IV Simpósio Paranaense de Modelagem, Simulação e Controle de Processos
ISSN : 1984-7521

Artigo: 11

Páginas: 75 - 81

CFD STUDY OF TWO-PHASE SLUG FLOW DEVELOPMENT IN HORIZONTAL PIPE

Carla Nayara Michels dos Santos*[1], Sarah Laysa Becker[1], Vinícius Basso de Godoy[1], Celso Murilo dos Santos[2], Henry França Meier[1], Marcela Kotsuka da Silva Câmara Bastos[1]

1 - University of Blumenau– FURB-SC, Blumenau – SC, carlanayara.michels@gmail.com
2 – University of Campinas – UNICAMP-SP, Campinas – SP

Abstract – Slug flow is an intermittent two-phase flow pattern characterized by gas and liquid flowing in the stratified form with unstable waves that grow to the upper part of the pipe. The purpose of this study is to determine the tube length for the flow development of a slug pattern using Computational Fluid Dynamics (CFD) technique. A horizontal geometry pipe with 7 m of length and 74 mm of internal diameter was build, where the inlet was set as a stratified flow. The superficial velocities applied were 10 m/s for air and 1 m/s for water. Results of water volume fraction profile show that the mathematical models were capable of predicting the slug flow behavior and the flow is fully developed in L/D ≅ 68. In conclusion, the 7 m pipe is suitable to study this flow pattern.
Keywords: Two-phase flow. Computational Fluid Dynamics. Slug flow. Mathematical modeling. Fully developed flow.

Introduction

Two-phase gas-liquid flows was extensively discussed by many researchers in recent decades. Different from the single-phase flow, that is rated as laminar or turbulent, the two-phase flow is characterized by several flow patterns. These are different phase-distributions that represent comportment and interactions of the phases restricted by a geometry. The flow pattern presented depends on physical properties of the fluids, geometrical variables of the pipeline or equipment, and operational parameters [1].

Slug flow is an unstable two-phase flow pattern, quite common in chemical industries, petroleum wellbore and flowline, and nuclear industry. The importance of study this flow pattern comes with the issues caused in production facilities, such as pressure rise, liquid overflow, increased corrosion, and low production. Slug is characterized by waves formed in the stratified flow, blocking air passage in the pipeline. When the pipe is filled up, the slug is accelerated by the gas upstream, dragging the liquid downstream, developing slug flow in the pipeline [2].

The fully developed flow occurs when the flow pattern does not change with the distance downstream [3]. There is no agreement of a minimum length to consider a two-phase flow fully developed. Although, Woods, Fan and Hanratty [4] suggested that, for a simple phase mixing device with absence of curves, an equilibrium in the flow development is reached at a dimensionless distance of 60 diameters downstream of the inlet.

Models and flow pattern maps were created by a number of authors to predict two-phase flow [2,5,6]. They are applied just for a limited range of geometric, operation and fluids variables. For this reason, Computational Fluid Dynamics (CFD) technique was applied as an alternative to predict flow behavior in specific geometries and fluids.

The purpose of this study is to evaluate the slug flow development in a horizontal pipe using CFD technique to conduct numerical experiments. Mathematical modeling proposed by Jaeger [7] were applied to capture horizontal slug flow.

Experimental

Numerical simulations were conducted in ANSYS Fluent 19.1. The 3D geometry applied consists of a horizontal cylinder with 7 m length and 74 mm of internal diameter, resulting in a L/D ≅ 95, where L is the length and D is the diameter of the pipe. Domain discretization in small volumes was necessary to solve the governing equations of the model, so a numerical mesh was elaborated, with 57,564 hexahedral elements. The inlet of the phases in the test section was stratified, pipe cross-section was divided in two parts, water enters at the bottom and air at the top, as shown in Fig. 1.

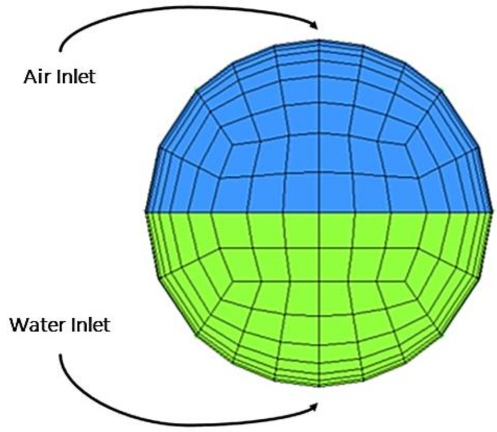

Figure 1 – Inlet numerical mesh.

The superficial velocities of air and water applied were 10 m/s and 1 m/s, respectively. This combination of superficial velocities was in the zone of slug flow according to the flow pattern map of Mandhane, Gregory and Aziz [5]. Water volumetric fraction monitoring points were inserted in the pipe upper region, in 4, 5, 6, and 7 m of the pipe length, corresponding to L/D ≅ 54, 68, 81 and 95, respectively, as presented in Fig. 2.

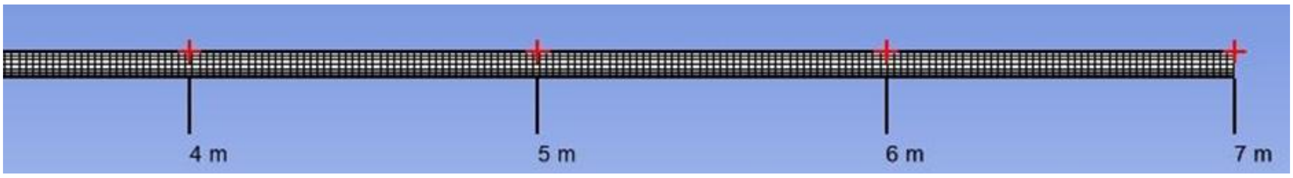

Figure 2 – Monitoring points location over the pipe.

Slug flow pattern is distinguished by the intermittent passage of liquid waves and elongated gas bubbles. These waves can block the cross-section of the pipe. Hence, when a wave flows through the pipe, liquid volumetric fraction in the monitoring points will be close or equal to 1. This indicates the occurrence of slug flow and if it is maintained over the pipe.

Mathematical Modeling

Employing mathematical models assessed by Jaeger [7], the Volume of Fluid (VOF) method was used to model the two-phase flow phenomenon and capture its time-dependent behavior. The following equations (1,2) describe mass and momentum conservation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \mathbf{v}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}[\rho \mathbf{v}] + \nabla.\{\rho \mathbf{v}\mathbf{v}\} = -\nabla p + \nabla.\mu_{eff}(\nabla \mathbf{v} + \nabla \mathbf{v}^{T}) + \rho \mathbf{g} \tag{2}$$

where t is the time, ρ is the density, $\mathbf{v}$ is the average vector velocity, p is the average static pressure, μ_{eff} is the effective viscosity, and $\mathbf{g}$ is the gravitational acceleration.

VOF introduces a new function in the set of momentum and turbulence equations that describes volumetric fraction in one mesh element. This function indicates the existence of an interface in the mesh element, but do not indicate its position [8]. It was presented in equation (3), where α_q is the function that defines volume fraction of the fluid q in the element.

$$\frac{\partial(\alpha_q \rho_q)}{\partial t} + \nabla.(\alpha_q \rho_q \mathbf{v}) = 0 \tag{3}$$

Several discretization methods were available to solve this equation. However, Geometric Reconstruction (Geo-reconstruction) achieved great results [7] and it was applied as a scheme to represent the area of interaction between fluids with high precision. This scheme indicates the position of the interface in one element, converting the gas-liquid interface into a straight line. Geo-reconstruction takes into account all the mesh elements in the neighborhood of the analyzed element to provide the interface position.

Turbulence effects were modeled with Reynolds-Averaged Navier-Stokes (RANS). The effective viscosity, from momentum equation (2), is the sum of molecular viscosity and the turbulent viscosity:

$$\mu_{eff} = \mu + \mu^T \tag{4}$$

This turbulent viscosity was calculated with k-ω Shear Stress Tensor (SST) model. It blends k-ω and k-ε model, using the first one in the core of the domain, and the second one near the walls [9]. The equation of turbulent viscosity (μ^T) that represent k-ω SST model is:

$$\mu^T = \frac{\rho a_1 k}{\text{Max}(a_1 \omega, \sqrt{2}\overline{\mathbf{D}} F_2)} \tag{5}$$

considering $a_1 = 0{,}31$, $\overline{\mathbf{D}}$ represents the rate of deformation, and F_2 is a mixture function of the model. Table 1 shows a resume of the configurations and models applied in the CFD simulator for this study.

Table 1 – Configurations and models applied in CFD simulations.

Condition	**Configuration**
Regime	Transient
Gravity	9,81 m/s²
Multi-Phase Model	VOF (Volume of fluid)
Turbulence Model	k-ω SST
Gas phase (Air)	Density: 1,225 kg/m³ Viscosity: 0,00001789 kg/m.s Fluid compressible (ideal gas)
Inlet Air	Superficial velocity 10 m/s Turbulence intensity 5%
Liquid phase (Water)	Density: 998,2 kg/m³ Viscosity: 0,001 kg/m.s Fluid incompressible
Inlet Water	Superficial velocity 1 m/s Turbulence intensity 5%
Outlet	Static pressure: 0 Pa
Pressure-Velocity Coupling	SIMPLE
Discretization Schemes	Pressure: PRESTO! Momentum: Second-order upwind scheme Volume fraction: Geometric Reconstruction Turbulence kinetic energy: First-order upwind scheme Specific dissipation rate: First-order upwind scheme
Time step	Adaptative
Courant Number	1

Results and Discussion

Volumetric fraction profiles over time obtained in the monitoring points are shown in Fig. 3. Peaks where water volumetric fraction was equal to 1 can be identified, demonstrating that slug flow pattern is presented in this condition. Observing the monitoring points, in L/D ≅ 54 (4 m) slug formation happens only between 9 and 14 seconds. The flow pattern is not maintained in the next seconds.

The points L/D ≅ 68 (5 m), 81 (6 m), and 95 (7 m), have a frequency of volumetric fraction peaks higher than in L/D ≅ 54. Peaks of pressure were maintained over time in these lengths. This shows a large incidence of slugs. Therefore, the flow is fully developed from 5 m forward, since this position the flow pattern remains the same, agreeing with Woods, Fan, and Hanratty [4].

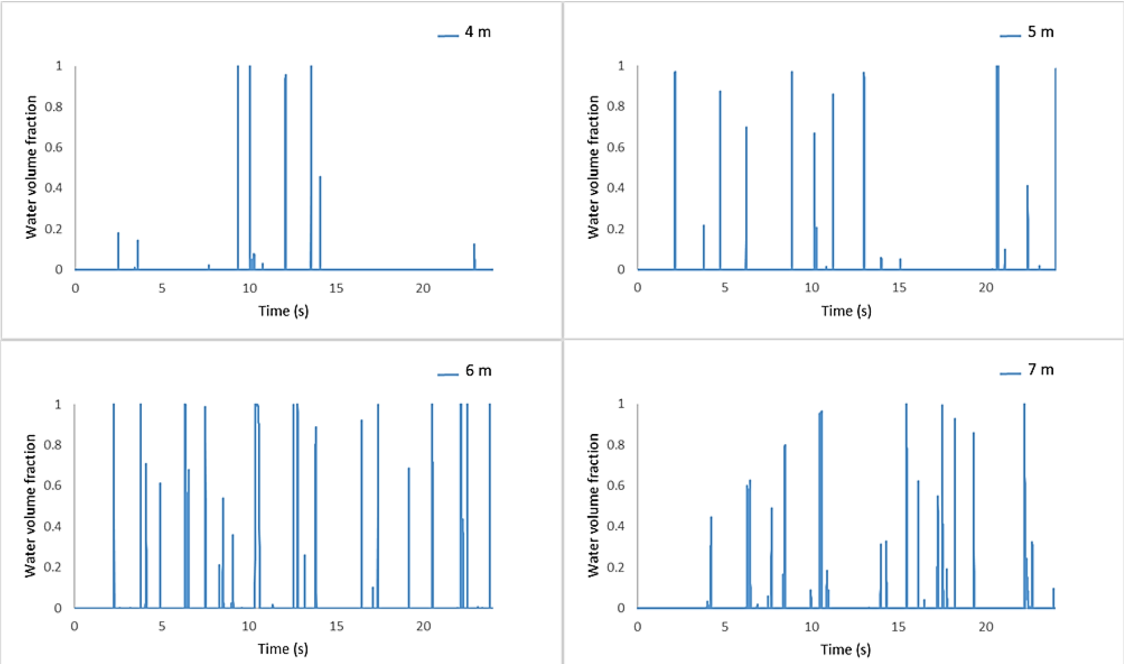

Figure 3 – Volumetric fraction profile in 4, 5, 6 and 7 m length.

Contours of water volumetric fraction over time are presented in Fig. 4 and 5. They represent the behavior of the volumetric fraction profiles in Fig. 3. It is possible to observe slugs flowing through the pipe and relate this with volumetric fraction peaks, in the same timestep. The contours also demonstrate that the flow pattern is maintained from 5 m forward.

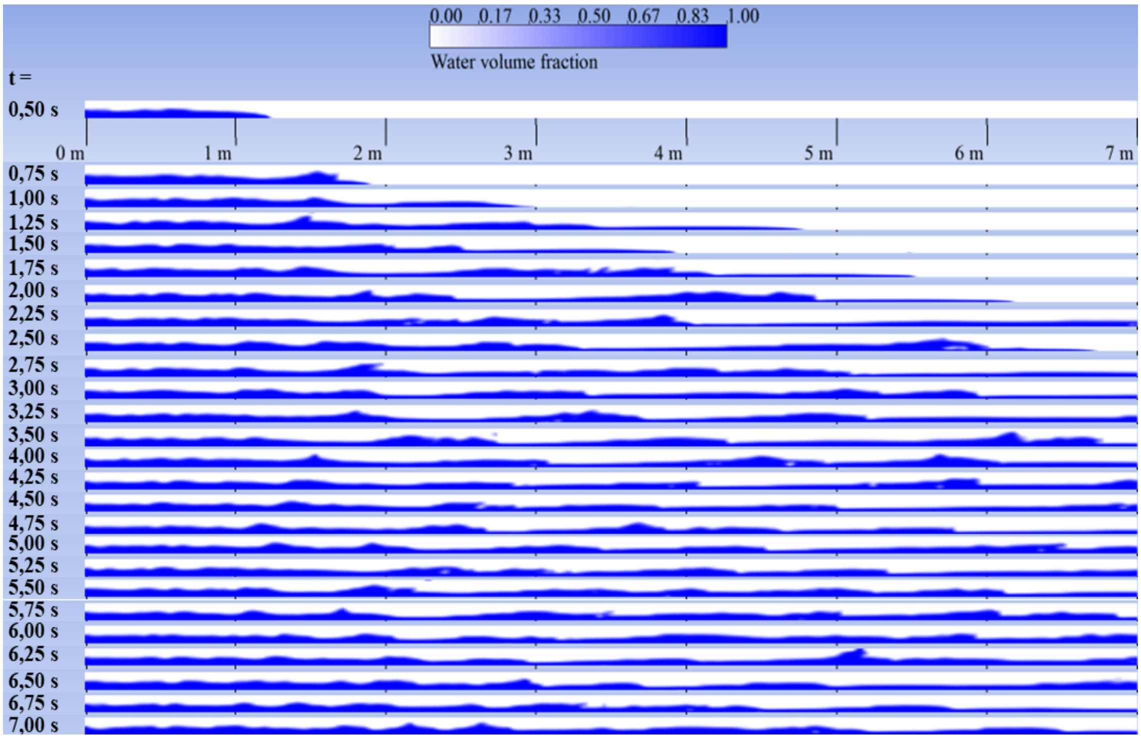

Figure 4 – Contours of water volumetric fraction over time between 0.50 and 7.00 s.

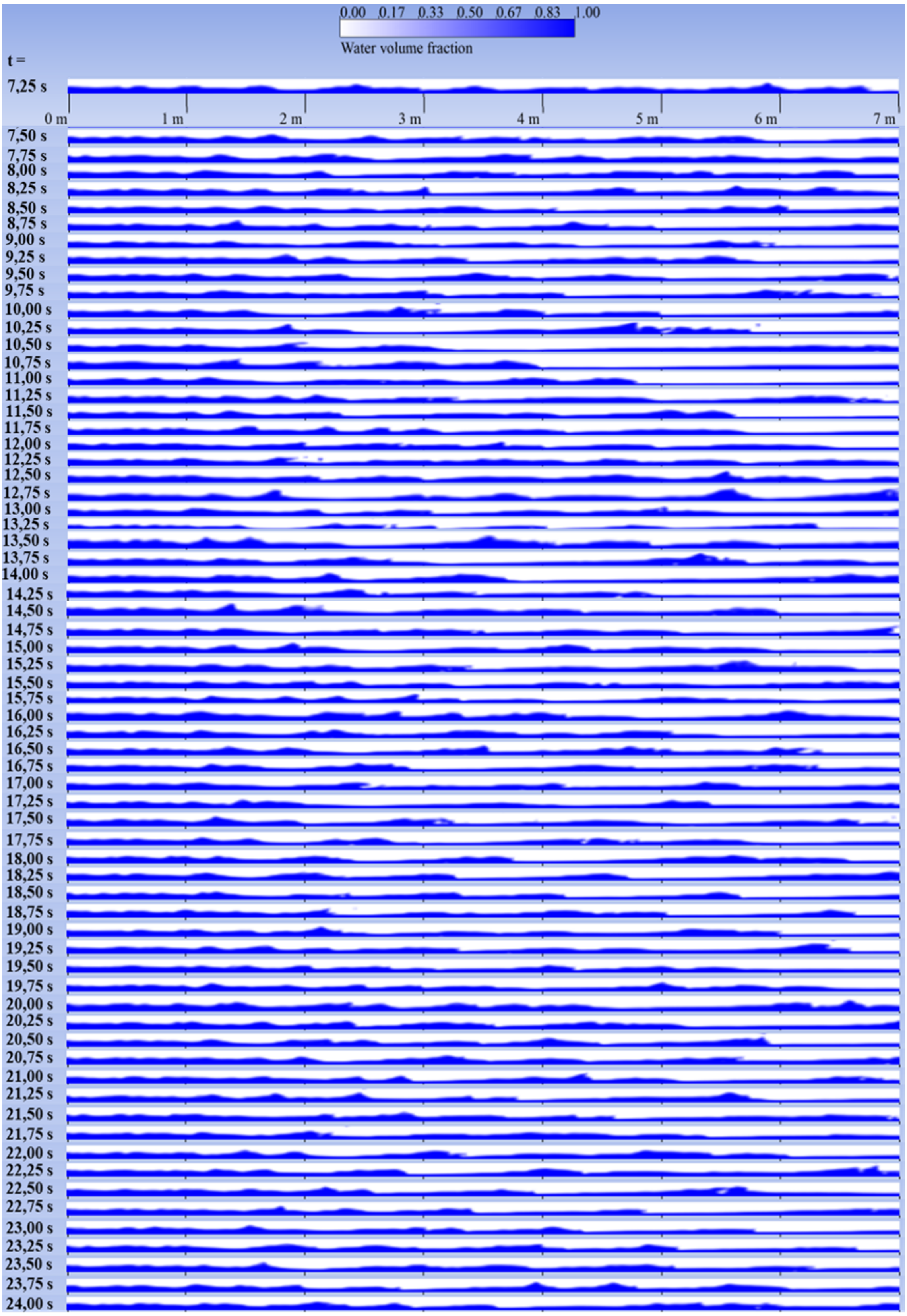

Figure 5 – Contours of water volumetric fraction over time between 7.25 and 24.00 s.

Conclusion

A horizontal pipe geometry was constructed to evaluate the tube length for the flow development of slug pattern, applying CFD technique to conduct the experiment.

Results of volumetric fraction profiles and contours showed that the models used were able to capture slug flow pattern in horizontal pipes. After performing the numerical tests, in summary, a 7 m tube length is suitable to study this type of flow, because in the tested case the flow development occurred after L/D ≅ 68 (5 m).

Acknowledgements

The authors are grateful for the financial support of PETROBRAS for this research project, through the cooperation agreement number 5850.0103010.16.9. This study was financed in part by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior – Brasil (CAPES) – Fincance Code 001.

References

1. O. Shoham. Mechanistic Modeling of Gas-liquid Two-phase Flow in Pipes, Society of Petroleum Engineers, 2006.
2. Y. Taitel; A. E. Dukler. A model for slug frequency during gas-liquid flow in horizontal and near horizontal pipes. International Journal of Multiphase Flow. 1977, v. 3, p. 585-596.
3. M. Abdulkadir et al. Two-phase air-water flow through a large diameter vertical 180° return bend. Chemical Engineering Science. 2012, v. 79, p. 138-152.
4. B. D. Woods; Z. Fan; T. J. Hanratty. Frequency and development of slugs in a horizontal pipe at large liquid flows. International Journal of Multiphase Flow. 2006, v. 32, p. 902-925.
5. J. M. Mandhane; G. A. Gregory; K. A. Aziz. A flow pattern map for gas-liquid flow in horizontal pipes. International Journal of Multiphase Flow. 1974, v. 1, p. 537-553.
6. O. Baker. Design of Pipelines for Simultaneous Flow of Oil and Gas. Oil & Gas Journal. 1954, p. 185-201.
7. J. Jaeger et al. Experimental and numerical evaluation of slugs in a vertical air–water flow. International Journal of Multiphase Flow. 2018, v. 101, p.152-166.
8. C. W. Hirt; B. D. Nichols. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics. 1981, v. 39, p. 201-225.
9. F. Moukalled; L. Mangani; M. Darwish. The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM and Matlab, Springer, 2016.