International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 5, Issue 9, pp 82-85, September-2021

https://doi.org/10.47001/IRJIET/2021.509010

CFD Modelling of Dispersed Bubble Two-Phase Flow in a Concentric Annulus Pipe

[1]Nyong, O. E, [2]Fakorede, D, [3]Ifere M, [4]Bepaye, A, [5]Igbong, D. I, [6]Ebieto, C. E

[1,2,3,4,5]Department of Mechanical Engineering, Cross River University of Technology, Calabar, Nigeria

[6]Department of Mechanical Engineering, University of Port Harcourt, Nigeria

Corresponding Author: nyong.oku@crutech.edu.ng

***Abstract* - Simulations of two-phase (air and water) flow in a pipe are very relevant topics; however, with the increased understanding of multiphase flow in pipes, the application of computational fluid dynamics (CFD) in other complex flow geometries involved in oil & gas industries are becoming more common. The current paper attempts to study two-phase flow characteristics in the horizontal concentric annulus using the CFD approach. The model was simulated in a concentric annulus test section with an overall length of 10.8m and outer diameter (OD) of 0.0768m and inner diameter (ID) of 0.060m. The model predicted the liquid holdup, and flow regime for the dispersed bubble flow. The volume of fraction (VOF) multiphase model and turbulence models (Realizable k- ε) were implemented to understand the gas and liquid holdup scenarios for flow in the horizontal annulus.**

Keywords: Dispersed bubble flow, CFD, Void fraction, Multiphase flow, concentric annulus pipe.

I. INTRODUCTION

In the oil and gas industry, it's necessary to quantify and predict the flow characteristics for the multiphase, liquids, and gas flows that are present within production and processing pipelines. The chemical, oil & gas, and nuclear industries are regularly involved in the transportation of multiphase flows. Two-phase flows such as oil-gas or oil-water systems are encountered more often in the oil & gas, oil, and brewery industries. The different phases of inflow can distribute themselves into different spaces in the pipe; this spatial distribution is termed flow patterns or regimes.

Flow patterns or regimes have been known to be greatly influenced by the pipe geometry in terms of size, length, inclinations, etc. fluid properties and flow conditions[1-3].

An idea of the flow regimes present in the multiphase flow allows the engineer to achieve optimum construction of the pipeline and downstream processes to attain a reliable design. Extensive studies on experiments[4-6],mechanistic model[7, 8]and correlations[9, 10] for flow patterns and other parameters obtained at various pipe diameters are studied on two-phase gas-liquid flow are investigated in the literature.

The complexity of two-phase gas-liquid flow is due to the influence of multiple flow parameters. Among the major flow parameters that affect two-phase flow are void fraction and pressure drop[11, 12]. To better understand the complexities of two-phase flow, the knowledge of void fraction and pressure drop is required. In two-phase flow systems, void fraction characterization is critical.

Several studies have been carried out over the years to acquire knowledge regarding flow behaviours of two-phase gas-liquid flow in annulus channels.

In the early 90's Xiao et al. [13] developed a comprehensive mechanistic model that can detect existing flow patterns and predict flow characteristics such as liquid holdup and pressure drop for stratified, intermittent, annular, and dispersed bubble flow patterns.

A unified steady-state two-phase flow mechanistic model was developed by Gomez et al. [14] to predict flow pattern, liquid holdup, and pressure drop for a wide range of inclination angles, from horizontal to upward vertical flow, for a variety of inclination angles.

Friedemann, et al. [15]worked on two-phase flow simulations at 0^0- 4^0 inclination in an Eccentric Annulus. They investigated co-current two-phase simulations of gas-liquid flow having mixture velocities of 1.2 - 4.2m/s in a partially eccentric annulus. The VOF model with k-ω (k-omega) turbulence model was utilized in OpenFOAM solver. Flow regimes were observed in the horizontal cases were wavy and slug flow was predominant.

Kiran, et al. [16]worked on an experiment and CFD modelling of two-phase flow in a vertical annulus, the flow comprises air and water flow having an annulus section of 82.5mm outer diameter, and 35mm inner diameter, the height of the annulus section was 5.5m. Their results show that the churn and annular flow regimes were encountered during the experiment, also at high superficial gas velocities, mainly

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 5, Issue 9, pp 82-85, September-2021

https://doi.org/10.47001/IRJIET/2021.509010

annular flow regimes existed. Their model was compared to the experimental result. Similar results in terms of flow patterns were observed. They concluded from their CFD study that SST κ-ω (k-omega) is better at numerically modelling two-phase flow compared to κ-ε (k-epsilon) turbulence flow.

CFD is one of the most widely used methods for understanding and analyzing flow characteristics in complex geometries. Although two-phase flow in the annulus has been studied analytically and experimentally, few studies have used CFD. The main objective is to investigate the dispersed bubble multiphase flow pattern in concentric annulus pipes.

II. COMPUTATIONAL METHODS

The geometry is created using the ANSYS 19 Design modeller[17, 18]. The geometry domain is made up of two inlets pipes forming a T-junction at the beginning of the pipe. The air and water inlets are 0.0768m in diameter, with a 2m long section preceding the annulus section to allow both fluids to mix[5]. In this case, the annulus section has an outer diameter of 0.076 and an inner diameter of 0.016. The overall length of the geometry is 12.85m[5].

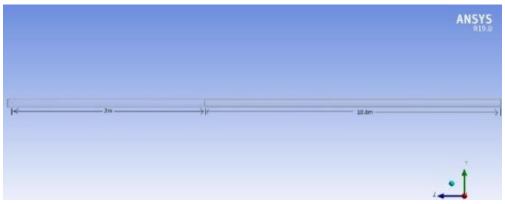

Figure 1: Geometry of the computational flow domain

For the mesh study, four meshes were investigated in this present study. The prism meshes were generated and used with the flow conditions of the superficial liquid velocity (U_{SL}) of water at 1.94 m/s and superficial gas velocity (U_{SG}) of air at 0.18m/s. Refinements were made at the annulus section of the geometry. The mesh sensitivity study was performed with mesh sizes in the range of 90000 and 305000 cells. An examination of the pressure gradient obtained from the CFD calculations of each mesh shows that an increase in mesh sizing at the annulus section reduces the pressure gradient, thus impacting remarkably on the results. Therefore, it can be concluded that a mesh of approximately 135000 cells was suitable to simulate without any changes in the result. The fluids understudy was treated as Newtonian fluids. The main governing equation is the Navier-Stokes momentum equation, which describes the motion of the Newtonian fluid. There is a linear relationship between viscosity and shear stress in these fluids. The Navier-Stokes momentum equation can be written in the form of the equation of continuum motion:

$$\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \bullet (\rho \boldsymbol{U}\boldsymbol{U}) = \nabla \bullet \sigma + \mathbf{f} (1)$$

Where ρ is the density of the cells, σ *is* the stress tensor, **f** the external forces and **U** is the velocity field. Neglecting compressibility effect due to low air velocities, therefore the densities are considered constant. The continuity equation, since fluids are incompressible:

$$\nabla \bullet \mathbf{U} = 0 (2)$$

Equations defining fluid properties such as dynamic viscosity and density, and the variation of the water volume fraction (α) with time and position are given by;

$$\rho = \alpha \rho l + (1 - \alpha) \rho g \quad (3)$$

$$\mu = \alpha \mu l + (1 - \alpha) \mu g \quad (4)$$

$$\frac{\partial u}{\partial t} + \nabla \bullet (\sigma \boldsymbol{U}) = 0 \qquad (5)$$

Where l and g depict the liquid and gas phases respectively. In this study, the VOF method in computational fluid dynamics is utilized.

In this study, the k-epsilon (k-ε) turbulence model was used in the CFD to simulate the flow conditions. The model focuses on the mechanisms that affect the turbulent kinetic energy. An assumption in this model is that the turbulent viscosity possesses the same physical properties in all directions, that is the ratio between Reynolds stress and mean rate of deformations is the same in all directions.

For turbulent kinetic energy k;

$$\frac{\partial (\rho_m k)}{\partial t} + \nabla \bullet (\rho_m v_m \epsilon) = \nabla \bullet \frac{\partial (\mu t_i m)}{\sigma_k} \nabla k + G_k \mathrm{m} - \rho_m \epsilon \quad (6)$$

For dissipation rate ϵ;

$$\frac{\partial (\rho_m \epsilon)}{\partial t} + \nabla \bullet (\rho_m v_m \epsilon) = \nabla \bullet \frac{\partial (\mu t_i m)}{\sigma_\epsilon} \nabla \epsilon + \frac{\epsilon}{k} (C_{1\epsilon} G_k \mathrm{m} - C_{2\epsilon} \rho_m \epsilon) \quad (7)$$

The pipe roughness height was considered to be 0.000015m for the wall, while the roughness constant was taken as 0.5. The turbulence was specified in terms of intensity and hydraulic diameter. The intensity was assumed to be 5%, while the hydraulic diameter was 0.0168m in this case. At the outlet, pressure boundary conditions were implemented. A pressure-based solver was chosen from the options offered in the Ansys fluent 19 packages, where a finite volume methodology is used to discretize the governing equations. Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) scheme was utilized for pressure-velocity coupling calculations. The fluid flow domain was subjected to boundary conditions such as velocity inlet, pressure outflow, and no-slip conditions. The inlet boundary condition was used to flow the multiphase fluid at a specific velocity and volume fraction, resulting in a stabilized flow along the length of the pipe. At the outlet section, the outlet boundary condition was used. There was no slip condition applied to the wall, and the

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 5, Issue 9, pp 82-85, September-2021

https://doi.org/10.47001/IRJIET/2021.509010

roughness constant was kept constant at 0.5. The momentum, turbulent kinetic energy, and turbulence dissipation rate were solved using the first-order scheme.

III. RESULTS AND DISCUSSION

The simulation's result is presented in this section. To initialize the inlet flow conditions, the standard initialization method was used. Following initialization, the liquid phase was patched across the entire flow domain. Each time step was 0.001 seconds, and 100 iterations were allowed to ensure that each step met the convergence criteria. Concerning flow time, parameters such as volumetric average pressure, and liquid holdup were monitored. The monitoring of these parameters was essential to ensure the complete development of the flow in the relevant section. It was inferred that the water volume fraction stabilizes and attains constant values during a fully developed flow.

3.1 Modelling the dispersed bubble flow

Here, operating conditions refers to pipe configuration, superficial velocities and fluid properties. For this case, the dispersed bubble flow regime was investigated through the annulus section of the computational domain. The dispersed bubble flow regime was observed at $U_{SG} = 0.18m/s$ and $U_{SL} = 1.94m/s$. Small Gas bubbles are dispersed in a continuous liquid phase initially concentrated at the top of the annulus cross-section. The gas plugs are formed during the breakdown of the elongated bubble or slug flow regimes. To further characterize the flow regime undergone for this case, a combination of time series and Probability Density Function (PDF) plots of the liquid holdup was obtained from the ANSYS fluent solver by capturing the area-weighted average liquid holdup fraction distribution. The probability density function was used to characterize the flow regimes and the calculated liquid holdup fraction in the spatial domains as shown in Fig 2. The kernel smoothing density function of MATLAB was utilized to obtain the Probability Density Function for the CFD multiphase flow. The kernel smoothing function has been used by several studies involving horizontal and vertical annulus configuration for two-phase flow [19].

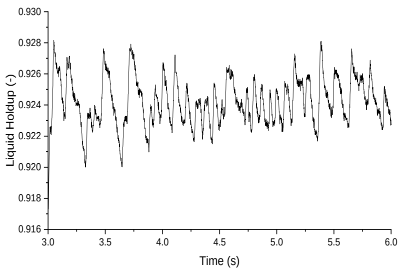

Figure 2: Shows the simulated liquid holdup fraction plotted against the time series for this case

The time-varying liquid holdup is seen to vary at a value close to 1, indicating the presence of spherical bubbles dispersed along with a continuous liquid phase. The PDF plot is unimodal having its peak value around 0.92 as shown in Fig. 2.

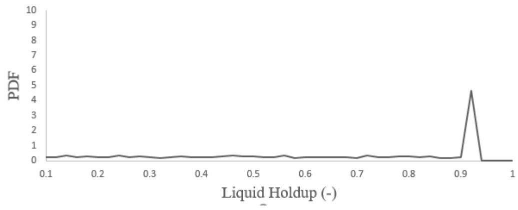

Figure 3: PDF vs Simulated Liquid holdup fraction trend for the dispersed bubble flow at $U_{SG} = 0.18m/s$ and$U_{SL} = 1.94m/s$

Fig. 4 shows the contour of the volume fraction along the annulus section. This shows that the annulus section is occupied by mostly the liquid phase as both fluids phases mix along the annulus section; several dispersed bubbles are seen to be located more at the top of the annulus section.

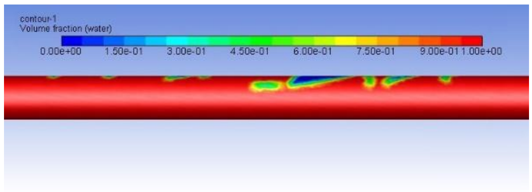

Figure 4: Contour of Volume fraction of water along the annulus section for the dispersed bubble flow

The results indicate that the dispersed bubble flow regime is detected, as shown in the contour of volume fraction of water in Fig. 4, several dispersed bubbles are seen to be vastly located at the top of the annulus section, and these bubbles are formed from the breaking of the slug and elongated bubbles moving along the annulus section. High values of the liquid holdup are present along the annulus section.

IV. CONCLUSION

This paper presents CFD simulation studies on two-phase flow characteristics in a concentric annulus at $U_{SG} = 0.18m/s$ and$U_{SL} = 1.94m/s$. The Realizable k-ε turbulence model was used in this study coupled with VOF to predict flow regimes and liquid hold up in the annulus pipe. Furthermore, the PDF were generated to identify the flow regimes.

At liquid gas superficial velocities, the dispersed bubble flow regime was observed. Tiny gas bubbles are dispersed in a continuous liquid phase that begins near the top of the annulus cross section and becomes concentrated near the top. The breakdown of gas plugs during the elongated bubble or slug flow regimes results in the formation of this flow regime.

ISSN (online): 2581-3048
Volume 5, Issue 9, pp 82-85, September-2021
https://doi.org/10.47001/IRJIET/2021.509010

ACKNOWLEDGEMENT

Thanks to The Federal Government of Nigeria through the office of the Tertiary Educational Trust Fund (TETFUND) for sponsoring this research through the Institutional Based Research (IBR) grant. A final thanks to the Department of Mechanical Engineering, Cross River University of Technology where this research was done.

REFERENCES

[1] H. Cheng, J.H. Hills, B.J. Azzorpardi. A study of the bubble-to-slug transition in vertical gas-liquid flow in columns of different diameter. International Journal of Multiphase Flow. 24 (1998) 431-52.

[2] Y. Taitel, D. Bornea, A.J.A.J. Dukler. Modelling flow pattern transitions for steady upward gas- liquid flow in vertical tubes. 26 (1980) 345-54.

[3] D. Barnea, O. Shoham, Y. Taitel, A.E. Dukler. Flow pattern transition for gas-liquid flow in horizontal and inclined pipes. Comparison of experimental data with theory. International Journal of Multiphase Flow. 6 (1980) 217-25.

[4] G. Das, P. Das, N. Purohit, A. Mitra. Flow pattern transition during gas liquid upflow through vertical concentric annuli—part I: experimental investigations. (1999).

[5] E.N. Eyo, L. Lao. Gas-liquid flow regimes in horizontal annulus. Journal of Petroleum Science and Engineering. 175 (2019) 573-86.

[6] A.S. Izwan Ismail, I. Ismail, M. Zoveidavianpoor, R. Mohsin, A. Piroozian, M.S. Misnan, et al. Experimental investigation of oil–water two-phase flow in horizontal pipes: Pressure losses, liquid holdup and flow patterns. Journal of Petroleum Science and Engineering. 127 (2015) 409-20.

[7] J.M. Mandhane, G.A. Gregory, K. Aziz. A flow pattern map for gas—liquid flow in horizontal pipes. International Journal of Multiphase Flow. 1 (1974) 537-53.

[8] V. Jagan, A. Satheesh. Experimental studies on two phase flow patterns of air–water mixture in a pipe with different orientations. Flow Measurement and Instrumentation. 52 (2016) 170-9.

[9] L. Pan, T.J.J.I.j.o.m.f. Hanratty. Correlation of entrainment for annular flow in horizontal pipes. 28 (2002) 385-408.

[10] S. Rassame, T.J.I.J.o.H. Hibiki, F. Flow. Drift-flux correlation for gas-liquid two-phase flow in a horizontal pipe. 69 (2018) 33-42.

[11] C.C. Tang, S. Tiwari, A.J. Ghajar. Effect of void fraction on pressure drop in upward vertical two-phase gas-liquid pipe flow. Journal of engineering for gas turbines and power. 135 (2013).

[12] O.M.H. Rodriguez, L.S. Baldani. Prediction of pressure gradient and holdup in wavy stratified liquid–liquid inclined pipe flow. Journal of Petroleum Science and Engineering. 96-97 (2012) 140-51.

[13] J. Xiao, O. Shonham, J. Brill. A comprehensive mechanistic model for two-phase flow in pipelines. SPE Annual Technical Conference and Exhibition. OnePetro1990.

[14] L. Gomez, O. Shoham, Z. Schmidt, R. Chokshi, T.J.S.j. Northug. Unified mechanistic model for steady-state two-phase flow: horizontal to vertical upward flow. 5 (2000) 339-50.

[15] C. Friedemann, M. Mortensen, J. Nossen. Two-phase flow simulations at 0− 4o inclination in an eccentric annulus. International Journal of Heat and Fluid Flow. 83 (2020) 108586.

[16] R. Kiran, R. Ahmed, S. Salehi. Experiments and CFD modelling for two phase flow in a vertical annulus. 153 (2020) 201-11.

[17] ANSYS FLUENT 14 Theory Guide. (2011).

[18] A.J.A.R.R. Fluent. Ansys fluent. 14 (2015).

[19] R. Kiran, R. Ahmed, S. Salehi. Experiments and CFD modelling for two phase flow in a vertical annulus. Chemical Engineering Research and Design. 153 (2020) 201-11.

Citation of this Article:

Nyong, O. E, Fakorede, D, Ifere M, Bepaye, A, Igbong, D. I, Ebieto, C. E, "CFD Modelling of Dispersed Bubble Two-Phase Flow in a Concentric Annulus Pipe" Published in *International Research Journal of Innovations in Engineering and Technology - IRJIET*, Volume 5, Issue 9, pp 82-85, September 2021. Article DOI https://doi.org/10.47001/IRJIET/2021.509010

****

