Numerical investigations on gas-liquid distribution characteristics of intermittent flows in a pipeline-riser system

Mengchen Gao, Jingyu Xu*, Yingxiang Wu, Jun Guo

LMFS Laboratory, Institute of Mechanics, Chinese Academy of Sciences
Beijing
* E-mail: xujingyu@imech.ac.cn

ABSTRACT: In this work, VOF model has been introduced to simulate gas-liquid two phase intermittent flow phenomena in a pipeline-riser system. During the flow processing, air and water has been set as the fluid medium, and various operation conditions have been tested in details. As been shown in results, there are mainly air bubbles and slugs in the riser section under different operation conditions. In addition, various fluid medium has been tested for comparison, which will have a positive impact on further research.

KEY WORDS: Pipeline-riser system; gas-liquid two phase flows; VOF model; computational fluid dynamics.

INTRODUCTION

In offshore oil processing, the structure of pipeline-riser system will be widely used during the process of pipeline transportation. Due to the variation of inlet conditions and pipeline fluctuation affected by terrain factors, gas-liquid two phase intermittent flows will occur during the flow process in pipeline-riser system. This kind of flow pattern can cause lots of significant phenomena which can be badly influence the production, such as pressure fluctuation, flow rate variation, frequency coupling between internal and external flow field. As a result, the wellhead production is badly reduced as the flow state changes periodically.

Based on the research of former scholars, there are four stages divided from one periodical flow statement, as slug formation, slug production, blowout, and liquid fallback [1]. During the stage of slug formation, liquid flows into the base of the riser which blocked the gas passes by. On the second stage, called slug production, a large number of liquid rises from the riser base with gas bubbles dispersed among it. Followed by the third stage, gas blows out rapidly and forms gas-liquid interment flows in the riser section. In this period, all kinds of flow characteristics, such as void fraction and pressure drops, are fully depended on the gas-liquid two phase flow patterns, which are important to be considered but hard to be investigated [2]. At last, liquid falls back as the gas phase velocity reduces to the minimum value and starts to a brand new cycle. As discussed above, when gas-liquid two phase flows occur in the structure of pipeline-riser system, it will effect badly on the ocean architecture safety, thus need to be careful consideration [3-5].

In order to simulate the process of gas-liquid two phase flows in a pipeline-riser system, computational fluid dynamics has been chosen as the research method using the large-scale commercial software FLUENT to investigate detail information during flow process. In one periodical flow cycle the stage of gas-liquid two phase flow has been captured through VOF model. Besides, different fluid medium have been changed for comparison. At beginning, we start from air-water two phase flows, and then air-oil/water emulsion cases are followed.

NUMERICAL METHOD

Physical Parameters

In this work, the liquid phase working medium was first set as water. The density was 998kg/m3, the viscosity was 0.00103kg/(m • s), and the initial temperature was 298K. The gas phase was set as air, and it was assumed to be the ideal compressible gas. Inlet mass flow conditions were taken both for liquid phase and gas phase. Pressure outlet boundary condition had been set as the outlet condition, and it was assumed that the pressure at the outlet was referred to standard atmospheric pressure. No slip solid wall boundary condition was set for pipe wall. At initial time, the pipeline-riser system was filled with gas; the liquid phase flowed into the pipeline-riser system at a fixed flow rate. Quadrilateral structured grid was used in geometric structure meshing with Gambit@2.3.16. Boundary layer effect existed near the wall surface, thus partial encryption was adapted in the near wall of grid.

Mathematical Model

By solving separate momentum equation, VOF model, a fixed Eulerian mesh surface tracking method, can be applied to simulate two or three kind of fluid which cannot be mixed up. In this work, VOF model is introduced to capture the fine gas-liquid fine interface. Assuming that the gas liquid two phases does not dissolve, the interphase interface is determined by solving the phase volume fraction of each control unit. In present study, the assumption is that the air is an ideal compressible gas, and water is incompressible fluid. Besides there is not mass transfer between two phases, so the continuous equation of the volume fraction is:

$$\frac{\partial}{\partial t}(\alpha_1\rho_1)+\nabla\cdot\left(\alpha_1\rho_1\vec{v_1}\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_2\rho_2)+\nabla\cdot\left(\alpha_2\rho_2\vec{v_2}\right)=0 \tag{2}$$

The subscript 1 represents the gas phase, and 2 for the liquid phase. For volume fraction of each phase, the sum of the volume fraction of each phase in each control unit is equal to 1, i.e., for the density, for the velocity vector.

In the whole calculation region, the mean Reynolds momentum equation and energy equation are shared between two phases:

$$\frac{\partial}{\partial t}\left(\rho\vec{v}\right)+\nabla\cdot\left(\rho\vec{v}\vec{v}\right)=-\nabla p+\nabla\cdot\left[\mu_{eff}\left(\nabla\vec{v}+\nabla\vec{v}^{T}\right)\right]+\nabla\cdot\left(-\rho\overline{\vec{v}'\vec{v}'}\right)+\rho\vec{g}+\vec{F} \tag{3}$$

$$\frac{\partial}{\partial t}(\rho E)+\nabla\cdot\left(\vec{v}(\rho E+p)\right)=\nabla\cdot\left(k_{eff}\nabla T\right)+\nabla\cdot\left(-\rho\overline{\vec{v}'E'}\right)+S_h \tag{4}$$

The physical properties of the equation (3) and the equation (4) are determined by gas-liquid two phases in each control volume:

$$\rho=\alpha_1\rho_1+\alpha_2\rho_2 \tag{5}$$

$$\mu_{eff}=\alpha_1\mu_1+\alpha_2\mu_2 \tag{6}$$

$$E=\frac{\alpha_1\rho_1E_1+\alpha_2\rho_2E_2}{\alpha_1\rho_1+\alpha_2\rho_2} \tag{7}$$

$$k_{eff}=\alpha_1k_{eff1}+\alpha_2k_{eff2} \tag{8}$$

Where, μ_{eff} is the effective viscosity coefficient, $\vec{g}$ is the gravity acceleration, $\vec{F}$ is the volume force, p is the pressure, E is the energy, T is the temperature, k_{eff} is the effective fluid heat transfer coefficient, and S_h is the viscous dissipation term.

RNG $k-\varepsilon$ model is introduced as the turbulence model, where turbulent kinetic energy k is defined by:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho v_i k)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(a_k\mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M \tag{10}$$

dissipation rate ε is defined by:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho v_i\varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(a_\varepsilon\mu_{eff}\frac{\partial\varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon \tag{11}$$

Where, G_k is the mean velocity gradient of turbulent kinetic energy, G_b is the turbulent kinetic energy generated by buoyancy, Y_M is the overall dissipation rate in compressible turbulent expansion, a_k and a_ε are the inverse of effective Prandtl number k and ε respectively, and $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants.

A key problem to be solved in numerical calculation is the surface tension of gas liquid two phase interface. Therefore, CSF model, established by Brackbill, has been introduced for numerical processing. In this model, assuming that two media boundary regions have a finite thickness, it makes α to be a continuous function at the entire transition region. Thus, the surface forces at each point are proportional to the curvature of the point. When the thickness of the interfacial region is close to zero, the surface tension can be expressed as the volume force which acts on the grid cell of the phase interface, and the solution is obtained through momentum equation:

$$\vec{F}=\sigma_{12}\frac{2\rho\kappa\nabla\alpha_1}{\rho_1+\rho_2} \tag{12}$$

Where, σ_{12} is the surface tension between gas-liquid two phase, α_1 is the volume fraction for gas phase, $\kappa=\nabla\cdot\hat{n}$ is the surface curvature, $\hat{n}=\vec{n}/|\vec{n}|$ is the unit normal vector, $\vec{n}=\nabla\alpha_1$ is the normal vector at surface. Near wall region, the unit normal vector can be calculated by $\hat{n}=\hat{n}_w\cos\theta_w+\hat{t}_w\sin\theta_w$. Where, θ_w is the contact angle between liquid phase and wall surface, $\hat{n}_w$ and $\hat{t}_w$ represents the unit normal vector and the unit tangent vector of the wall surface, respectively.

Computing Method

The main idea of finite volume method is to divide the calculated region into a series of control volume, and each control volume has a node as a representative. Flow field near the wall can be treated as standard wall function, thus a second-order upwind scheme is applied for convective terms and turbulent transport terms. Furthermore, geometric reconstruction scheme has been chosen for gas-liquid two phase interface. As one of the most accurate gas-liquid interface solution, this kind of scheme uses piecewise method to depict the interface between fluid phases.

RESULTS AND DISCUSSION

Air-Water Flows

Fig. 1 shows phase volume fraction distributions of riser under four extreme conditions, where (a) corresponded to minimum velocity both for gas and liquid phase, (b) was the case of the maximum gas phase velocity and minimum liquid phase velocity, (c) was for maximum liquid velocity and minimum gas phase velocity, and (d) was for the case of maximum phase velocity both for liquid phase and gas phase. It can be seen from results that when the gas phase and the liquid phase flow rate were minimum, it corresponded to slug flows referred to intermittent flow pattern. When the liquid flow rate arrived at minimum value, and gas velocity arrived at maximum value, the liquid distribution of riser section was more dispersed, called churn flows. Bubble flows occurred at maximum liquid velocity and minimum gas phase velocity. Slug flows occurred at maximum two phase velocity, but different from the first case, much more dispersed small gas bubbles accompanied.

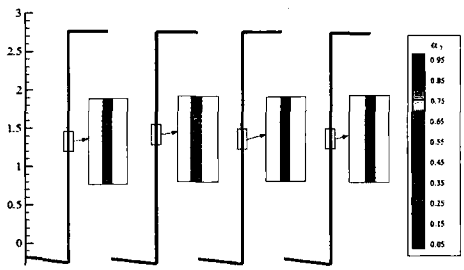

Fig. 1 Gas-liquid two phase flow statements in pipeline-riser system

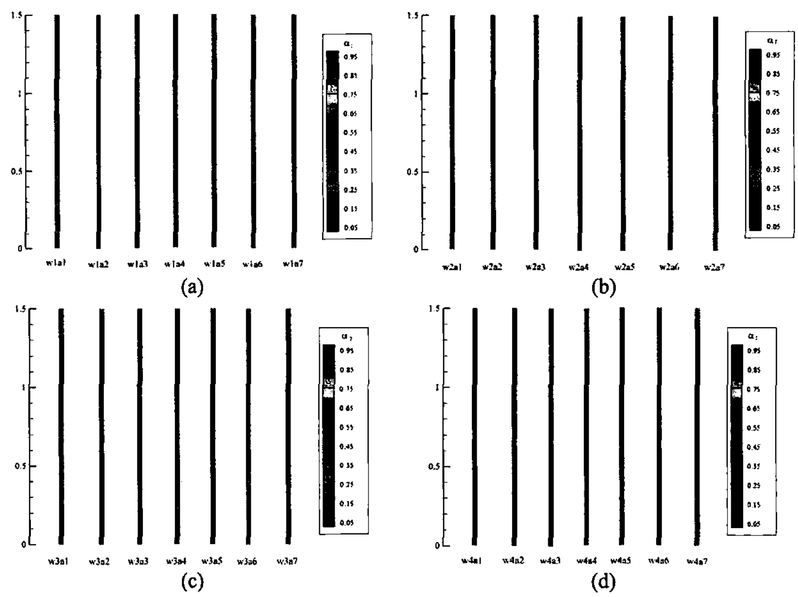

Fig. 2 Flow patterns of air-water intermittent flows under different operation conditions

Fig. 2 shows gas-liquid interface characteristics under different gas-liquid flow rate ratio in gas-water two phase flows. At minimum velocity values, it can be seen that there were gas slugs as well as part of small bubbles. With liquid phase and gas phase velocity increased, the liquid distribution of riser was more dispersed. When the liquid flow rate maintained minimum value, small gas bubbles occurred in the riser section, referred to bubble flows. When the liquid flow rate increased, the flow pattern came to more close to dispersed bubble flows.

Gas-Oil/Water Emulsion Flows

Based on field experience, the rate of water content for output liquid is about 80%. Therefore, this section will use this proportion for emulsion to conduct numerical investigation. Fig. 3 shows the flow state of gas-oil/water emulsion flows in the riser under different operation conditions. It can be seen from results, there were slightly differences from cases under air-water flows. However, due to different fluid properties, it will present significant change in other flow characteristics, pressure drops for example.

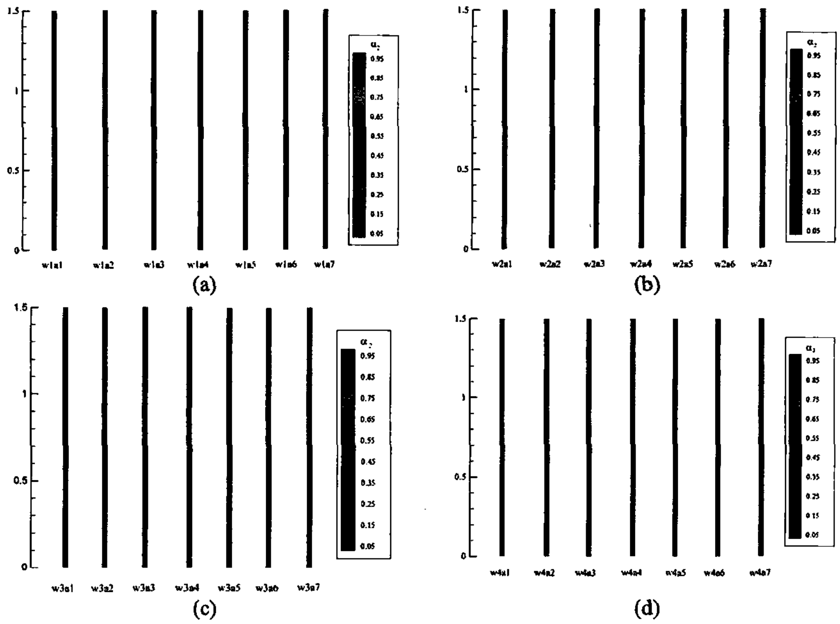

Fig. 3 Flow patterns of gas-oil/water emulsion intermittent flows under different operation conditions

CONCLUSION REMARKS

In this work, combined with consideration of surface tension, VOF model has been introduced successfully to simulate gas-liquid two phase intermittent flows in a pipeline-riser system. It can be seen from results that gas phase in riser section were mainly characterized by bubbles and slugs under operation conditions. With gas-liquid flow condition change, there were different flow patterns in the riser section. Moreover, different liquid medium affected slightly on gas-liquid two phase distributions, but changed significantly in pressure drops.

REFERENCES

[1] Schmidt Z., Brill J. and Beggs H. Experimental study of severe slugging in a two phase-flow pipeline–riser pipe system[J]. Society of Petroleum Engineers Journal, 1980, 20(5): 407-414

[2] Van Nimwegen A. T., Portela L. M. and Henkes R. A. W. M. The effect of surfactants on air–water annular and churn flow in vertical pipes. Part 1: Morphology of the air–water interface[J]. International Journal of Multiphase Flow, 2015, 71: 133-145.

[3] Luo X. M., He L. M. and Ma H. W. Flow pattern and pressure fluctuation of severe slugging in pipeline-riser system[J]. Chinese Journal of Chemical Engineering, 2011, 19(1): 26-32.

[4] Li N. L., Guo L. J. and Li W. S. Gas–liquid two-phase flow patterns in a pipeline–riser system with an S-shaped riser[J]. International Journal of Multiphase Flow, 2013, 55: 1-10.

[5] Xing L., Yeung H. and Shen J. et al. A new flow conditioner for mitigating severe slugging in pipeline/riser system[J]. International Journal of Multiphase Flow, 2013, 51: 65-72