4ème Conférence Internationale des Energies Renouvelables (CIER-2016)
Proceedings of Engineering and Technology – PET
Vol.17, pp.8-14

On two-phase flow patternsprediction using CFD modeling in horizontal pipe: validation

I.BELGACEM[1], E.K. SI-AHMED[1,2], Y SALHI[1]

[1] *USTHB- LMFTA Faculty of Physics B.P. 32, El-Alia, 16111, Algiers, Algeria*
[2]*LUNAM, GEPEA, CNRS, UMR-6144,CRTT 37, Bd de l'Université BP406 St-Nazaire 44602 France*

Abstract

Gas-liquid two-phase flow occurs in a wide range of engineering applications in various industrial fields. For example, in the chemical and process industry two-phase flow is encountered in such facilities as boilers, condensers, evaporators and reactors. In the petroleum industry, two-phase flow is observed during the production and transportation of oil and gas.
Unlike single-phase, two-phase flows are classified according the phenomenological description of the geometric distributions of flow patterns. In this work experimental and numerical methods are applied to study two-phase flow in horizontal pipe. Specifically, the study aims to predict the flow patterns numerically. For that purpose numerical methods include the use of two-phase computational dynamics CFD calculations based on VOF method were used to identify the flow patterns. The comparison between Belgacem et al (2015)published experimental results and CFD prediction of two-phase flow patterns shows that all horizontal flow regime appearing in the test rig can thus be calculated using CFD code, after validation a detailed numerical study was performed to generate more information about the hydrodynamic behavior of the two phase flow.
Key words: *two-phase flow, flow pattern, horizontal pipe, CFD, VOF.*

Introduction

Liquid-gas two-phase flows are widely encountered in industrial applications including chemical processes, petroleum engineering and energy manufacturing units systems. Unlike single-phase flows, two-phase flows are classified according to the phenomenological description of the flow geometric patterns distributions. The latters in horizontal tubes are influenced by several factors including gravity that acts to stratify the liquid to the bottom of the tube and gas to the top. Two-phase flow patterns for co-currents flow are, on the overall, recognized as stratified, wavy, intermittent (slug/plug elongated bubbles) bubbly flow, annular and mist flow. Two-phase patterns for co-currents flow are show in figure 1.

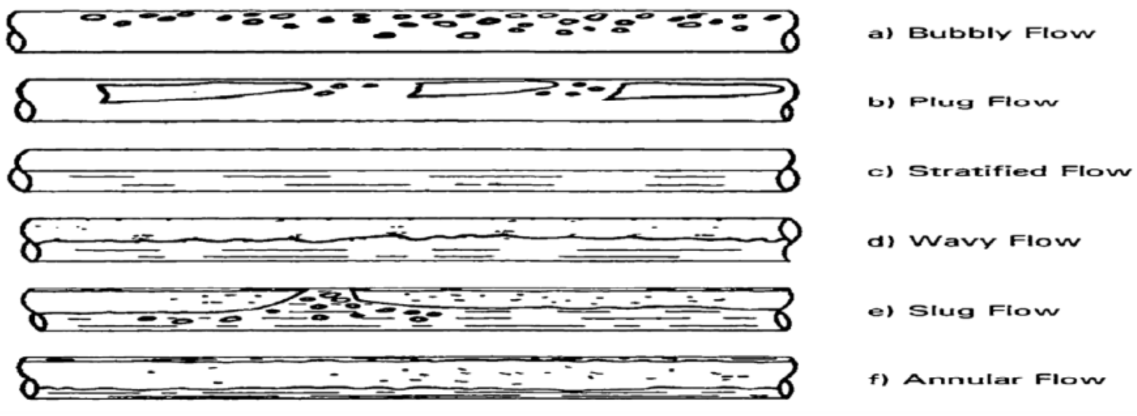

Figure 1.Some flow patterns in horizontal pipe.

Generally, a flow regime map, based on experimental data, is employed to envisage the structure of the flow.

ISSN 2356-5608

Nevertheless, the limits of each structure or mode are not well established and there is no consensus among the recorded published results. Several parameters related to the conditions of entry can affect the structure of flow. Moreover, a flow map is often specific to pipe geometries and fluids considered. A number of investigators (Baker 1954, Schicht 1969) have attempted to find generalized coordinates that would allow the map to cover different fluids and pipes. However, such generalizations can only have limited value because several transitions are represented in most flow pattern maps and the corresponding instabilities are governed by different sets of fluid properties. Several works were proposed in the last six decades for the prediction of flow pattern in pipeline; however, none of them gives consistently reliable results for all the identified flow patterns in two-phase flow. Kosterin (1949) was probably the first to suggest the use of flow pattern map. The first map due to Baker (1954) was based on the experimental work of Kosterin (1949) and Alves (1954). In these chart seven different regimes were distinguished: stratified flow, wavy flow, plug flow, slug flow, bubble flow, spray or dispersed flow. The Baker flow regime map shows the boundaries of the various flow regimes as functions mainly of mass fluxes of both phases and other parameters including the liquid viscosity. Hoogendoor (1959) took into account the effects of the diameter and the properties of the fluid like viscosity, density and the surface tension in addition of the influence of the roughness in the transition between various patterns. Two-phase flow of natural gas-water, natural gas-crude oil, and natural gas-distillate water in horizontal pipes with inner diameter of 5.08cm and 10.16cm was studied by Eaten et al (1967). The authors represented the data in a flow pattern map as function as a two-phase Reynolds number and a two-phase Weber number.

Mandhane et al (1974) collected more than 6000 data points for air-water system, a total of 1178 points were selected in creating a two-phase flow map based on the superficial liquid and gas velocities. However, fluid properties, diameter, and inclination specify the flow map which applies. Nonetheless, the Mandhane et al (1974) chart has since become a standard format for publishing flow regime data in multiphase flow. One should note the appreciable range in the parameters values in the Mandhane’s map validity.

Taitel and Dukler (1976) considered a flow map for concurrent gas-liquid in horizontal pipe on the basis of the analytical analysis of the flow transition, defined by several dimensionless parameters including ratio of both phases’ pressure gradients, pipe inclination superficial velocities as well as Reynolds number. The flow patterns defined in their study were stratified, wavy, annular and intermittent which includes the elongated bubbles as well as slug flow.

Other authors such as Breber (1980) proposed a plot of the Martinelli number against a modified gas superficial velocity. The consistency between the observations and the predictions of the Breber map was verified by the experimental study of Ewing et al (1999).

De Schepper et al.(2008) show that CFD codes are able to calculate the different horizontal two-phase gas-liquid and vapor-liquid flow regimes as predicted by the Baker chart.

Thepresentworkisdevoted to investigate air-water co-currenttwo-phase flow behaviorresulting in horizontal pipe. Computationalfluiddynamic CFD calculationusing VOF techniques are employed to generate gas liquid pattern,The numericalresults are validatedagainstexperimental data from the literature and arefound to be in good agreement.

2.NUMERICAL PROCEDURE

2.1. Mathematical model

For the mathematical model, Eulerian based volume of fluid VOF technique for two phase modeling were employed to investigate the two phase pattern in horizontal pipe. In this model, liquid is considered to be the continuous and primary phase, and gas considered to be the dispersed and secondary phase. The fluid in both phases is Newtonian, viscous and incompressible. The uniform pressure field is assumed to be shared by both phases, the flow is considered isothermal so the energy equations are not needed.

The VOF method has the advantages of high precision, and traces the volume of fluid in the grid, not the motion of fluid particles. In the VOF model, a single set of momentum equations is shared by the fluids, and the fluid volume fraction in each computational cell is tracked throughout the domain. This model has been found to be suitable for simulating interface among two or more fluids (Ghorai et al, 2006).

ISSN 2356-5608

The VOF method utilizes the volume fraction , which means the fraction of the filled fluid volume in the grid to achieve the goal. The indicator function is defined as 0 for a cell with pure gas, 1 for a cell with pure liquid, and for a cell with a mixture of gas and liquid. An interface exists in those cells that give a volume of fluid value of neither 0 nor 1. Since the indicator function is not explicitly associated with a particular front grid, an algorithm is needed to reconstruct the interface (Hirt and Nichols, 1981):

$$\alpha = \begin{cases} 0 & \text{in pur gas} \\ 0 < \alpha < 1 & \text{gas} - \text{liquid interface} \\ 1 & \text{in pur liquid} \end{cases} \quad [1]$$

2.2.Governing equations

Numerical simulation of any flow problem is based on solving the basic flow equations describing turbulence, continuity and momentum. The principal equations are solved for each phase and can be written as follow:

Continuity equation

$$\frac{\partial(\alpha\rho)}{\partial t} + \nabla.(\alpha\rho\vec{v}) = 0 \qquad [2]$$

Momentum equation

$$\frac{\partial(\alpha\rho\vec{v})}{\partial t} + \nabla.(\alpha\rho\vec{v}\vec{v}) = -\alpha\nabla p + \alpha\nabla.[\mu(\nabla\vec{v} + \nabla\vec{v}^T)] + \alpha\rho\vec{g} + \alpha\vec{F} \ [3]$$

The void fraction α is the void fraction of water or liquid phase.

Turbulent model

The Reynolds Stress Model (RSM) is a higher level, elaborate turbulence model. It is usually called a Second Order Closure. This modeling approach originates from the work by Launder 1975, in RSM, the eddy viscosity approach has been discarded and the Reynolds stress is directly computed. The modelcan be used to predict the turbulent anisotropic level in the flow. Given that the two-phase flows are very unstable and highly anisotropic.

$$\begin{aligned} \frac{\partial(\rho\bar{\alpha}\tilde{R}_{ij})}{\partial t} + \frac{\partial}{\partial \mathrm{x_k}}\left(\rho\bar{\alpha}\widetilde{\boldsymbol{U}_k}\tilde{R}_{ij}\right) \\ = -\rho\bar{\alpha}[\tilde{R}_{ij}(\nabla\widetilde{\boldsymbol{U}_k})^T + (\nabla\widetilde{\boldsymbol{U}_k})\tilde{R}_{ij}] + \frac{\partial}{\partial x_k}[\bar{\alpha}\mu\frac{\partial\tilde{R}_{ij}}{\partial x_k}] - \frac{\partial}{\partial x_k}[\rho\bar{\alpha}\overline{\mathrm{u_i'}\mathrm{u_j'}\mathrm{u_k'}}] \\ + \bar{\alpha}p\overline{\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i}\right)} - \bar{\alpha}\rho\,\tilde{\varepsilon}_{ij} + \frac{2}{3}\delta_{ik}\Pi_k \qquad [4] \end{aligned}$$

2.3. Numerical procedure

The experimental geometry with and without contraction has been modeled using an axi-symmetric 2D geometry. The simulation was performed using the commercial CFD code Fluent 6.3.26 at double precision solver mode, with an implicit scheme for all variables and a fixed time step t= 0.001 s for computation. To solve the momentum transport equation the Quick (quadratic upwind interpolation) scheme was used, for pressure the PRESTO (PREssureSTaggering Option) scheme increases stability in the solution. The phase – coupled PISO (Issa (1986)) algorithm is used for the pressure –velocity

ISSN 2356-5608

coupling. RSM model has been used for turbulent two phase-flows. These schemes ensured, in general, satisfactory accuracy, stability and convergence. In addition, the steady-state solution strategy was employed. Meshing the geometry was achieved by using a software GAMBIT (2.4.6). We used the quadratic elements and the dimension of each cell is 0.004 making the number of cells equal to 842 205. The convergence criterion is decided based on the residual value of the calculated variables, namely mass, velocity components and pressure. In the present study, the numerical computation is considered converged when the residuals of the different variables are lowered by five orders of magnitude.

Inlet boundary: For both geometries the velocity of the fluids is specified at the inlet.

Outlet boundary condition: At the outlet, pressure outlet boundary is used.

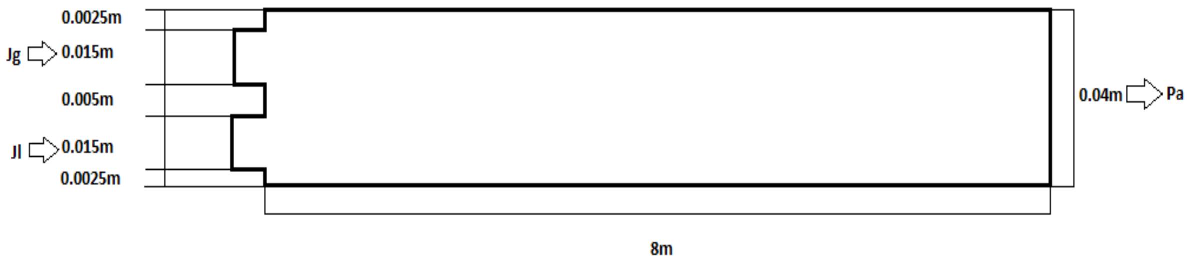

Figure 2.Computational domain and boundary conditions for contraction section.

3. Results and discussions

The mean goal of the simulation work involves 2D simulation of the two-phase flow patterns of water and air two-phase system in horizontal pipe. The CFD modeling was carried out at four different inlet conditions to model the flow patterns detected by Belgacem et al 2015.

In order to validate the simulation, the results are compared with the experimental data reported in the first section of this paper. Figure represent a few representative simulations of flow distribution, the blue and red colors indicate the gas and water respectively with their corresponding photographs as obtained from experiments. Figure 3 depict a reasonable matching between the simulated and experimental flow pattern.

Table 1. Operating conditions for the CFD simulations of air water flow corresponding to the experimental Belgacem et al (2013).

Flow pattern	Stratified	Wavy	Slug	Elongated bubbles
Operating conditions	Jg=1m/s Jl=0,08m/s	Jg=4m/s Jl=0,09m/s	Jg=5m/s Jl=0,13m/s	Jg=2m/s Jl=0,2m/s

ISSN 2356-5608

4ème Conférence Internationale des Energies Renouvelables (CIER-2016)
Proceedings of Engineering and Technology – PET
Vol.17, pp.8-14

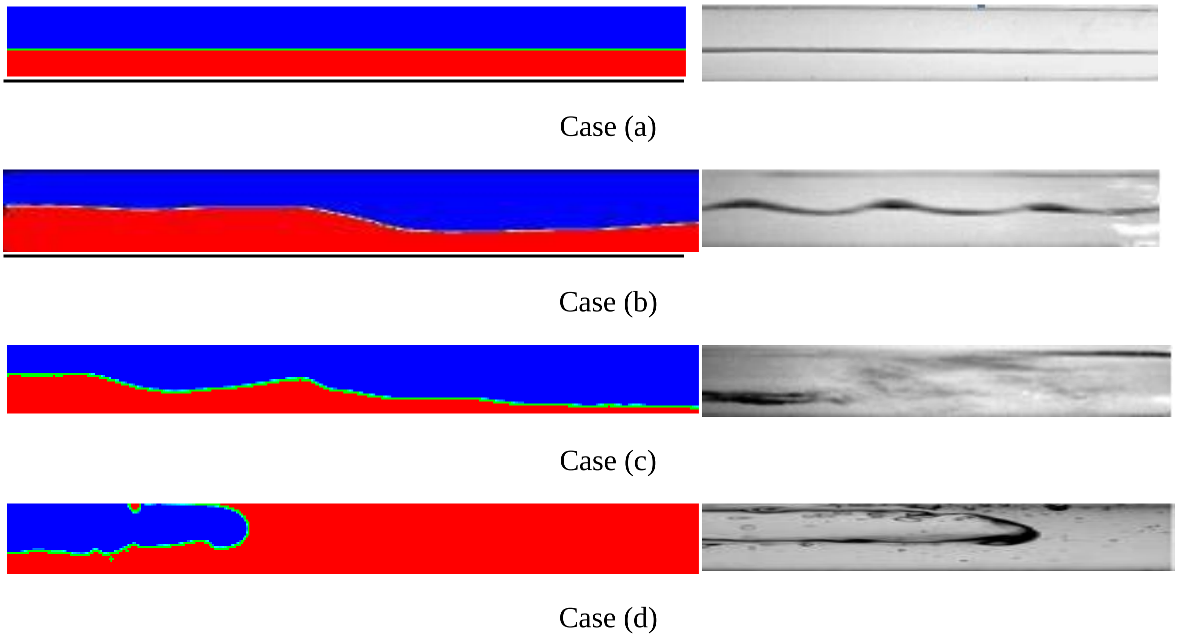

Figure.3. CFD- predicted compared with experimental results (Belgacem et al 2015) of two-phase flow in horizontal pipe.

4.Conclusion

Numerical investigation of two –phase flow in horizontal pipe has been analyzed in this paper using the computational fluid dynamics software (Fluent) applying the volume of fluid VOF method and the RSM model. The main conclusions can be summarized as follows:

1- After comparison between experimental prediction of two phase flow patterns and simulations, we find that the numerical model predict the flow patterns, it can be concluded that all flow patterns observed in the experimental setup can be simulated using CFD code.
2- The analysis shows that the model can be used to generate useful information of the hydrodynamics of two –phase flow in horizontal pipe.

References

Baker, O.1954. Simultaneous flow of oil and gas.Oil and Gas J.53, 185-195.

Schicht, H. H. (1969). Flow patterns for an adiabatic two phase flow of water and air within a horizontal tube. Verfahrenstechnik, 3, 153-161.

Kosterin, S.I., 1949. An Investigation of the Influence of the Diameter and Inclination of a Tube on the Hydraulic Resistance and Flow Structure of Gas-liquid Mixtures.Izvest.Akad.Nauk.S.S.S.R.OtdelTekh.Nauk. 12, 1824-1830

ISSN 2356-5608

Alves, G.E., 1954. Co-current Liquid Gas Flow in a Pipeline Contractor.Chem. Engng. Prog.50, 449-456.

Hoogendoor, C.J. 1959. Gas-Liquid flow in horizontal pipes, Chem. Engng Sci. 9, 205-217.

Eaton, B., Knowles, C., and Silberbrg, I. H. (1967). The prediction of flow patterns, liquid holdup and pressure losses occurring during continuous two-phase flow in horizontal pipelines. Journal of Petroleum Technology, 19(6), 815-828.

Mandhane, J.M., Gregory, G.A. and Aziz, K.1974.A flow pattern map for gas–liquid flow in horizontal pipes.Int.J.Multiphase flow 1,537-553.

Taitel, Y. and Dukler, A.E., 1976a. A model for predicting flow regime transitions in horizontal and near horizontal gas-liquid flow, Aiche Journal,22,47-55.

Breber, G., Palen, J. W., and Taborek, J. (1980).Prediction of horizontal tubeside condensation of pure components using flow regime criteria. Journal of Heat Transfer, 102, 471.

Ewing, M. E., Weinandy, J. J., and Christensen, R. N. (1999).Observations of two-phase flow patterns in a horizontal circular channel. Heat transfer engineering, 20(1), 9-14.

De Schepper, S. C., Heynderickx, G. J., & Marin, G. B. (2008).CFD modeling of all gas–liquid and vapor–liquid flow regimes predicted by the Baker chart. Chemical Engineering Journal, 138(1), 349-357.

Ghorai, S., Nigam ,K.D.P.(2006). CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes, Chemical engineering and processing journal, 45 (1), 55-65.

Hirt, C. W., and Nichols, B. D. (1981).Volume of fluid (VOF) method for the dynamics of free boundaries.Journal of computational physics, 39(1), 201-225.

Launder, B.E, Reece, G.J. and Rodi, W. 1975. Progress in the development of a Reynolds-Stress Turbulent closure.Journal of fluid Mechanics, 68(3), 537-566.

Issa, R.I., 1986. Solution of implicitly discretized fluid flow equations by operator splitting.J.Comput.Phys.62, 40-65.

Belgacem, I., Salhi, Y., SI-Ahmed, E. K., Legrand, J., &Rosant, J. M. Experimental investigation of slug pattern in a horizontal two-phase flow. Wit transaction on Engineering Sciences, Vol 79, WIT Transactions on Engineering Sciences, Vol 79, Wit Press, 2013,ISSN 1743-3533, doi:10.2495/MPF130351