric, S. Kalali, E. Duret, and D. Larrey, “Comparison of commercial multiphase flow simulators with experimental and field databases”, *BHR Group’s Multiphase Production Technology Conference in Cannes,* 2011.

[5] L. Djamel “Advanced simulation of transient multiphase flow & flow assurance in the oil & gas industry”, *Can. J. Chem. Eng*. 9999:1–14, DOI 10.1002/cjce.21828, 2013.

[6] A. A. Olusiji A. “Modeling of Petroleum gas – liquid stratified flow in an inclined wellbore and a bend”, *ASPES, Vol.2, No.4, ISSN: 1937-7991*, 2010.

[7] P. F. Sergio, and B. G. Marcela, “A numerical model for multiphase flow on oil production wells”, *Center for Industrial Research, Tenaris – Campana, Argentina*, 2006.

[8] A. Salem, M. David, and A. Abdulmajid, "Upstream multiphase flow assurance monitoring using acoustic emission" Available on: http://www.intechopen.com/books/acoustic-emission/multiphase-flow-assurancemonitoring-using-acoustic-emission, 2012.

[9] B. S. Arthur, F. C. S. António, and R. M. Clovis, "Numerical solution of the multiphase flow of oil, water and gas in horizontal wells in natural petroleum reservoirs", *Asociación Argentina de Mecánica Computacional http://www.amcaonline.org.ar*, *Vol XXXI, págs. 683-693*, 2012.

[10] W. D. Baines, and J. S. Turner, "Turbulent Buoyant Convection from a Source in a Confined Region", *Journal of Fluid Mechanics*, Vol. 37, pp. 51 – 80, Issue 01, 1969.

[11] Z. H. Tabor, "Velocity of large drops and bubbles in media of infinite or restricted extent", *AIChE Journal*, Vol. 6, pp. 281 – 288, Issue 2, DoI: 10.1002/aic.60222, 1960.

[12] N. Zuber, and J. A. Findlay, "Average Volumetric Concentration in two Phase flow System", *Trans. Of ASME, Journal of Heat Transfer*, Vol. 87, pp. 453 – 468, 1965.

[13] H. Asheim, "Mono: An Accurate Two – Phase Well Flow Model Based on Phase Slippage", *SPE – Production Engineering*, pp. 221 – 230, 1986.

[14] Y. Taitel, and A. E. Dukler, "A Model for Predicting Flow Regime Transitions in Horizontal and near Horizontal Gas – Liquid Flow", *AIChE Journal*, Vol. 22, pp. 47 – 55, Issue 1, 1976.

APPENDIX – 1

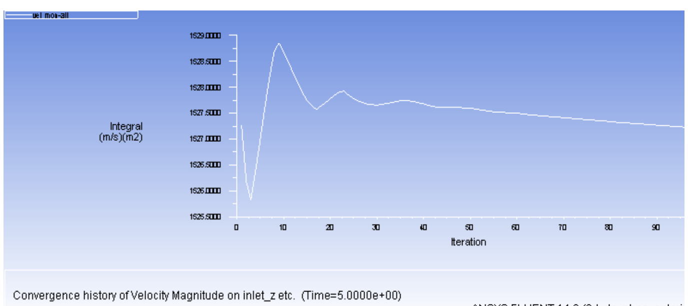


Fig. 1.1: Convergence History of Velocity on inlet after 90 iteration

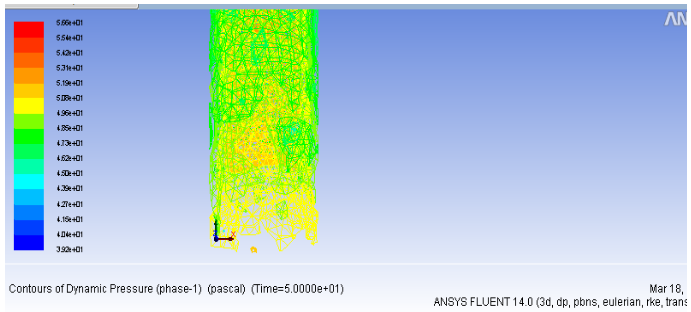


Fig. 1.2: Contour Plot of Dynamic pressure showing the gas bubbles along the pipe.

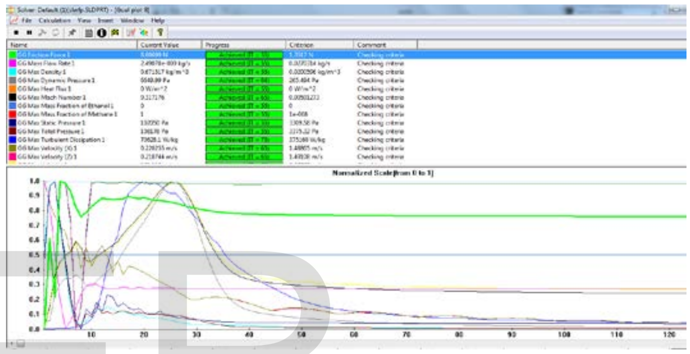
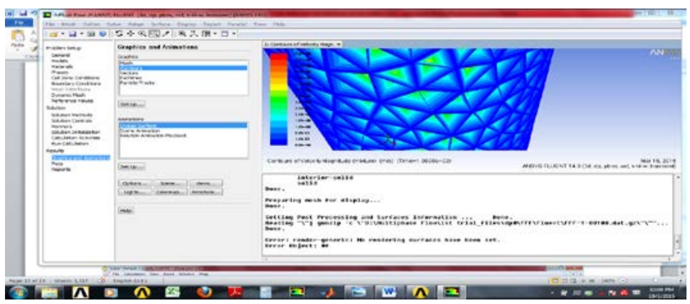


Fig. 1.3: Summary of Initial Result from Solidworks Flow

APPENDIX – 2



ISSN 2229-5518