24th COBEM - 2017

24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-1718
NUMERICAL AND EXPERIMENTAL STUDY OF FREE SURFACE FLOWS IN A ROCK-FLOW CELL

Alex V. L. Machado
Fabio A. Schneider
Paulo H. D. Santos
Rigoberto E. M. Morales
Multiphase Flow Center (NUEM), Postgraduated Program in Mechanical and Materials Engineering (PPGEM), Federal Technological University of Paraná (UTFPR) – Rua Deputado Heitor Alencar Furtado 5000, Bloco N, CEP 81280-340, Curitiba, Brazil
alexvlmachado@gmail.com, fabioschneider@sefter.com.br, psantos@utfpr.edu.br, rmorales@utfpr.edu.br

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

Abstract.
In this paper is proposed a new algorithm using the OpenFOAM® C++ library which solves the volume-of-fluid equations in moving meshes using isoAdvector, a recently published approach to geometrically reconstruct and advect fluid interfaces. The motivation for addressing this problem is to numerically simulate the free surface in rock-flow cells, a benchtop testing used to study hydrates kinetics formation. Hydrates particles agglomeration and blockage is one of the most concerns in deep-sea petroleum production. Rock-flow cell rig experiments have been shown to be suitable to capture most of the hydrates formation related phenomena. However, a vast issue in all studies conducted so far is how to correlate the data observed experimentally with real flow pipelines. 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 our new algorithm, experiments were carried out with water and air in a rock-flow cell with different liquid loadings and rotational speeds. Flow patterns which usually occur in multiphase flows were observed in the results, such as stratified and slug flow patterns. The numerical approach was capable of simulating the experimental results with good agreement.

Keywords: *hydrates, multiphase flow, rock-flow cells, VOF, isoAdvector.*

1. INTRODUCTION

Offshore petroleum production is frequently 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 is one of the biggest issues in oil and gas production and transportation, since this phenomenon can increase pumping costs or, in extreme conditions, triggers risks of the pipeline blockage by solid agglomeration (Straume *et al*, 2016). In order to allow deep sea petroleum production, a huge amount of hydrate inhibitors are added continually in the pipelines. However, this procedure does not guarantee safe operation conditions at all times, especially at shut-in and restart situations (Sloan, 2005).

For these adverse cases, remediation techniques are needed, which are based on kinetics studies. Currently, the most appropriate test apparatuses to experimentally study hydrates kinetics are flowloops and rock-flow cells, because these systems enable observations of all steps related to hydrates formation evolution, such as agglomeration, deposition, bedding and blocking (Sloan *et al*, 2011). Rock flow cell rigs have the advantage of being about twenty times less expensive to construct and ten times less expensive to operate than flowloops. However, the flow in the rock-flow cells has not the same behavior when compared with real oil and gas production lines. Therefore, understanding the multiphase flow characteristics of these systems are a very significant factor in order to better correlate the results

from the benchtop testing with real flow conditions (Sum, 2016). Modern tools, 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, once the ad hoc model has been validated by experiments.

To start shedding light into these issues, simulations were performed with a new solver implemented in the OpenFOAM® code which captures the free surface with the volume-of-fluid (VOF) method. This new solver has the ability to deal with meshes in movement and draws upon of the isoAdvector concept proposed by Roenby *et al* (2016) to geometrically advect fluid interfaces. The model was validated with experimental tests performed at Hydrate Innovation Laboratory of Colorado School of Mines (CSM) with a rock-flow cell with visual capabilities.

2. EXPERIMENTAL SETUP

Figure 1 shows an overview of the experimental rig, the cell has a cylindrical shape with internal diameter of 0.0508 m and internal length of 0.5842 m. It is supported on a pillow block and the ends are free to oscillate by a motor between positive and negative inclinations. Different flow patterns can be obtained by varying the rotational speed, liquid loading and maximum inclination angle. All the experiments were carried out with water and air, and a very small quantity of blue dye was added to improve the flow visualization, but it did not interfere in the results.

Figure 1. CSM's rock-flow test rig.

3. COMPUTATIONAL METHODOLOGY

The system considered in this work is 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):

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

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

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. In the current implementation the curvature κ is function of the volume fraction gradient, which has the same direction of the interface normal vector $\mathbf{n}$:

$$\kappa = -\nabla \cdot \mathbf{n} = -\nabla \cdot \left(\frac{\nabla\alpha}{|\nabla\alpha|}\right) \tag{3}$$

The density and viscosity of the mixture are calculated as a function of volume fraction field:

$$\rho = \rho_{water}\alpha + \rho_{air}(1-\alpha) \tag{4}$$

$$\mu = \mu_{water}\alpha + \mu_{air}(1-\alpha) \tag{5}$$

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.

Even though the mesh points are moving during the simulation, the volume of each cell does not change with time. Thus, there are not topological changes in the mesh, which in OpenFOAM® is referred as solid body motion (Jasak, 2009). The momentum equation is similar as the version presented by Deshpande *et al* (2012), with differences in the divergent term (the second term in l.h.s of equation 2) in which the mesh velocity arises and the direction of the gravity acceleration vector, which changes when the mesh moves.

As mentioned before, the interface is tracked using the VOF method. The idea of using the volume fraction field to advance the interface was first proposed in the literature by Hirt and Nichols (1981). Since then, a vast amount of papers has been proposed to solve the same equations and trying to guarantee at the same time boundedness of the volume fraction field and sharpness of the interface. Numerical approaches have been divided in two categories: geometric approaches which use a reconstructed interface to advect the volume fraction; or algebraic approaches, which a reasonable sharp interface is reached by high order schemes or additional compression terms (Denner, Van Wachem, 2014).

IsoAdvector is a geometric method devised to work well in arbitrary polygonal meshes. The novelty of this geometric method lies on the ability to reconstruct interfaces using isosurfaces inside cells and the use of face-interface intersections lines of the reconstructed interface to obtain the time evolution of submerged face area within a time step, as illustrated in figure 2.

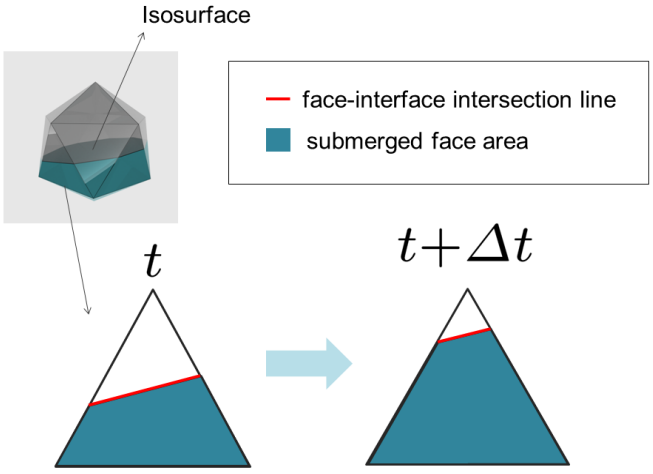

Figure 2. An arbitrary polygonal mesh element where an isosurface is calculated and advanced from time t to $t+\Delta t$. The bottom shows the definition of face-interface intersection line and submerged area for one of the cell faces.

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

$$\alpha_i(t+\Delta t) = \alpha_i(t) - \frac{1}{\Omega_i}\int_t^{t+\Delta t}\int_{\partial\Omega_i} H(\mathbf{x},t)\mathbf{u}(\mathbf{x},t)\cdot d\mathbf{S}d\tau \tag{6}$$

where $\alpha_i(t+\Delta t)$ and $\alpha_i(t)$ are the current and old water volume fraction values for an arbitrary cell, respectively. $H(\mathbf{x},t)$ 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) or in the source code available online in the directory of Roenby (2016).

Hernandez-Perez *et al* (2010) compared different grid topologies for multiphase flow simulations in pipelines and concluded that the so-called butterfly grid was the best choice for VOF simulations. Therefore, this type of mesh was chosen, as showed in figure 3. Meshes with different number of cells were compared using the present code and the optimal size mesh is shown in figure 3.

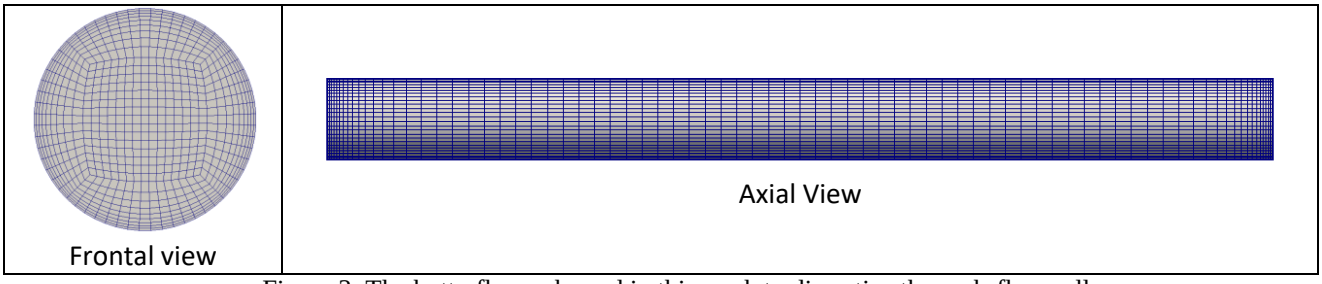

Figure 3. The butterfly mesh used in this work to discretize the rock-flow cell.

4. RESULTS

Figure 4 compares three time frames of the experiment (blue) and the CFD simulation (red) for a two-phase flow composed by water and air with 50% liquid loading. The rotational movement is a sine wave with a rotational speed of 6 rpm and maximum inclination of 15º. All the simulation results were analyzed in the post-processing software Paraview®. The simulation pictures presented in this paper were obtained using a volume rendering technique. As can be seen, the pictures of the experiment and the axial view of simulation have a good agreement. This case could be classified as a kind of stratified flow, and some waves are generated when the cell changes the rotational direction. In the first frame of the experiment, it can be seen some small bubbles after the wave splash, and the present algorithm was capable of capture this phenomenon, which are the white bubbles in the simulation results.

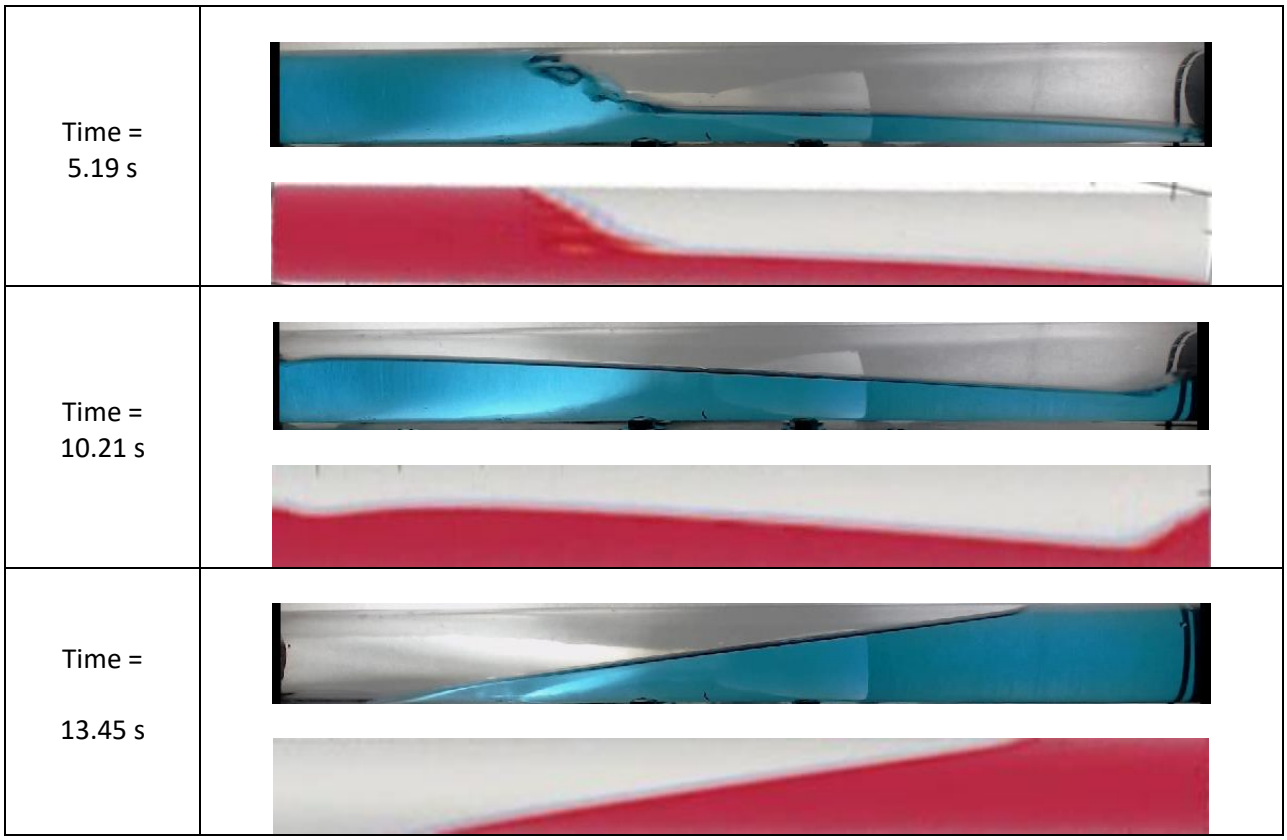

Figure 4. Comparation of experiment pictures obtained at CSM's rock-flow cell (blue) and simulations conducted with isoAdvector (red) for a test with 50% liquid loading and rotational speed of 6 rpm.

Figure 5 compares pictures of experiment and simulation for a slug flow test. This pattern flow was obtained using 90% liquid loading and a rotational speed of 20 rpm. Once again, reasonable agreement was observed for the liquid distribution between the experiments and CFD simulations. Even though VOF models are not able to describe bubbles

smaller than the grid size, as can be seen this limitation did not have an impact in the large-scale-length-interface or in the general velocity profile, once both experiment and simulations are synchronized.

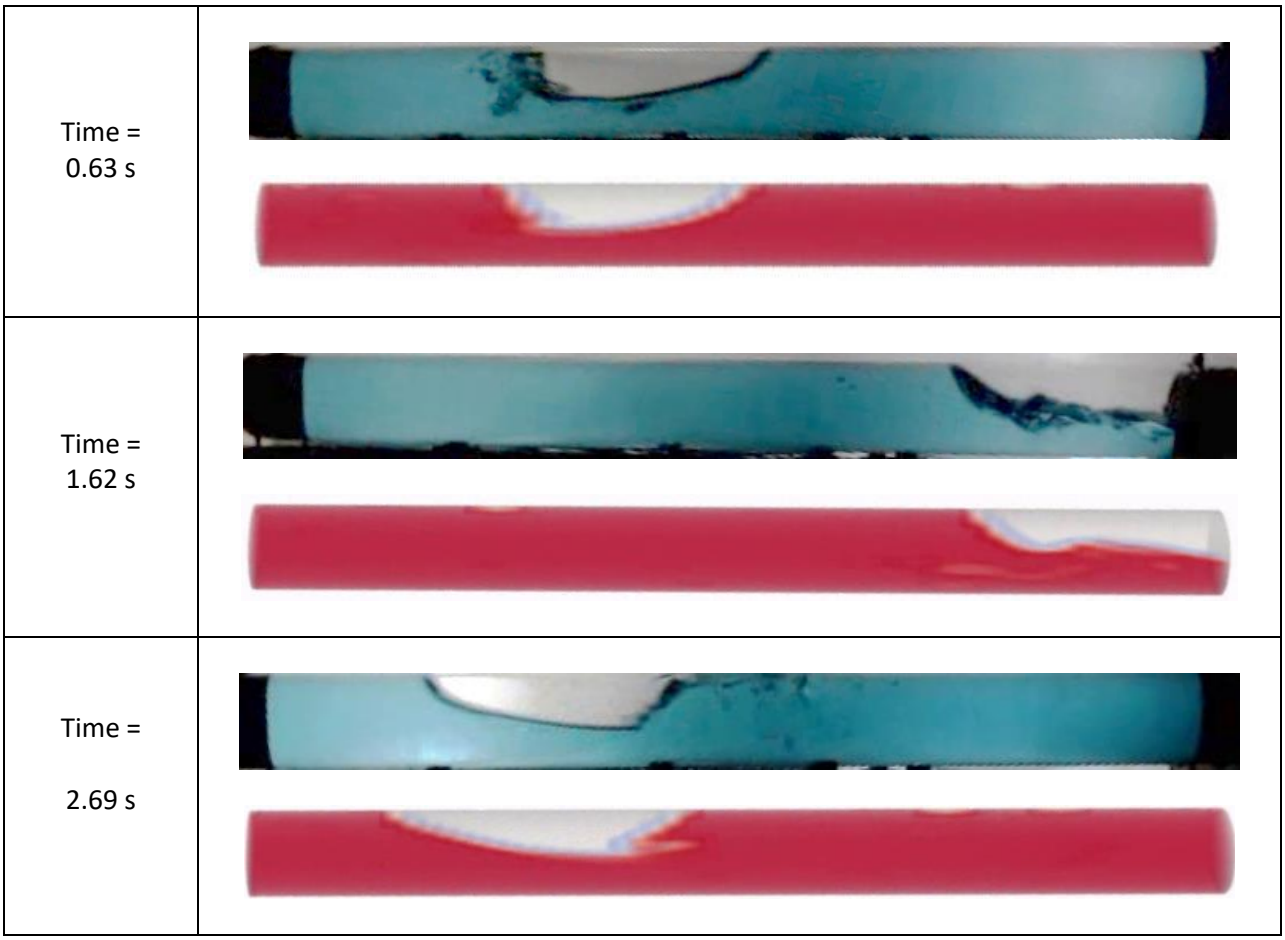

Figure 5. Comparation of experiment pictures obtained at CSM's rock-flow cell (blue) and simulations conducted with isoAdvector (red) for a test with 90% liquid loading and rotational speed of 6 rpm.

In order to give a better description of the flow behavior in the tests presented so far, Figure 6 shows the streamlines for a specific time frame for each test presented in this article. In the test with 50% liquid loading (Fig. 6a) the flow is distributed for all over the cell, with the streamlines going from one end to another. In the case of 90% liquid loading (Fig. 6b), the velocity is concentrated near the Taylor bubble.

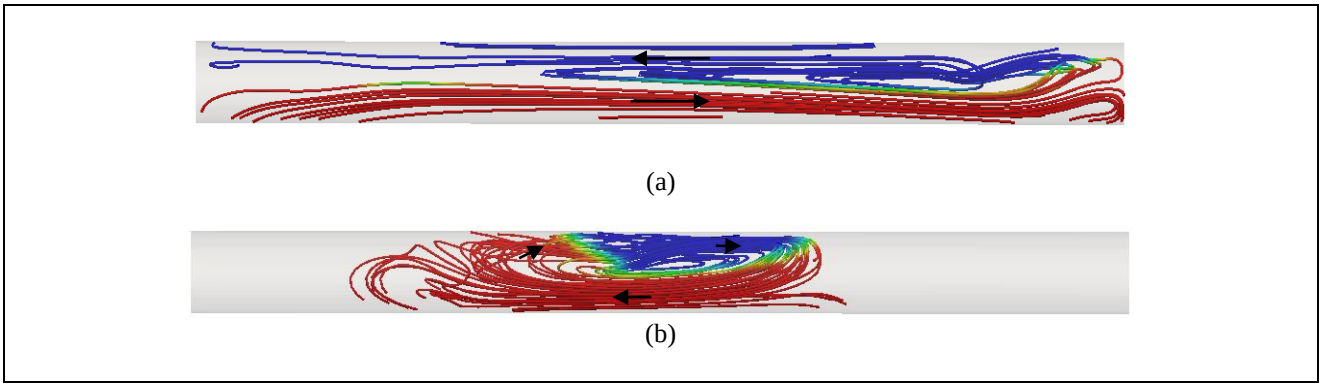

Figure 6. Streamlines of rock-flow cell simulations. (a) Stratified flow with 50% liquid loading, (b) Slug flow with 90% liquid loading.

5. CONCLUSIONS

An algorithm which enables simulations of the isoAdvector method in moving meshes has been developed in the OpenFOAM® library. The motivation for this work was to have a better description of the free surface in rock-flow

cells using a geometric scheme to solve the VOF equations. Nevertheless, it is important to mention that other applications that deal with multiphase flows in moving meshes may benefit with this new solver, such as the flow in mixers, sloshing tanks, pumps and others.

The simulations tests carried out in this work had visual good agreement with the experiments performed in the CSM's rock-flow cell for both 50% and 90% liquid loading tests with water and air. Once the algorithm was validated, the next step will be using the CFD simulations of rock-flows cells to provide quantitative information about the flow, such as velocity profiles, shear rate and interface area. These parameters will be important to correlate the experiments in the benchtop testing and the real flow in oil and gas production lines.

6. ACKNOWLEDGEMENTS

This research was funded by Repsol Sinopec Brasil and Coordination for the Improvement of Higher Education Personnel (CAPES).

7. 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.

Denner, F., Van Wachem, B.G.M., 2014. "Compressive VOF method with skewness correction to capture sharp interfaces on arbitrary meshes", *Journal of Computational Physics*, Volume 279, Pages 127-144.

Deshpande, S. S., Anumolu, L., Trujillo, M. F., 2012. Evaluating the performance of the two-phase flow solver interFoam," *Computational Science & Discovery*, vol. 5, no. 1, p. 014016.

Hernandez-Perez, V., Abdulkadir, M., Azzopardi, B.J., 2010. "Grid Generation Issues in the CFD Modelling of Two-Phase Flow in a Pipe". *Journal of Computational Multiphase Flows*, vol. 3, nº 1.

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.

Jasak, H., 2009. "Dynamic Mesh Handling in OpenFOAM", *47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition, Aerospace Sciences Meetings*.

Menter, F. R., 1994. "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", *AIAA Journa*l, vol. 32, no 8. pp. 1598-1605.

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.

Roenby, J., 2016. "isoAdvector." Available in <www.github.com/isoAdvector>.

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

Sloan, D., Koh, C., Sum, A. K., 2011. *Natural Gas Hydrates in Flow Assurance*. Elsevier.

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.

Sum, A. K., 2016. "Rock-flow cell for hydrates studies with multiphase flow". Presentation. Colorado School of Mines.

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

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.