Fluid Flow, Energy Transfer and Design

Edited by
Antonio F. Miguel, Luiz Rocha
and Andreas Öchsner

TTP TRANS TECH PUBLICATIONS

Fluid Flow, Energy Transfer and Design

Special topic volume with invited peer reviewed papers only.

Edited by

Antonio F. Miguel, Luiz Rocha and Andreas Öchsner

TTP

Trans Tech Publications Ltd
Kreuzstrasse 10
CH-8635 Durnten-Zurich
Switzerland
http://www.ttp.net

Volume 348 of
Defect and Diffusion Forum
ISSN print 1012-0386
ISSN cd 1662-9515
ISSN web 1662-9507
(Pt. A of Diffusion and Defect Data – Solid State Data ISSN 0377-6883)

Full text available online at *http://www.scientific.net*

***Distributed** worldwide by*

Trans Tech Publications Ltd
Kreuzstrasse 10
CH-8635 Durnten-Zurich
Switzerland

Fax: +41 (44) 922 10 33
e-mail: sales@ttp.net

and in the Americas by

Trans Tech Publications Inc.
PO Box 699, May Street
Enfield, NH 03748
USA

Phone: +1 (603) 632-7377
Fax: +1 (603) 632-5611
e-mail: sales-usa@ttp.net

printed in Germany

Editorial

Fluid flow, thermodynamics and heat/mass transfer are central pillars of science and technology. They have been central to the development of our civilization because we use them to understand natural-world phenomena but also to move forward through incremental improvements in technology.

Nowadays, some long-standing fundamental problems remain unsolved while current developments are giving rise to many more of them. Therefore, advances in the understanding of fluid flow, thermodynamics and heat/mass transfer continue to be crucial in science as well as in almost all fields of engineering. They are usually part of applied mathematics, physics and engineering research, and can be involved in astrophysics, meteorology, geophysics, oceanography, biology, and much more, including the traditional branches of engineering (mechanical, civil and chemical engineering) and recent ramifications (bioengineering and bio-technology). Consequently, the tendency to become compartmentalized into subjects with different groups may leads to particular advancements being known only inside each area.

The special session "Fluid Flow, Energy Transfer and Design" held at the 9th International Conference on Diffusion in Solids and Liquids (DSL 2013) sheltered papers of different areas ranging from physics, mathematics and chemistry to engineering. It served as a link under which authors of different areas and backgrounds came together, and make their research accessible to the varied audience. In this sense worked to counter the possible divisive tendency.

This special issue is a fitting tribute to the different views since this is not a divisive tendency but the seethe of science that shapes the ever-changing landscapes of our research world.

Antonio F. Miguel
Luiz Rocha
Andreas Öchsner

Table of Contents

I. Multiphase Flow

II. Diffusion, Osmosis and Phase Change

III. Forced Convection

IV. External Flows

V. Energy, Design and Optimization

VI. Learning and Teaching Techniques

I. Multiphase Flow

Defect and Diffusion Forum Vol. 348 (2014) pp 3-8
© (2014) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/DDF.348.3

Multiphase Flow and Heat Transfer in Risers

L.R.B. Sarmento[1, a], G.H.S. Pereira Filho[2,b], A.G.B. Lima[2,c], S.R. Farias Neto[1,d], E.S. Barbosa[2,e] and A.L. Cunha[1,f]

[1]Federal University of Campina Grande, Center of Science and Technology, Department of Chemical Engineering, Campina Grande, Brazil

[2]Federal University of Campina Grande, Center of Science and Technology, Department of Mechanical Engineering, Campina Grande, Brazil

[a]ligiarafaely@hotmail.com, [b]georgesarmento@gmail.com, [c]gilson@dem.ufcg.edu.br, [d]fariasn@deq.ufcg.edu.br, [e]enivaldo@dem.ufcg.edu.br, [f]actolimacunha@yahoo.com.br

Keywords: Heavy oil, pipeline, multiphase flow, heat transfer, numerical simulation.

Abstract. Multiphase flows commonly occur in the production and transportation of oil, natural gas and water. In this type of flow, the phases can flow in different spatial configurations disposed inside the pipe, so called multiphase flow patterns. The identification of flow patterns and the determination of the pressure drop along the pipe lines for different volumetric flows are important parameters for management and control of production. In this sense, this work proposes to numerically investigate the non-isothermal multiphase flow of a stream of ultraviscous heavy oils containing water and natural gas in submerged risers (catenary) via numerical simulation (ANSYS CFX 11.0). Results of the pressure, volumetric fractions and temperature distributions are presented and analyzed. Numerical results show that the heat transfer was more pronounced when using the largest volume fraction of gas phases.

Introduction

Considered of great importance in the oil industry, heavy oils constitute an important reserve to be exploited and produced. The heavy oil reserves in the world are estimated at three trillion barrels representing 15% of all world reserves. Given the enormous potential and reduction of conventional oil, commercial exploitation of accumulations of heavy oils in deep water represents a major technological and economic challenge for the oil companies, due to the high American Petroleum Institute gravity index, °API, (range 10° - 20°) and high viscosity (range 10^2 cP – 10^4 cP). All of these factors have an important effect on production and transportation, mainly of ultraviscous heavy oils in offshores. Multiphase flows are commonly encountered in the production and transportation of oil, natural gas and water. In this type of flow, the phases present can be arranged in different spatial configurations within the duct, called multiphase flow patterns [1]. The identification of flow patterns is essential for issues that are closely related to the economic return from the field as well as to determination of the pressure drop along the flow lines, measurement of volumetric flow rate transported, management and supervision of production [2]. These aspects are critical in conditions of offshore production, where large distances and high costs are involved. The study of the three-phase flow of heavy oil and water containing free gas presents as useful due to the high complexity and large number of flow patterns that can occur [3, 4].

In this sense, this study proposes to numerically investigate the non-isothermal multiphase flow of a stream of ultraviscous heavy oils containing water and natural gas in submerged risers (catenary type).

Geometry and Mesh

Figure 1 illustrates the three-dimensional structured mesh (800, 240 tetrahedral elements) generated in ICEM CFD, representing the physical domain of study. In this figure we can see refinement regions due to recirculation zones and the greater heat transfer in the vicinity of the walls.

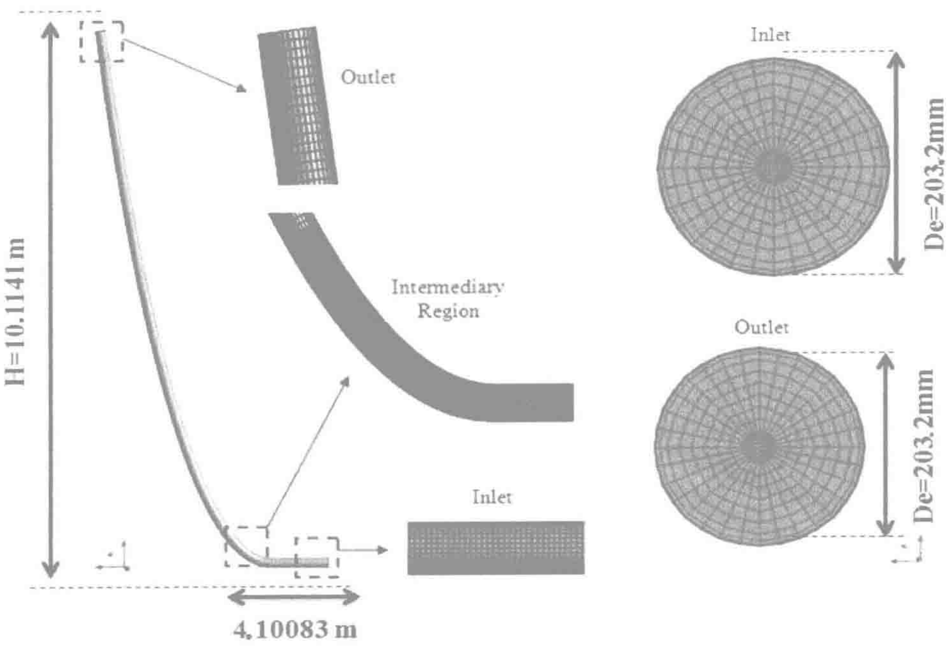


Figure 1. Geometrical parameters of the riser.

Mathematical Modeling

The following equations of the thermo-hydrodynamic of the multiphase flow were used.

Mass conservation equation:

$$\frac{\partial}{\partial t}(f_\alpha \rho_\alpha) + \nabla \bullet \left(f_\alpha \rho_\alpha \vec{U}_\alpha\right) = S_{MS\alpha} + \sum_{\beta=1}^{Np} \Gamma_{\alpha\beta} \tag{1}$$

Momentum conservation equation:

$$\frac{\partial}{\partial t}\left(f_\alpha \rho_\alpha \vec{U}_\alpha\right) + \nabla \bullet \left[f_\alpha \left(\rho_\alpha \vec{U}_\alpha \otimes \vec{U}_\alpha\right)\right] = -f_\alpha \nabla p_\alpha + \nabla \bullet \left\{f_\alpha \mu_\alpha \left[\nabla \vec{U}_\alpha + \left(\nabla \vec{U}_\alpha\right)^T\right]\right\} + \\ + \sum_{\beta=1}^{N_P}\left(\Gamma^+_{\alpha\beta}\vec{U}_\beta - \Gamma^+_{\beta\alpha}\vec{U}_\alpha\right) + \overrightarrow{S_{M\alpha}} + \overrightarrow{M_\alpha} \tag{2}$$

where α an β denote the involved phases (water, oil and natural gas), t is the time, f is the volumetric fraction, ρ is the density, $\vec{U}$ is the velocity vector, N_P is the number of phases, p is the pressure and μ is the dynamic viscosity.

Energy conservation equation:

$$\frac{\partial}{\partial}(r_\alpha \rho h_\alpha) + \nabla \bullet \left[r_\alpha \left(\rho_\alpha \vec{U}_\alpha h_\alpha - \lambda_\alpha \nabla T_\alpha\right)\right] = \sum_{\beta=1}^{Np}\left(\Gamma^+_{\alpha\beta} h_\beta - \Gamma^+_{\beta\alpha} h_\alpha\right) + Q_\alpha + S_\alpha \tag{3}$$

where h_α denotes the static enthalpy, λ_α is the thermal conductivity, T_α is the temperature of phase α and S_α describes external heat sources, and Q_α is the interphase heat transfer to phase α across interfaces with other phases. The term $(\Gamma^{+}_{\alpha\beta}h_{\beta s} - \Gamma^{+}_{\beta\alpha}h_{\alpha s})$ represents heat transfer induced by interphase mass transfer. Further, in this research we use the k-ε turbulence model.

Initial and Boundary Conditions. The properties of the fluids used in the simulation are presented in Table 1.

Table 1. Parameters of the fluids a boundary conditions.

Parameters	Water (Disperse)	Heavy oil (Continuous)	*Natural gas (Disperse)
ρ [kg/m^3]	997	989	766
bubble diameter [mm]	8	-	3
T [K]	333	333	333
k [W/m.K]	0.6069	0.147	0.809
μ [Pa.s]	$\mu_w = 0{,}00002414x10^{\left(\frac{247.8}{T-140}\right)}$	$\mu_o = 5{,}187e\left[-2{,}3935\left(\frac{T-273}{573-273}\right)\right]$	$\mu_g = 0{,}0744x10^{\left(\frac{7}{177-140}\right)}$
σ [N/m]	0.045 (water/heavy oil)		
	0.015 (natural gas/heavy oil)		
h [W/m^2.K]	30		
inlet velocity [m/s]	2		

Boundary Contours	Values		
	f_w	f_g	f_o
Case 1	0.4	0.1	$1-(f_w+f_g)$
Case 2	0.1	0.4	$1-(f_w+f_g)$

*Natural Gas (Natural Gas Composition: CH_4=0.90; C_2H_6=0.05; C_3H_8=0.025; C_4H_{10}=0.025)

Results and Discussion

Pressure Drop. Fig. 2 shows the pressure drop at the longitudinal center of the catenary as a function of Z, where it senses a total pressure drop for z between 1 and 4 m.

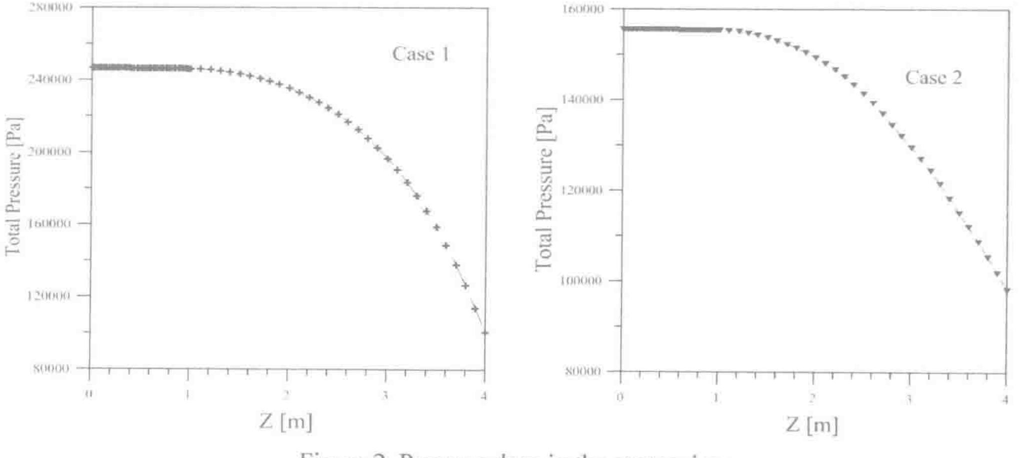
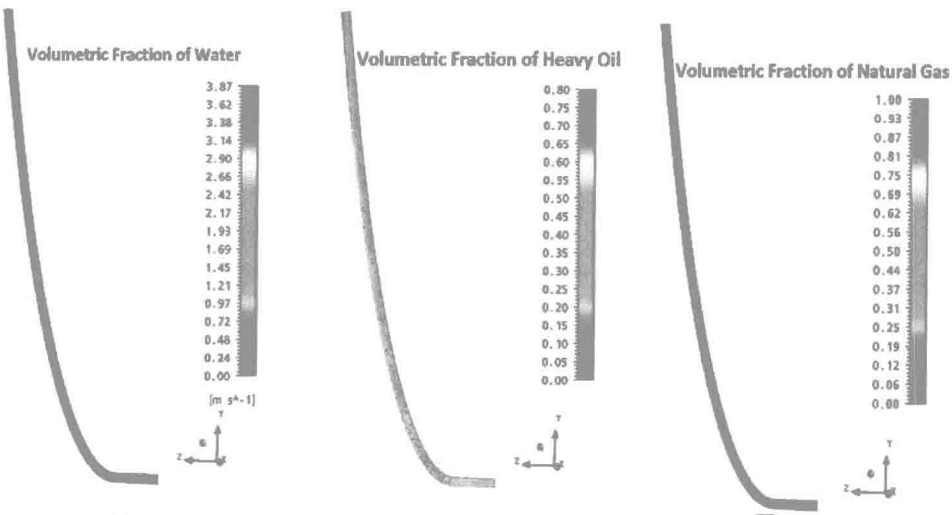


Figure 2. Pressure drop in the center riser.

It can be seen that the total pressure drop is more pronounced for the case where the gas fraction is smaller (Case 1). This is because the smaller amount of gas increases the density of the mixture and pressure losses through friction. Fig. 2 shows the maximum pressure inlet and minimum pressure outlet for two cases. For Case 1 was required a pressure differential = 149900 Pa, and in Case 2 was required a pressure differential = 58900 Pa to drain the three phases. This difference is the barrier needed to transport and lift the heavy oil and ultraviscous water and natural gas.

Analysis of the Volumetric Fraction Fields and temperature. Fig. 3 shows the volume fraction in a ZY-plane for the phases involved. The heavy oil is at the center of the pipe while the other phases more close to the walls.



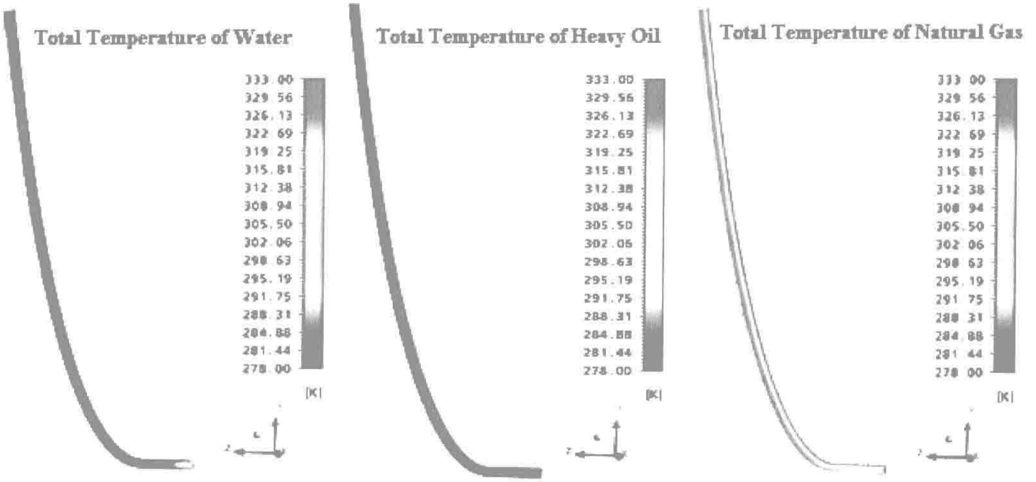
(a) (b) (c)

Figure 3. Volume fractions of the phases for the case 1, $f_g = 0.1$.

Fig. 4 (a) shows the rapid growth of the thermal boundary layer of water that occurs due to the low Prandtl number, Fig. 4 (b) there isn't an increase in the apparent thermal boundary layer due to the high Prandtl number of the oil. In Fig. 4 (c) there is an increase in the thermal boundary layer due to the low Prandtl number.

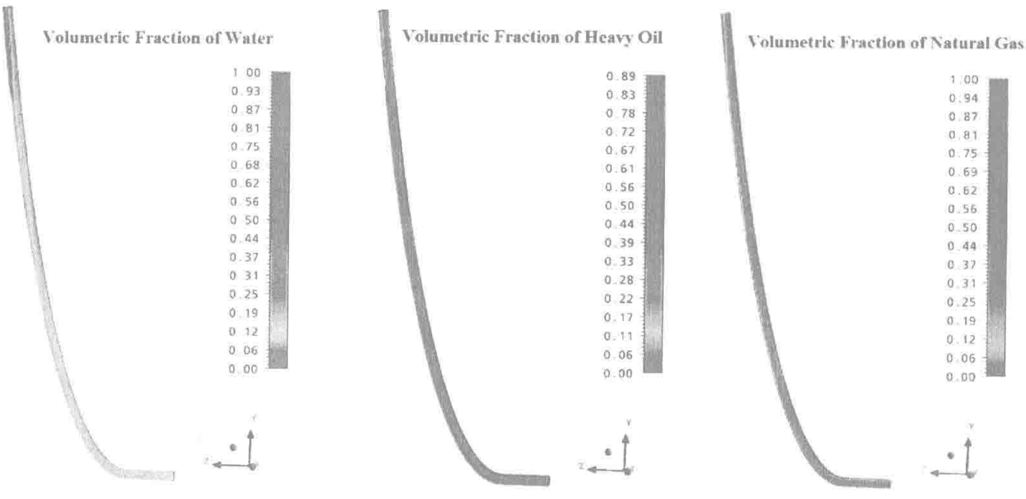
Analogously to Fig. 3, Fig. 5 shows the volume fraction in the *ZY* plan to phases involved, where heavy oil has a preferred way through the central pipe and other phases have a preferred way for the periphery of the pipe. However there is a greater mixing region due to turbulence caused by the gas.

Fig. 6 (a) shows the rapid growth of the thermal boundary layer of water that occurs due to the low Prandtl number, in Fig.6 (b) there is a thermal boundary layer growth visible due to increased turbulence caused by gas causing a rise of the oil temperature. In Fig. 6 (c) there is an increase in the thermal boundary layer due to the low Prandtl number.



(a) (b) (c)

Figure 4. Temperature of the phases for the case 1, $f_g = 0.1$.



(a) (b) (c)

Figure 5. Volume fractions of the phases for the case 2, $f_g = 0.4$.

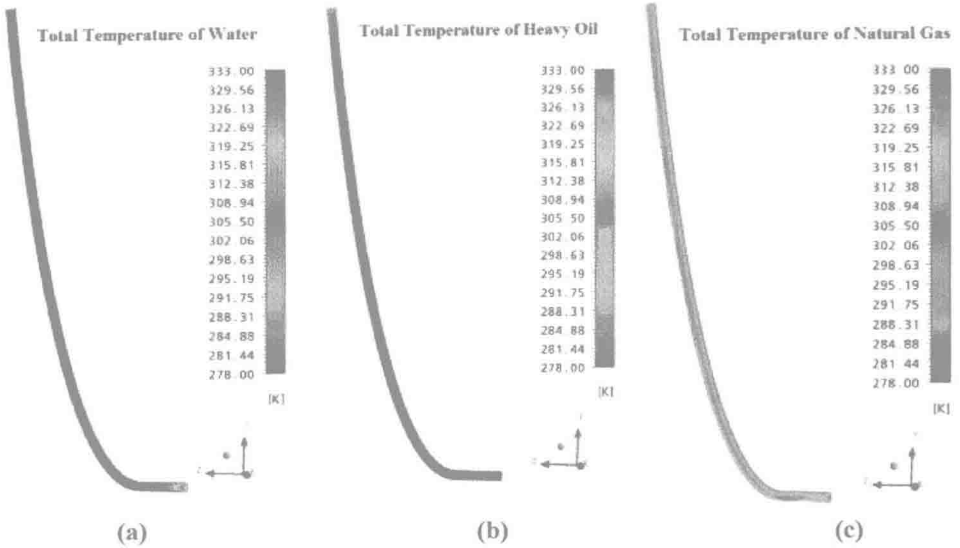


Figure 6. Temperature of the phases for the case 2, $f_g = 0.4$.

Conclusions

From the results presented and discussed, it can be concluded that the pressure loss in the flow using higher fraction of natural gas is lower than when using lower gas fraction, due to the lower density. The results of the simulation showed that the heavy oil has a preferred way through the center of the pipe and the other phases have a preferred way for the periphery of the pipe. Although we found that the heat transfer was more pronounced when using the higher volume fraction of gas.

Acknowledgments

The authors would like to express their thanks to Brazilian Agencies: PETROBRA S/A, UFCG-PRH-25, ANP, CNPq, CAPES, FINEP and JBR Engenharia Ltda. for supporting this work and are also grateful to the authors of the references in this references in this paper that helped in the improvement of quality.

References

[1] E.E. Paladino, Study of Multiphase Flow in Flowmeters TYPE Differential Pressure, PhD Thesis, Federal University of the Santa Catarina, 2005.

[2] M. Bonizzi, R.I, Issa, On the simulation of three-phase slug flow in nearly horizontal pipes using the multi-fluid model, International Journal of Multiphase Flow 29 (2003) 1719-1747.

[3] G. Oddie, H. Shi, L.L. Durlofskys, K. Aziz, B. Pfeffer, J.A. Holmes, Experimental study of two and three phase flows in large diameter inclined pipes, International Journal of Multiphase Flow 29 (2003) 527-558.

[4] A. Wegmann, J. Melke, P.R. Rohr, Three phase liquid-liquid-gas flows in 5.6 mm and 7 mm inner diameter pipes, International Journal of Multiphase Flow 33 (2003) 484-497.

[5] G.H.S. Pereira Filho, Multiphase Flow and Heat Transfer in Riser. Graduate thesis, Federal University of Campina Grande, 2010.

Defect and Diffusion Forum Vol. 348 (2014) pp 9-19
© (2014) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/DDF.348.9

A Two-Phase Flow Solver for Incompressible Viscous Fluids, Using a Pure Streamfunction Formulation and the Volume of Fluid Technique

Raphaël Comminal[1,a], Jon Spangenberg[2,b] and Jesper Henri Hattel[1,c]

[1]Department of Mechanical Engineering, Technical University of Denmark, Produktionstorvet, 2800 Kgs. Lyngby, Denmark

[2]Civil and Environmental Engineering, Princeton University, Princeton, NJ-08540, USA

[a]rcom@mek.dtu.dk, [b]js43@princeton.edu, [c]jhat@mek.dtu.dk

Keywords: Multiphase flow; streamfunction formulation; volume of fluid; Rayleigh-Taylor instability; broken dam problem

Abstract. Accurate multi-phase flow solvers at low Reynolds number are of particular interest for the simulation of interface instabilities in the co-processing of multilayered material. We present a two-phase flow solver for incompressible viscous fluids which uses the streamfunction as the primary variable of the flow. Contrary to fractional step methods, the streamfunction formulation eliminates the pressure unknowns, and automatically fulfills the incompressibility constraint by construction. As a result, the method circumvents the loss of temporal accuracy at low Reynolds numbers. The interface is tracked by the Volume-of-Fluid technique and the interaction with the streamfunction formulation is investigated by examining the Rayleigh-Taylor instability and broken dam problem. The results of the solver are in good agreement with previously published theoretical and experimental results of the first and latter mentioned problem, respectively.

Introduction

The simulations of multiphase and free-surface flows are of particular interest for the modeling of manufacturing processes involving molten materials, such as in metal casting or polymer molding. Free-surface flow simulations are useful to predict manufacturing defects related to improper mold filling. In addition, the development of advanced co-processing manufacturing techniques for multilayered composite structures and functionally graded materials (e.g. co-extrusion, co-injection molding, multilayer tape casting, etc.) requires the control of stratified flows. Depending on the flow conditions, the interface between immiscible fluids can be subjected to interfacial instabilities and deformations. In co-extrusion for instance, the mismatches of the rheological properties induce secondary flows which result in the deformation of the interfaces. Those phenomena are known as viscous encapsulation and elastic rearrangement [1]. In general, interfacial instabilities are generated when the two materials have different rheological properties, which lead to a discontinuity in the shear-rate across the interface. In this context, Computational-Fluid-Dynamics is a powerful tool to understand the kinetics of interfaces, and to improve the quality of production.

The main difficulty associated with incompressible flow is that the pressure field is not a state variable of the fluid. It is a dynamical parameter which does not have an evolution equation. In order to alleviate the difficulties with the velocity-pressure coupling, incompressible flows are generally solved with fractional-step methods, first introduced by Chorin [2]. It consists in splitting the Navier-Stokes problem into a general convection-diffusion equation for the momentum, and a Poisson equation for the pressure [3]. The semi-implicit methods (SIMPLER, SIMPLEC and PISO) can also be classified as fractional step method as they lay on the same idea of velocity-pressure decoupling [4]. Fractional step methods have demonstrated their efficiency to solve a wide range of aerodynamic and hydrodynamic problems, usually characterized by Reynolds numbers above unity. However, those methods present some disadvantages in the context of low-Reynolds number flows, typical of polymer processing and microfluidic. Some of those disadvantages are listed below:

1) The decoupling between the velocity and pressure introduces splitting errors. Some methods can ensure a certain reduction rate of the splitting error when the time-step increment is reduced. However, an analysis of the splitting error with a Taylor series shows that the dominant error term is proportional to $(1/Re)^{\alpha}$, where α is a natural number [3]. As a consequence, the temporal accuracy of fractional step methods is severally degraded when $Re < 1$. Therefore, the convergence of the calculation might require a large number of iterations, and drastically small time-step increments.

2) The solution from the fractional-step methods does not fulfill the mass conservation to the machine precision [3]. Since the continuity equation is not solved directly, the accuracy of the mass conservation depends on the condition number of the Jacobi matrix for the velocity equation. It is a rule of thumb that the numerical results lose precision on a number of digits equal to the order of magnitude of the conditioning number. At low Reynolds number, the momentum equation is dominated by diffusive terms, and it results in a Jacobi matrix with very high condition number. Therefore, when $Re < 1$, the accuracy of the numerical results (and the mass conservation) is degraded [5].

3) The pressure equation requires boundary conditions which are not known a priori. There have been debates about the correct boundary condition to be used [6,7]. Moreover, its value at the boundary does not necessarily correspond to the physical pressure at the wall. It is rather a factitious value calculated so that the normal pressure gradient is compatible with the shear-rates at the wall. The uncertainty on the correct boundary condition can have dramatic effects, since the pressure equation is a boundary value problem (elliptic equation).

4) Errors in the pressure field have large impact on the motion of the interface. Indeed there is a coupling between the pressure field and the position of the interface. On one hand, multiphase flows at low Re numbers are driven by pressure gradients. (Viscous forces act as resistance forces). On the other hand the pressure field is directly related to the density field, which is determined by the interface position.

In this paper, we use an algorithm based on a pure streamfunction formulation, in order to avoid the disadvantages of the fractional step methods. The pure streamfunction formulation is more accurate than the fractional-step methods because it does not have any splitting errors. By construction it fulfills the mass conservation to the machine accuracy, without solving the continuity equation. In addition, it reduces the number of unknowns as it avoids the calculation of the pressure [8]. The pure streamfunction formulations have already been used for the simulation of single-phase flows [8,9]. The novelty of our method lies in the use of the streamfunction formulation for the simulation of multiphase flows of incompressible fluids. The Volume of Fluid (VOF) technique is used to track the position of the interface. The rest of this paper describes the derivation of the streamfunction evolution equation, and the numerical techniques used to solve it. We present results of 2D test case simulations that have been done to validate the implementation of our algorithm.

Governing Equations and Numerical Techniques

Derivation of the Streamfunction Formulation. The two-phase flow is modeled as a continuum single media, governed by the Navier-Stokes equations

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

$$\frac{\partial(\rho \boldsymbol{u})}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}\boldsymbol{u}) = -\nabla p + \nabla \cdot (\mu \dot{\varepsilon}) + \rho \boldsymbol{g} \tag{2}$$

where $\boldsymbol{u}$ and the velocity vector, p is the isostatic pressure, ρ is the density, μ is the dynamic viscosity, g is the gravitational acceleration, and $\dot{\varepsilon}$ is the deformation rate tensor defined as