A COMPARATIVE STUDY OF MULTIPHASE CFD SIMULATION WITH LABORATORY EXPERIMENT FOR PRESSURE DROP AND VELOCITY PROFILE IN A VERTICAL PIPE

BY

GINA FAITH NWOKOCHA,
ANDREW HUNT,
PHILIP CUNNINGHAM
&
LAWRENCE TOM
CENTRE FOR FLOW MEASUREMENT AND FLUID MECHANICS RESEARCH, COVENTRY UNIVERSITY, UK.

ABSTRACT

Technology has always been the driving force of the oil and gas industry. Due to the complex nature of multiphase flow, expert knowledge is required in the industry to make reasonable prediction of flow parameters. The question of accuracy, however, remains a matter of debate, thus the need for comparison. By setting similar conditions of pipe geometry, temperature, and input flow conditions; pressure drop and fluid velocity for oil and gas were modelled and measured using computational fluid dynamics (CFD) and experimental method respectively. A comparison is made from both methods to validate CFD. This was achieved by an experiment conducted in the Coventry University multiphase flow loop laboratory. The flow configuration is a two-phase oil and gas flow through a vertical pipe flow of 1.5m height and 80mm internal diameter. Pressure was measured from point 1 to 4 corresponding to heights of 1.3m, 1m, 0.5m, 0.2m respectively across the vertical pipe. The flow regime and velocity profile were obtained using an Electrical Capacitance Tomography (ECT) sensor. CFD simulation was run with the same conditions used in the experiment with the aid of ANSYS 16.1 CFX software. The final results were compared to check error margin between the simulation and the experiments. A 4.03% error was calculated from the experimental results for pressure drop from PT 1 to PT 4 which is not too far from expected. However, the gas velocity results obtained from CFD simulation were quite different from that of the Electrical Capacitance Tomography (ECT) sensor from the experiments as it gave a higher error margin of 23.76%. The analysis concludes that CFD is a valid tool for making prediction in the oil and gas industry.

KEYWORDS:* *Multiphase flow, Computational Fluid Dynamics (CFD), Electrical Capacitance Tomography (ECT), Vertical Riser section.

INTRODUCTION

Multiphase flow is the simultaneous flow of more than one phase of fluid (immiscible fluids - gas, oil or water) in any medium that allows flow.

In the oil and gas industry, there is a need placed on engineers to make reasonable predictions of pressure drop, flow patterns, velocity profile and the relationship between them used in multiphase flow design studies. To this end, different correlations have been developed experimentally over the past decades. Many of the correlations date to the 1940s using small pipes, frequently 1 inch (25mm) diameter (Lockhart & Martinelli, 1949). These correlations are often used for flows in much larger pipe sizes today, potentially as a factor of disparity between modelling and measurement of flow parameters.

As opposed to single phase flow systems, different flow regimes exist for multiphase flow in vertical pipe. These regimes are dependent on the gas, liquid and solid flow rates, fluid properties, pipe sizes, inclination angle and other factors that could affect the impact velocity of solid particles resulting in erosion problems (Trallero, Sarica, & Brill, 1997). The regimes are slug, churn bubble annular, etc. However, this work will focus on bubble flow regime (a situation where gas is dispersed in the liquid) in a two-phase flow (Oil and Gas).

This project looks at the validation of multiphase Computational Fluid Dynamics (CFD) simulation against laboratory measurement. It identifies slight variation of results and explains the reason for the variations. Its aim is to identify the capabilities and limitations regarding determining the pressure drop and velocity profile of a multiphase flow system and the role it plays in production optimization. Accurate prediction of pressure drop and velocity profile encountered during a multiphase flow is required for a good engineering design and development of a multiphase flow model. Hence the importance of this research in improving multiphase flow production technology.

The study involves laboratory measurement of multiphase flow systems and CFD simulations for multiphase flow using ANSYS 16.1 CFX software. The configuration is a vertical pipe configuration, typical of the vertical section of a riser system. The results of which are compared to show the validity and reliability of simulation results.

In the oil and gas industry, pressure drop is a very sensitive and important aspect of production. A good prediction of the pressure drop helps in pump and production tubing sizing as well as equipment maintenance. However, a wrong prediction of the pressure drop over a period can lead to unplanned work over activities, and decline in production resulting in significant financial losses.

Multiphase production and CFD

Multiphase production is a vital issue of consideration as almost all oil and gas well flow occurs in multiphase conditions before separation takes place. From the initial flow of fluid in the reservoir to the final production, elements of the three phases (gas, oil and water) exist to give rise to multiphase flow systems. An estimate of over half of anything produced in an industrial society, to a reasonable degree relies on a multiphase process (Prosperetti & Tryggvason, 2003). However, the challenges associated with multiphase flow are issues that need to be tackled with technological expertise. Some of these challenges are those resulting in flow assurance problems. For example; the formation of hydrates by the combination of natural gas and water at high pressure and low temperature, corrosion of the pipe due to the presence of liquid water in the hydrocarbon, scaling from mineral salt deposits, wax formation, asphaltenes and other problems from the environment of production (Saleh, 2002).

The introduction of computers in the oil and gas industry has improved investigation and possible solutions to multiphase flow problems. Over the years, multiphase flow modelling and simulation has become a great engineering tool to conduct virtual experiments during design and development for most industries. This is as a result of the significant improvement in the approach of modelling that gives more details of multiphase flow physics and phenomena (Lun, Calay, & Holdo, 1996).

Computational Fluid Dynamics CFD is the application of mathematics, physics and computational software to solve and analyse problems that involve fluid flow. It makes use of numerical analysis and algorithms to solve fluid flow problems.

Numerical methods for unsteady state (transient) single–phase flow are readily available and in extensive use for practical implementation while that of multiphase flow is still developing. However, there is a great deal of experience available for two-phase flows especially gas-liquid and gas-dispersed solid particle flow (specifically for the less concentrated dispersed phase) (Kolev, 2007). It can be used in solving fluid flow related problems like density, flow velocity, chemical concentration and temperature.

Multiphase flow in a vertical pipe

The analysis of multiphase flow through a vertical pipe is very important in the oil and gas industry as all drilled wells (including horizontal, deviated and multilateral wells) have a vertical section to bring the fluid to the well head. A typical example is the riser, whose function is to transport fluid from the sea bed to an FPSO or other production platforms.

Velocity Measurement with Electrical Capacitance Tomography (ECT)

Electrical Capacitance Tomography (ECT) is a non-invasive and non-intrusive method of measuring velocity and concentrations. This method was introduced in the 1980s and since then various studies have been carried out based on this technology. An ECT system consists of a set of capacitance electrodes which are usually mounted around the object of interest whose measurement is required. This is done by the measurement of all possible electrode combination (Yang, Beck, & Byars, 1995). As stated by Yang, Beck and Byars, the first ECT real-time system in the world was developed by a combined effort of the University of Leeds, University of Manchester Institute of Science and Technology (UMIST), Schlumberger and Process Tomography Ltd.

In 2003, a novel tomographic flow analysis system was developed by Hunt, Pendleton, and White. The system acquires a comprehensive representation of concentration structure and velocity in a picture form by means of a twin-plane tomographic data. The depth of detail which can be derived from a flow measurement based on ECT is shown by a simple experimental setup of gravity drop flow (Hunt, Pendleton, & White, 2003). From their work, the flow was divided based on user defined zones, where concentration-time plot was viewed and transit velocity is established from correlation at different times within each zone. Calculation of the volumetric flow rate is done by the integration of velocity multiplied by concentration, giving rise to an easy calculation of the entire volume going through any period. These results are obtained from an ECT flow analysis system (Tomoflow R100) which has a high speed design capacitance measurement.

For ECT, the change in capacitance which is caused by changes in the dielectric substance is measured, from a multi-electrode sensor. The electrodes are typically 8 or 12 in number (Ismail, Gamio, Bukhari, & Yang, 2005). In the study of "Tomography for multi-phase flow measurement in the oil industry", the group came up with conclusion that ECT can be applied in situations where flow-regime dependency is a problem in multiphase flow measurements.

FLOW REGIMES FOR MULTIPHASE FLOW

It is important to know the different flow regimes as these have a significant effect on the void fraction and frictional pressure drop. In terms of void fraction, there is a significant increase when the gas flow rate increases for bubbly and slug regimes, while it remains almost constant for annular flow even with a significant gas flow rate increase as seen in the works of Ghajar and Bhagwat (2014).

Over the past four decades, considerable attention and research has been given to the study of bubble flow. This is due to its recent wide applicability range and the effect it has on some processes (Abdulmouti, 2015). It has proven to be most prevalent flow regime experienced in the laboratory experiments, due to the reduced flow rate of the gas. Usually for a bubbly flow to occur, the liquid flow rate has to be high enough to break up the stream of gas into small bubbles that will be dispersed in the liquid (McCready, 1998). Also, in the field, bubbly flow can occur in the production of under saturated oil sometimes referred to as black oil, due to the reduced gas oil ratio, the high oil flow rate compared to gas and the fact that the oil is in the continuous phase with little amount of dispersed gas in it. Better prediction of pressure drop and heat transfer is achieved with a flow pattern dependent model

OVERVIEW OF CFD SIMULATIONS AND EXPERIMENTS

Previous models on multiphase flow are based on one-dimensional simulation tools that are usually used for designs in the oil and gas industry. Pipesim and OLGA are two of the software packages that are one-dimensional simulation tools. Comparative studies have been carried out with experimental data and field data to validate these software. As seen in the work of Belt, Duret, Larrey, Djoric, and Kalali (2011). the result shows good agreement between the software (OLGA and LedaFlow) and the field measurements for temperature and pressure drop as summarized in the table below. However, they noted that some predictions were not good for some flow regimes that are not properly modelled in the simulation.

Table 1: Well Measurements and Simulation Results (Belt et al., 2011)

Parameters	Measured	LedaFlow®	OLGA® 5.3
P at gauge (bara)	189.8	187.3	187.3
T at gauge (^{0}C)	54.0	53.5	53.5
P at wellhead (bara)	128.9	128.7	128.9
T at wellhead (^{0}C)	49.2	49.3	49.2

Advances in technology have given rise to 3-dimensional simulations CFD tools which provide detailed simulation of liquid-gas, heat and mass transfer, temperature and pressure. The modelling approaches used in most CFD software packages are Eulerian-Lagrangian or Eulerian-Eulerian of which the latter is more general because it allows simulation of all types of flow provided the phase interaction models are properly defined. Both Eulerian-Lagrangian and Eulerian-Eulerian can be used to model dynamic motion of dispersed phases in a continuous flow (Gharaibah, Read, & Scheuerer, 2015).

CFD is a fast growing branch of fluid mechanics and a high-fidelity predictive method that is required in the industry to complement experimental results for better safety and control measures as well as production optimization. It is important to validate simulations with experimental results because of the complexity in gas-liquid flows, where the phenomenon of flow occurs on a large range of time and space scale. The advantage of CFD simulation comparison is that the dimensions of equipment and fluids can be easily varied or replaced and it does not require building an experimental setup (Simcik, Mota, Ruzicka, Vicente, & Teixeira, 2011).

METHODOLOGY

The experimental measurements were undertaken in a mobile flow loop in the Coventry University multiphase flow laboratory. This loop is a three phase flow well simulator manufactured by Cussons Technology Ltd, though the work reported here is two-phase (oil and gas). The loop comprises of two vertical cylinders of height 1.5m. The outer diameters (OD) are 90mm are 200mm respectively, each fitted with four pressure transducers PT1 to PT4 corresponding to heights of 1.3m, 1m, 0.5m, and 0.2m respectively. The gas flows via a connected tube to the 1.5m vertical cylinders. The results presented here are from the 90mm pipe with the clamped on ECT. A silicone based oil of 872.5kg/m^3 density and 1.744cP viscosity is used as the liquid phase while air of 1.23kg/m^3 density and 0.0186cP viscosity was used, both at 25^{o}C and 1bar. The schematic of this process shown below.

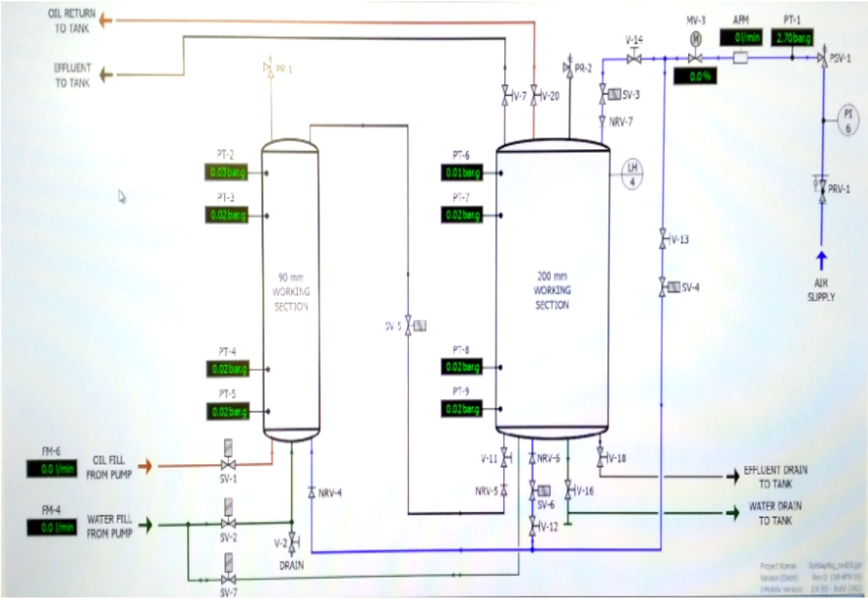

Figure 1 Schematic of flow control process

The pressure drop is calculated from the difference between the pressure transducer readings of point 1 and 4 while the Bernoulli and continuity equations are applied to calculate the density of the mixture as function of pressure drop.

$$P + \rho gh + \frac{1}{2}\rho v^2 = constant \qquad (2$$

From continuity equation, velocity in = velocity out.

This reduces the equation to:

$$P + \rho g\mathrm{h} = constant \qquad (3)$$

Therefore density of the mixture will be;

$$\rho m = \frac{P2 - P1}{(\mathrm{h}1 - h2)g} \qquad (4)$$

Density can be expressed as a function of the phase fraction from ECT results as:

$$\rho m = \rho o(\gamma o) + \rho g(\gamma g) \qquad (5)$$

The software used in the CFD simulation is ANSYS 16.1 CFX. The solution method of this software is the finite volume technique. A replica of the experimental geometry was created in the 3D CAD model with a structured triangular surface mesh as shown.

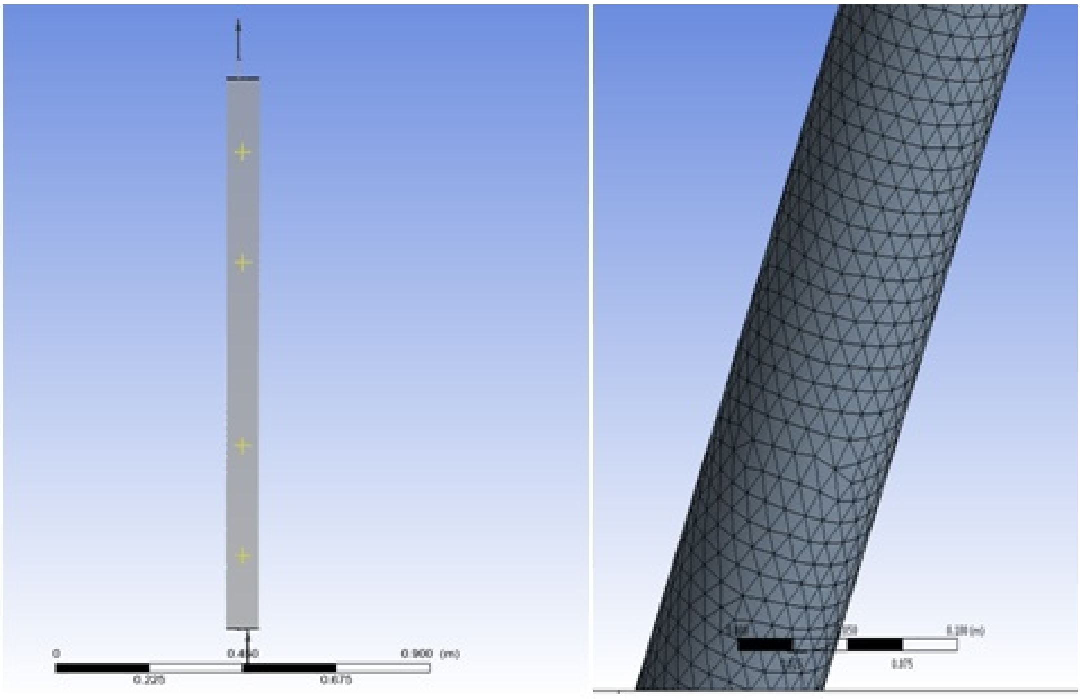
Figure 2: Pipe Geometry and Meshing

The Eulerian-Eulerian model is used. Also, the dispersed phase model is selected as opposed to homogenous model. The fluid type is selected as gas-liquid with physical time scale of 0.1s and 10000 iterations.

RESULTS AND DISCUSSION

The results of the experiment and simulation are shown below in various plots for analysis. The measured pressures from the experiment are shown in Fig.3. This trend is synonymous with a reservoir fluid moving up the production tubing or riser. The increase in height column of the fluid gives a corresponding increase in the pressure drop. The drop in pressure can also be function of the smoothness of the pipe etc. From the plot engineers can extrapolate to predict the final pressure that is expected when the fluid is out at the well head and separator. This will aid the sizing of pump for optimum production.

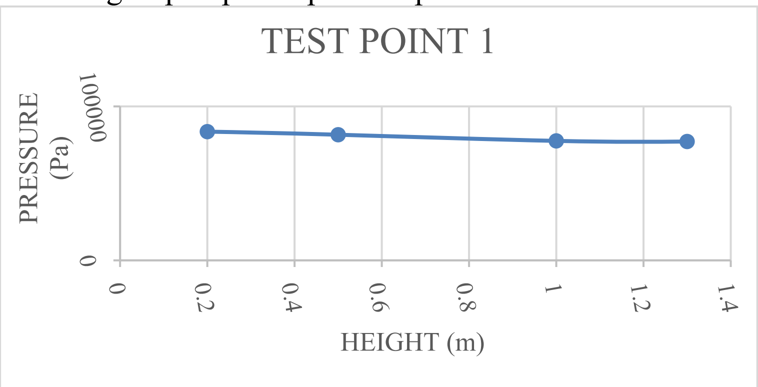

Figure 3: Plot of pressure against height of pipe (Experiment)

The flow snippets from the ECT clearly shows bubbly flow regime as the blue colour indicates tiny bubbles coming out in the centre of the liquid. The green colour shows the transition between the liquid and gas. The measurement is done by means of the change in capacitance which is caused by changes in the dielectric substance. A typical result for test point 1 from Flowan software is shown.

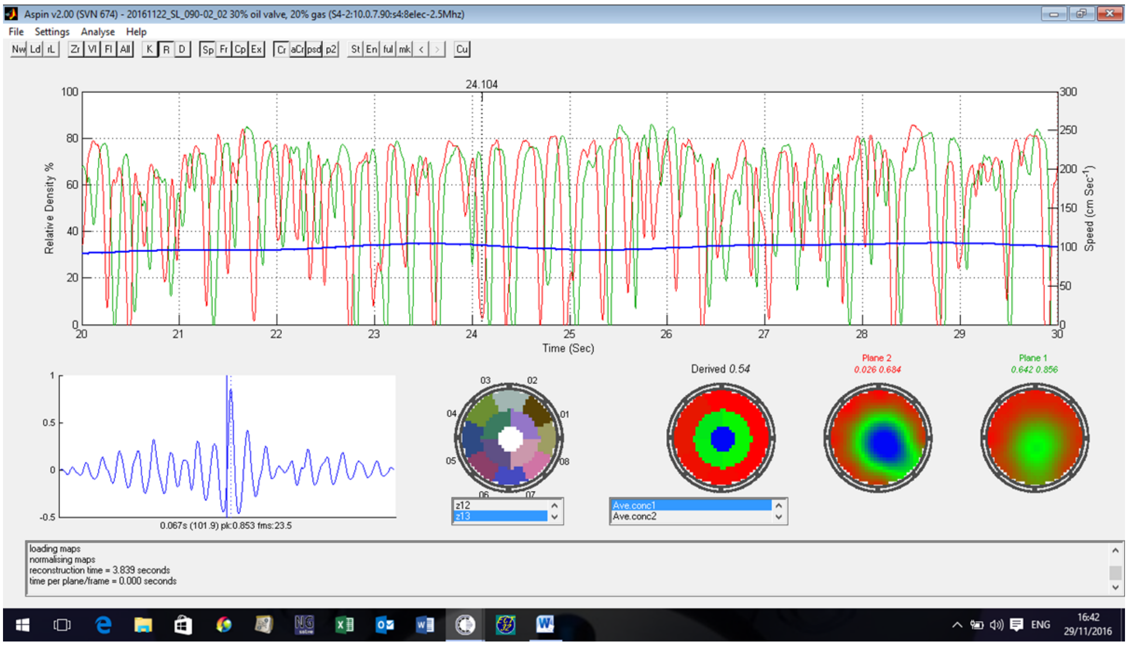

Figure 4: Screenshot of Flowan display screen

Based on the input parameter used from the experiments series of data were obtained from CFD simulation which were used to make similar plots from experiment to enable comparison. The plot of pressure against height from CFD results is given below

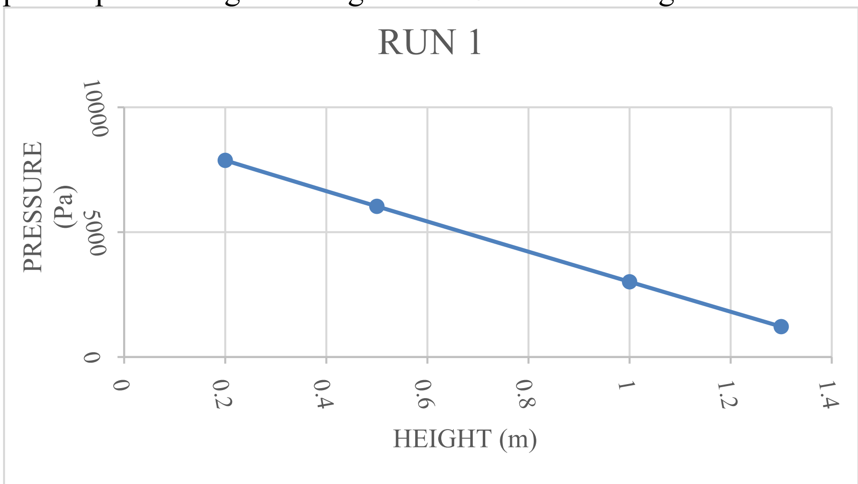

Figure 5: Pressure against height of pipe

The CFD simulation for pressure and velocity is also shown below.

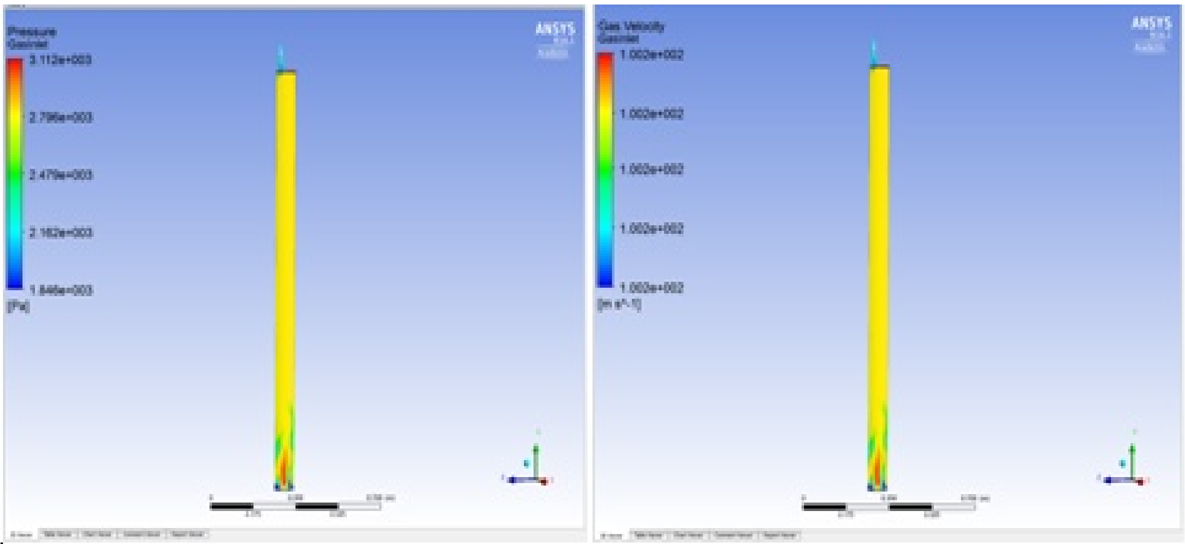

Figure 6: CFD simulation for a fully developed flow

Table 2 shows results of pressure drop and velocity for both experiment and CFD with the calculated percentage error.

Table 2: Experiment and CFD Result

TEST 1	EXPERIMENTS	CFD	PERCENTAGE ERROR (%)
Pressure drop (Pa)	6400	6658	4.0
Gas velocity (m/s)	0.96	0.73	23.8

The percentage difference for pressure drop is calculated as 4.03% while that of the average velocity was calculated to be 23.8%. The two methods calculate gas velocity differently: for ECT this is the cross-correlation velocity while for CFD it is the average velocity of the entire pipe section. This result compares well with Rahimi and Abbaspour (2008) CFD prediction of 14% and 21% for pressure and velocity respectively.

Other comparisons were made in terms of terms of mixture density as a function of the phase fraction and pressure drop. This was calculated by applying Bernoulli's and continuity equation to arrive at the eqn. 4. A plot of both CFD and Experiment gives very close relationship as shown below.

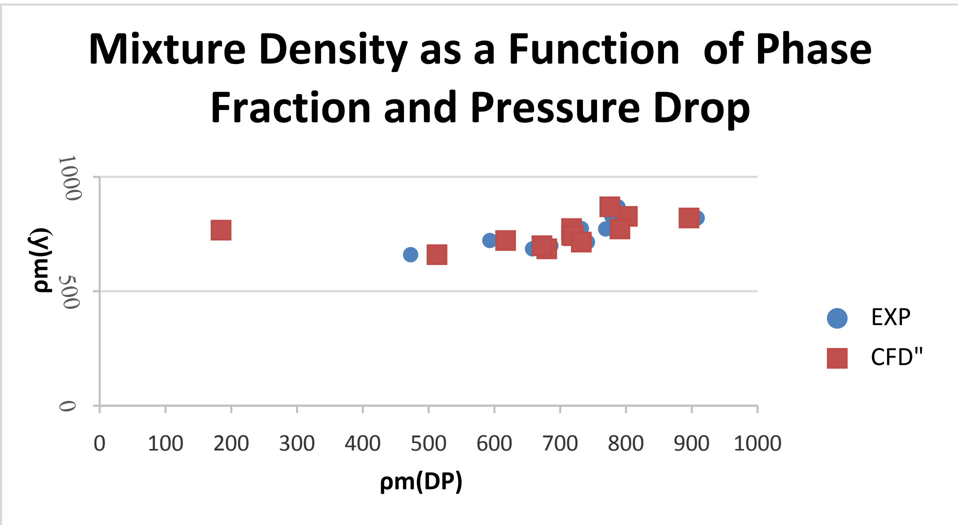

Figure 7**:** Mixture density plot

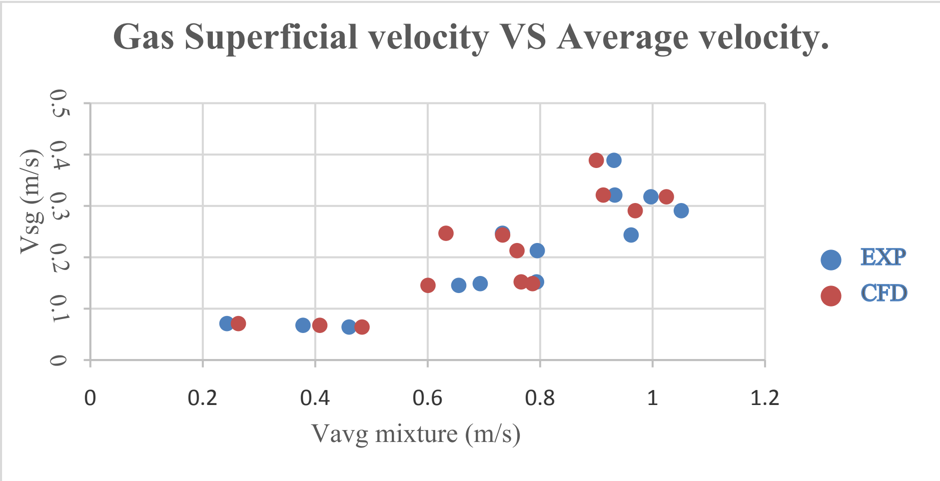

Figure 8**:** Gas superficial velocity vs. average velocity

The gas superficial velocity was plotted against average velocity of the mixture to give the plot above. The plot shows a scattered graph with a gradual increase of the mixture velocity as the gas superficial velocity increases. This can be compared with Simcik et al. (2011). They plotted liquid interstitial velocity against gas superficial velocity with for both CFD and experiments. Their result gave a 0.7% to 4.4% relative difference. However for this plot, the average percentage difference between CFD and experiment is 2.28% which compares favourably with Simcik et al. (2011).

Conclusions

A lot of research has been done in the area of multiphase flow analysis. Application of CFD and experimental approach has been on the frontline in recent literature. The observation from this is that CFD results are often very close to experiment, although with few exceptions. This research work adds to existing literature by comparing CFD and experimental results for multiphase flow.
The liquid-gas test was conducted in a three-phase flow loop with vertical pipe of 1.5m height and 80mm ID, while ANSYS CFX 16.0 was used for simulation. The comparative results in terms of pressure drop and velocity shows a slight difference of 4.0% and 23.8% respectively. The results from the entire work shows the prediction from CFD are relatively close to that of the experiments, despite the limitations in definition of initial conditions and the short development length in both experiment and model. The results presented here are limited in their range, but show reasonable comparisons between CFD and experiment measurements pressure drop and gas velocity in a multiphase system.
One of the limitations with CFD is that it requires a lot of technological expertise and lots of experience to confidently run the simulation. Due to the complexity of multiphase flow systems, the appropriateness of the modelling technique in CFD for a particular case is difficult to select.

Recommendations

In other to reduce/eliminate these limitations for better results and reliability, continuous training on the use of CFD should be adopted. Increase research of multiphase flow should always adopt CFD simulations to grow the knowledge base of CFD. Further work is required for sensitivity tests based on choices for turbulence, drag model, bubble size etc.
This research shows that CFD has proven to be a valid tool for prediction; however, improvements are required to reduce the error margin of predictions as well as increase the validity range of the software.

REFERENCES

Abdulmouti, H. (2015). Experimental measurement for surface flow characteristics generated by a bubble plume. *Journal of Flow Visualization and Image Processing,*

Belt, R., Duret, E., Larrey, D., Djoric, B. & Kalali, S. (2011). (2011). Comparison of commercial multiphase flow simulators with experimental and field databases. Paper presented at the *15th International Conference on Multiphase Production Technology,*

Ghajar, A. J. & Bhagwat, S. M. (2014). Flow patterns, void fraction and pressure drop in gas-liquid two phase flow at different pipe orientations. *Frontiers and progress in multiphase flow I* (pp. 157-212) Springer.

Gharaibah, E., Read, A. & Scheuerer, G. (2015). (2015). Overview of CFD multiphase flow simulation tools for subsea oil and gas system design, optimization and operation. Paper presented at the *OTC Brasil,*

Hunt, A., Pendleton, J. D. & White, R. B. (2003). (2003). A novel tomographic flow analysis system. Paper presented at the *3rd World Congress on Industrial Process Tomography, Banff, Canada,* 3-5.

Ismail, I., Gamio, J., Bukhari, S. A. & Yang, W. (2005). Tomography for multi-phase flow measurement in the oil industry. *Flow Measurement and Instrumentation, 16*(2), 145-155.

Kolev, N. (2007). Multiphase flow dynamics, vol. 2 thermal and mechanical interactions, 3d extended Ed.

Lockhart, R. & Martinelli, R. (1949). Proposed correlation of data for isothermal two-phase, two-component flow in pipes. *Chem.Eng.Prog, 45*(1), 39-48.

Lun, I., Calay, R. & Holdo, A. (1996). Modelling two-phase flows using CFD. *Applied Energy, 53*(3), 299-314.

McCready, M. (1998). *Demonstration of the effect of flow regime on pressure drop*. Retrieved from http://www3.nd.edu/~mjm/Effect.of.Flow.Regime.nb

Messa, G. V., Ferrarese, G. & Malavasi, S. (2015). A mixed Euler–Euler/Euler–Lagrange approach to erosion prediction. *Wear, 342*, 138-153.

Prosperetti, A., & Tryggvason, G. (2003). Appendix 3: Report of study group on computational physics. *International Journal of Multiphase Flow, 29*(7), 1089-1099.

Rahimi, R. & Abbaspour, D. (2008). Determination of pressure drop in wire mesh mist eliminator by CFD. *Chemical Engineering and Processing: Process Intensification, 47*(9), 1504-1508.

Saleh, J. (2002). *Fluid flow handbook* McGraw-Hill Professional.

Šimčík, M., Mota, A., Ruzicka, M., Vicente, A. & Teixeira, J. (2011). CFD simulation and experimental measurement of gas holdup and liquid interstitial velocity in internal loop airlift reactor. *Chemical Engineering Science, 66*(14), 3268-3279.

Trallero, J., Sarica, C. & Brill, J. (1997). A study of oil-water flow patterns in horizontal pipes. *SPE Production & Facilities, 12*(03), 165-172.

Yang, W., Beck, M. & Byars, M. (1995). Electrical capacitance tomography—from design to applications. *Measurement and Control, 28*(9), 261-266.