17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT2018-0494
TWO-PHASE FLOW IN A ROCK-FLOW CELL: COMPARISON OF VOF-ISOADVECTOR MODEL WITH EXPERIMENTS

Alex V. L. Machado
Paulo H. D. Santos
Eduardo Nunes dos Santos
Marco José da Silva
Fabio A. Schneider
Rigoberto E. M. Morales
Multiphase Flow Research Center (NUEM), Postgraduated Program in Mechanical and Materials Engineering (PPGEM), Federal University of Technology - Paraná (UTFPR) – Rua Deputado Heitor Alencar Furtado 5000, Bloco N, CEP 81280-340, Curitiba, Brazil
alexvlmachado@gmail.com, psantos@utfpr.edu.br, e.n.santos@ieee.org, mdasilva@utfpr.edu.br, fabioschneider@sefter.com.br, rmorales@utfpr.edu.br

Amadeu K. Sum
Hydrates Energy Innovation Laboratory, Department of Chemical & Biological Engineering, Colorado School of Mines, 1500 Illinois St, Golden, CO 80401, USA.
asum@mines.edu

Abstract. *This paper presents a numerical and experimental study to capture the free surface of a two-phase flow in a rock-flow cell. The numerical code was implemented in OpenFOAM® C++ library using the volume-of-fluid (VOF) method for moving meshes and the interface was tracked by the isoAdvector scheme. Rock-flow cell is a novel system for hydrate formation analysis, and it has been proved to capture most hydrates phenomena, however, more information of the flow is needed. Computational fluid dynamics methods can play a crucial role in given detailed information and deepen the understanding of the flow behavior in rock-flow cells, once the method is validated by experimental results. In order to ensure the reliability of present algorithm, experiments were carried out with water and air in a rock-flow cell with different rotational speeds. It was used two wire-mesh sensors in order to measure the hydrodynamic characteristics and patterns of multiphase flow. It was observed stratified and slug flow patterns. The comparison between numerical and experimental results showed a good agreement.*

Keywords: *Multiphase flow, Rock-flow cells, OpenFOAM, VOF, IsoAdvector.*

1. INTRODUCTION

Offshore petroleum production is generally accompanied by produced water which in combination with high pressures, low temperatures and gas phase presence can cause the formation of gas hydrates, an additional solid phase. Gas hydrates occurrence has been a vast issue in oil and gas production and transportation, because in some scenarios, there are risks of the pipeline blockage by solid agglomeration and deposition (SLOAN, 2005).

The most used experimental system to study hydrates formation has been the flowloop, because it enables observations of all steps related to hydrates formation evolution, and it is the most similar to field production. The problem with flowloop is the high cost to build and operate, and the limited visualization of the flow. Recently, a new system has been used to study hydrates formation, called rocking cell or rock-flow cell (RFC). The most advantages of RFC are being about twenty times less expensive to construct and operate than flowloop, and the better visualization and control capacities. However, most studies conducted so far did not focus on the fluid mechanics of this system. Therefore, there is a necessity of understanding the multiphase flow characteristics of the RFC in order to translate the results from this benchtop testing to real flow conditions (STRAUME et al, 2016). Modern tools, such as computation fluid dynamics (CFD) can help bringing detailed information about the flow in rock-flow cells, as well as can be used to predict real situations in pipelines.

Based on these gaps, a new rock-flow cell was built in the Multiphase Flow Research Center (NUEM) at UTFPR. First of all, this new RFC is used to acquire more knowledge about the multiphase flow characteristics and analyze the capacity to generate different flow patterns. Then the experimental results are used to validate a new solver implemented in the OpenFOAM® code which captures the free surface movement with the volume-of-fluid (VOF) method. This new solver has the ability to deal with meshes in movement and uses the isoAdvector scheme proposed by Roenby et al (2016) to geometrically advect fluid interfaces.

2. EXPERIMENTAL SETUP

Figure 1 shows an overview of the rock-flow cell test rig which is installed at Multiphase Flow Center (NUEM/UTFPR). The cell has a cylindrical shape with internal diameter of 5.4 cm and internal length of 100 cm. It is supported near the left end and the cell is free to oscillate by a motor between positive and negative inclinations. All the experiments were carried out with water and air, and different two-phase flow patterns were obtained by varying the rotational speed. The system has two Wire-Mesh Sensors (WMS) in the center in order to get the void fraction over time. A high-speed camera was installed near the right end, and all the tests were filmed. The numerical simulations need the inclination of the cell over time, and it was obtained by an accelerometer, which was trigger by the WMS acquisition system.

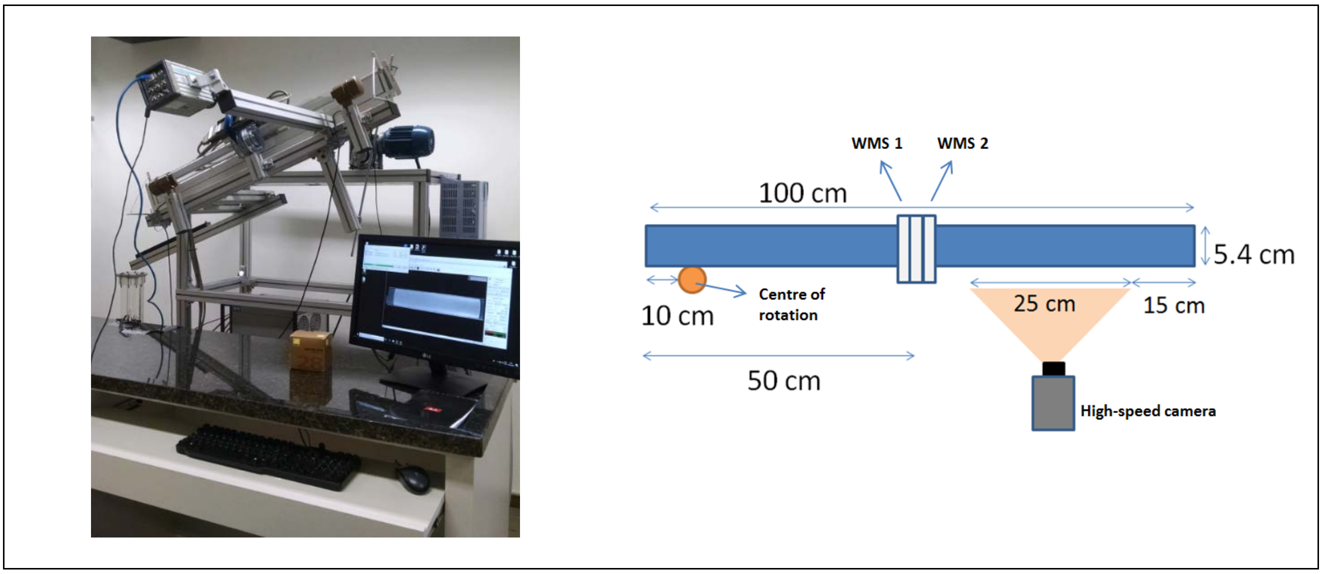

Figure 1. (a) Picture of the rock-flow test rig at NUEM/UTFPR. (b) Schematic representation of the RFC, showing the position of high-speed camera, centre of rotation and WMS.

Figure 2 presents the wire-mesh sensor, which was used in order to measure the two-phase flow characteristics. Wire-mesh sensor consists of transmitter and receiver wires measuring the electrical properties of the flow within its slightly spaced crossing points. The transmitter electrodes are activated consecutively, while keeping all other transmitter electrodes at ground potential. The receiver wire measures the capacitance (permittivity) of the surrounding flow phase at the crossing point. In this study, two wire-mesh sensors of 16x16 wires with a frame rate of 2000 Hz were used to measure the void fraction, phase velocities and to visualize the flow pattern (DA SILVA, 2007).

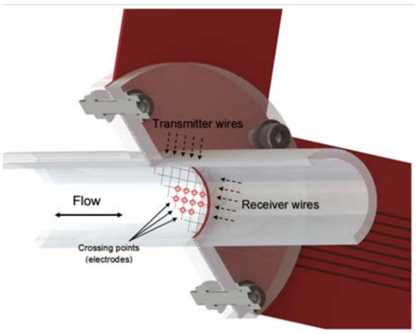

Figure 2. Schematic representation of Wire-mesh sensor.

3. COMPUTATIONAL METHODOLOGY

The simplifying assumptions considered in this work are restricted to incompressible and isothermal flows, there is no mass transfer between the phases and the flow is turbulent. The mathematical model comprises primarily one set of mass and momentum conservation for both phases, which is often referred as one-fluid approach (Tryggvason et al, 2011):

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\frac{d\rho\mathbf{u}}{dt} + \nabla \cdot \left[\rho(\mathbf{u} - \mathbf{u}_m)\mathbf{u}\right] = -\nabla p + \left[\nabla \cdot (\mu_{eff} \nabla \mathbf{u}) + \nabla \mathbf{u} \cdot \nabla \mu_{eff}\right] + \rho \mathbf{g} + \sigma \kappa \nabla \alpha \quad (2)$$

where $\mathbf{u}$ is the fluid mixture velocity, $\mathbf{u}_m$ is the mesh velocity, ρ is the mixture density, p is the pressure, μ_{eff} is the effective mixture viscosity, which is based on the Boussinesq approximation $\mu_{eff} = \mu + \mu_t$, μ is the mixture absolute viscosity, μ_t is the eddy viscosity, $\mathbf{g}$ is the gravity acceleration vector, σ is the surface tension coefficient, κ is the interface curvature, and α is the water volume fraction for all the simulations of this paper. The viscous term (the second term in right hand side) was rewritten from $\nabla \cdot \left[\mu_{eff} (\nabla \mathbf{u} + \nabla \mathbf{u}^T)\right]$.

The eddy viscosity μ_t is obtained from the two-equation Shear Stress Transport (SST) model proposed by Menter (1994). This paper does not have the objective to describe the SST model used in the algorithm, however further information about the OpenFOAM® current version of the SST model can be found in the work of Menter *et al* (2003). The last term in the right hand side of the Eq. (2) is the surface tension force on the free surface, in which the continuum surface force formulation of Brackbill *et al* (1992) was employed.

The mathematical approach was implemented and solved in OpenFOAM® version 4.1, which is an open source object-oriented library written in the C++ programming language for numerical simulations in continuum mechanics (JASAK *et al*, 2007). The new solver proposed in this work is a new application called interIsoDyMFoam and the equations are numerically solved in the framework of the finite volume method.

The fundamental equation of isoAdvector is the integral form of the volume fraction transport equation:

$$\alpha_i(t + \Delta t) = \alpha_i(t) - \frac{1}{\Omega_i} \int_t^{t+\Delta t} \int_{\partial\Omega_i} H(\mathbf{x}, t)\mathbf{u}(\mathbf{x}, t) \cdot d\mathbf{S} d\tau \quad (3)$$

where $\alpha_i(t + \Delta t)$ and $\alpha_i(t)$ are the current and old water volume fraction values for an arbitrary cell, respectively. $\mathrm{H}(\mathbf{x}, t)$ is the Heaviside Step Function, which in this work is 1 for water and 0 for air. Ω_i and $\partial\Omega_i$ are the volume and boundary of the arbitrary cell, respectively. More numerical details about isoAdvector scheme can be found in the original publication of Roenby *et al* (2016). This numerical model will be described in expanded format in the final paper which will be submitted to this conference.

3. RESULTS

Figure 3 compares three time frames of the experiment (left) and the CFD simulation (right) for a two-phase flow composed by water and air with 30% volume liquid loading. The amplitude was ± 11° and rotational speed was 9 rpm. It is possible to observe that the present numerical model captured fairly well the movement of the two-phase mixture. More accuracy is observed for the free surface, in which the model was designed. The model was capable of capturing some bubble entrainment, as a result of wave splashing, when the liquid reaches the right end of the cell. However more deep investigation is needed in order to quantify the bubble size and interfacial area. As previous results, it can be inferred the simulation of interface movement is acceptable for the objective of analyzing the bulk velocity of both fluids.

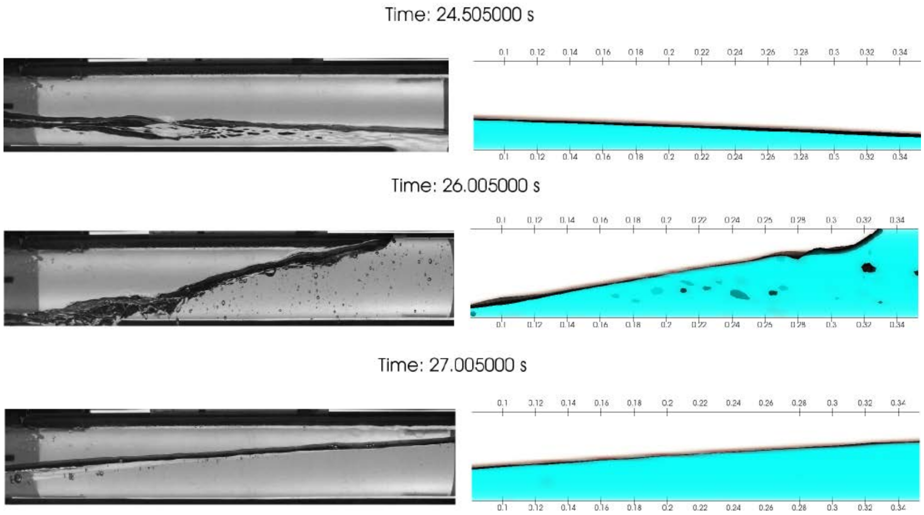

Figure 3. Experiments versus CFD. (Left) Images from High-speed Camera and (right) simulations with VOF-isoAdvector model.

Figure 4 compare the averaged void fraction in the middle of the cell from experiment with WMS and simulation results from CFD model. This result is important to give more quantitative information regarding on the capacity of the numerical model to describe the flow. The frequencies of the liquid passing the WMS, which are the low peaks, were captured by the model as well the inclination of the curve. Some deviation is observed for the minimum void fraction values, which can be generated by the difficult of the numerical model to deal with small bubbles or from the experimental system, in which the bubble entrainment was not considered in the averaged void fraction.

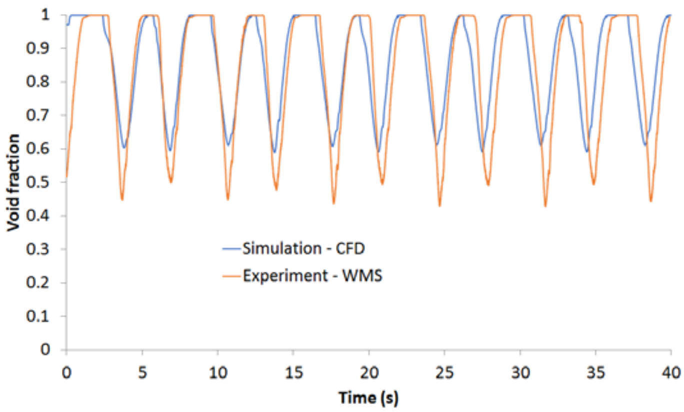

Figure 4. Comparison of averaged void fraction from CFD results and WMS results.

4. CONCLUSIONS

A numerical approach was established in this work to study the two-phase flow in rock-flow cells. Experiment results obtained with Wire-Mesh sensors and High-speed camera were in good agreement with the numerical results. The VOF-isoAdvector model was capable of tracking the free-surface flow and the occurrence of entrainment. Further work is needed in order to get more information about the phenomena in rock-flow cells, and it can be done from the simulation, for example from the phase velocities, interfacial area and flow shear. These data will be important to evaluate the mixture between the phases, which are fundamental to hydrates formation studies.

5. REFERENCES

Brackbill, J. U., Kothe, D. B. and Zemach, C., 1992. “A continuum method for modeling surface tension”. Journal of Computational Physics. Vol. 100, Pages 335–54.

Da Silva, M.J. Schleicher, E. Hampel, U. 2007. Capacitance wire-mesh sensor for fast measurement of phase fraction distributions. Measurement Science and Technology 18 (7), 2245.

Jasak, H., Jemcov, A., Tukovic, Z., 2007. OpenFOAM: A C++ Library for Complex Physics Simulations. International Workshop on Coupled Methods in Numerical Dynamics IUC, Dubrovnik, Croatia.

Menter, F.R., Kuntz, M., Langtry, R., 2003. Ten years of industrial experience with the SST turbulence model. In Proceedings of the fourth international symposium on turbulence, heat and mass transfer, pages 625–632, Antalya, Turkey, Begell House.

Roenby, J., Bredmose, H., Jasak, H., 2016. A computational method for sharp interface advection," Royal Society Open Science, vol. 3, no. 11, p. 160405.

Sloan, E. D., 2005. "A changing hydrate paradigm—from apprehension to avoidance to risk management". Fluid Phase Equilibria, Vol. 67–74, p. 228–229.

Straume, E. O., Kakitani, C., Merino-Garcia, D., Morales, R. E., Sum, A. K., 2016. "Experimental study of the formation and deposition of gas hydrates in non-emulsifying oil and condensate systems". Chemical Engineering Science, Vol. 155, p. 111–126.

Tryggvason, G., Scardovelli, R., Zaleski, S., 2011. Direct numerical simulations of gas liquid multiphase flows. Cambridge University Press.

6. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.