

16:00	OR30	Theoretical Study And Numerical Simulation Of Wavy Stratified Oil-Water Flow	PDF
16:50	OR31	Axial Development Of Bi-Directional Liquid Velocity Gas-Liquid Flow Patterns In A Long Vertical Tube	PDF
17:10	OR32	Flow Regime Identification In Natural Circulation Phenomenon Using Relap5 Code	PDF
17:30	OR33	Turbulent Flow Behavior Of Viscous Oil-Water Dispersions	PDF
17:50	OR34	Experimental Investigation Of Gas Presence In The Oil-Water Core-Flow Pattern Using Visualization Technique	PDF

POSTER PRESENTATION

PO1	Prediction Of Distribution Parameter And Drift Velocity In A Upward-Vertical Dispersed Two-Phase Flow In A Large Diameter Annular Duct	PDF
PO2	Hydrocarbons Solubility In Non Aqueous Fluids Impact In Influxes Detection While Drilling	PDF
PO3	Hydrodynamic Aspects Of Two-Phase Flow Across Tube Bundles	PDF
PO4	Review On Applications Of Wire-Mesh Sensor In Multiphase Flows	PDF
PO5	Comparative Study Of The Superficial Velocity Of Air Bubbles In Horizontal Two-Phase Flow: Level Set Method And Wire-Mesh Sensor	PDF
PO6	Experimental Validation Of The Algorithm For Bubbles Identification From Wire-Mesh Sensor Data	PDF
PO7	Identification Of Patterns Two-Phase Flow Using Theoretical And Experimental Data And Artificial Neural Networks	PDF
PO8	Experimental Investigation Of Spray Cooling Of Plain And Enhanced Surfaces	PDF
PO9	Flow Boiling Instabilities In Microchannels	PDF

THEORETICAL STUDY AND NUMERICAL SIMULATION OF WAVY STRATIFIED OIL-WATER FLOW

Ricardo Pereira de Ávila¹, Lucas Soares Baldani², Oscar Maurício Hernandez Rodriguez³

Department of Mechanical Engineering, School of Engineering of São Carlos, University of São Paulo
Av. Trabalhador São-carlense, 400 - São Carlos - SP, CEP 13566-590
¹ricpavila@usp.br, ²baldanils@gmail.com, ³oscarmhr@sc.usp.br

ABSTRACT

Two-phase stratified flow has been used as a convenient way to avoid the formation of emulsions of water in oil in pipelines, and it has a common occurrence in directional oil wells. Energy losses and high costs related to the transport of fluids have been subject of concern for the industry. Once ensuring the stability of the flow pattern in specific conditions, it is possible to reduce energy consumption and significantly increase the production, also reducing the costs, both design and operational. The theory of hydrodynamic stability has been widely used for modeling and prediction of the transition boundaries of two-phase flow patterns. According Wallis (1969), in some cases, flow pattern transitions can be assigned to hydrodynamic instabilities resulting from the amplification of a disturbance wave at the phases' interface. In this work, we suggest that the interfacial wave is the result of the balance of forces acting on the two-phase flow. CFD is applied in order to try to understand more deeply the interfacial phenomena related to the interfacial shear stress term and, ultimately, better predict pressure drop and in-situ volume fraction. Preliminary results show good agreement between the experimental and CFD results, both in the direct comparison between the data of volume fraction and pressure drop. A qualitative analysis of the wave's geometrical properties is also carried out.

INTRODUCTION

Two-phase stratified flow has been used as a convenient way to avoid the formation of emulsions of water in oil in pipelines, and also have a common occurrence in directional oil wells. The energy losses and high costs related to the displacement of fluids have been subject of concern for the industry, especially in conditions of offshore production. Once ensuring the stability of the flow pattern in specific conditions, it is possible to reduce energy consumption and significantly increase the production, also reducing the costs, both project and operational.

The theory of hydrodynamic stability has been widely used for modeling and prediction of the transition boundaries of two-phase flow pattern. According [13], in some cases flow-pattern transition can be attributed to instability resulting from the amplification of an interfacial disturbance wave. However, little attention has been paid to the interfacial wavy structure observed in stratified oil-water flow. Also, the poor modelling of the interfacial energy dissipation can be an explanation for the unsatisfactory prediction of pressure drop by one-dimensional models available in the literature and adopted by commercial codes, [4], [5]. A work was recently published about the geometric and kinematic properties of interfacial waves in stratified oil-water flow, [6]. In this work, they suggest that the wave has a kinematic nature, i.e, it is a result of the force balance. It is worthy of note that the formidable complexity of two-phase flow does not allow the analytical solution of the general governing equations. Thus, one of the options to access the details of the flow is the use of numerical methods in *CFD* (Computational Fluid Dynamics), which have been able to detail the flow in different circumstances. In this prospective study, CFD is used in an attempt to understand in greater depth interfacial phenomena that are related to the term of interfacial shear stress, as well as it is evaluated as a tool to predict the pressure drop and

volume fraction. Ultimately, this research aims to develop more complete closure equations for using in one-dimensional two-fluid models for prediction of pressure drop in liquid-liquid stratified flow.

METHODOLOGY

Experimental Work

The experimental work of [11] was conducted in the Multiphase-Flow Laboratory of the Thermal-fluids Engineering Laboratory (NETeF), Engineering School of São Carlos (EESC) of the University of Sao Paulo (USP) in 12m in length and 26mm in diameter horizontal borosilicate glass pipe.

Pressure drop data were collected by a differential pressure transducer Validyne, Model DP-15. It was used the diaphragm 34 with a range of scale of 22 kPa and uncertainty of 0.5% of full scale. The acquisition rate used was 5 kHz and data were processed and analyzed in a program implemented in Labview® platform (see also [5]). The pressure drop was taken via pressure taps 7.5 m apart from each other.

All data collected during testing reached a remote workstation consisted of a microcomputer and an acquisition board NI-PCI-6224, brand National Instruments®. An acquisition rate of 200 kHz and a resolution of 4096 measurement points or 12 bits, which can be amplified up to 50 times. It has 24 digital ports, used for the actuation of quick-closing valves, 8 differential inputs, used to connect all the instrumentation, and 2 analog outputs. The actuators that control the flow of water, air and oil are initiated via a low-frequency CAN system. A data acquisition program was implemented in Labview® platform and was used for the automatic acquisition of the values measured by flow meters, pressure and temperature meters.

The *in-situ* volume fraction of water and oil were measured using the technique of quick-closing valves, a synchronized system of solenoid valves and *bypass* line allowed the application of the technique.

The measurement of the geometric properties and interfacial wave speed were made with the aid of a high speed and high definition camera (1000 frames/s) model Optronis Camrecord-600. The camera was installed in a holder designed to stay fixed to the movable structure, regardless of the inclination angle. For lighting we used 2 xenon lamps positioned behind the visualization section.

To obtain the data through the high-resolution images it was developed a software in Labview® platform able to identify the wave corresponding to the oil-water interface and predict its width, length and speed from a process of binarization of the images.

Therefore, wave properties, *in-situ* volume fraction and pressure gradient data were collected at five angles θ of inclination (-20° , -10° , 0° , 10° and 20°), measured by a digital inclinometer brand Bosch DNM 60L model.

The densities of each fluid were measured with the use of a 5 ml pipette and a precision scale with 0.0005g precision, 5 measurements were made for each sample, with mean values: $\rho_o = 824 \text{ kg/m}^3$ for oil density, a standard mineral oil (Shell Vitrea 100); and $\rho_w = 988 \text{ kg/m}^3$ for the density of the process water.

The dynamic viscosity of the oil μ_o was measured using a Brookfield rheometer brand, model LVDV-III+ with SC4-18 rotor. The viscosity was measured as a function of oil temperature between 10°C and 50°C . It was of $\mu_o = 250 \text{ mPa.s}$ at 25°C .

The oil-water interfacial tension σ was measured using a tensiometer optical KSV brand, model MAC 200 which consists of a system formed by a chamber where a fluid is placed and one drop is generated by a syringe and needle. The drop is filmed and the image is transferred to a computer program that determines the size and measuring the interfacial tension by the Young-Laplace model. The procedure consisted of 4 measurements for each value of oil temperature between 10°C and 40°C . It was of $\sigma = 0.03 \text{ N/m}$ at 25°C .

The contact angle was also measured using the same optical tensiometer KSV brand, model CAM 200 already mentioned, with process water, a mixture of oil and water in an aquarium comprising plates of borosilicate glass, the same material of the pipe, 15 experiments have been made to do so, and 30° was the measured mean contact angle (hydrophilic/oleophobic).

Mathematical Models for the Closure Equations

In order to verify the reliability of the CFD results, not only the present experimental data, but also phenomenological models based on the 1-D two-fluid model were used for comparison purposes: the homogeneous model, the Rodriguez and Oliemans' model [9] and the Pereira model [12]. The flow was considered isothermal, incompressible, steady and no phase change or mass transfer was taken into account.

The homogeneous model, [13], considers the stratified flow as a homogeneous mixture or a pseudo-fluid with average properties, obeying the equations of single-phase flow.

In the model of [9], based on the model of [10], the wet perimeters of each phase and interface, and the cross-sectional areas of each phase are calculated by assuming a flat

interfacial geometry. The wall shear stresses of water and of oil and the interfacial shear stress are functions of *in-situ* phases' velocities and corresponding friction factors. The laminar friction factors correspond to the same of single-phase flow, but with respective hydrolic diameters; the turbulent flow friction factors are calculated by the correlation of Haaland [10]. The interfacial friction factor is assumed to be the wall friction factor of the fastest phase.

In the model proposed by [11], the wet perimeters of oil, water and interface and the cross-sectional areas of each phase are calculated considering a concave or convex interface geometry, corresponding to an arc of a circle as a function of the contact angle, volume fraction and the Eötvös number. It is assumed that the interfacial shear is function of the wavy structure and, therefore, it would be possible to model the energy dissipation related to the wavy interface by proposing an effective interfacial roughness. Therefore, the interfacial tension would be greater than that predicted by the model used in [9]. Furthermore, it was suggested that the effective roughness is proportional to the interfacial wave amplitude α in wavy stratified oil-water flow. Thus, the new interfacial friction factor f_i is given by:

$$f_i = f_f \left(1 + C_k \frac{\alpha}{D} \right) \quad (1)$$

where C_k is a parameter to be adjusted empirically, the friction factor of each phase f_f can be either f_w or f_o , of the water and oil, respectively, depending on the velocities of the phases, and D is the internal diameter.

It is an aim of this work to improve the modeling of stratified oil-water flow from a best fit of the interfacial friction factor f_i given by:

$$f_i = f_f [1 + f_r(\alpha) + g_r(\lambda)] \quad (2)$$

where $f_r(\alpha)$ and $g_r(\lambda)$ are functions dependent on, respectively, the amplitude α and the wavelength λ of the interfacial wave.

To this end, a consistent theoretical study has to be carried out, which includes, as an important part of the work, CFD simulations.

Computational Fluid Dynamics

CFD aims primary solving numerically nonlinear partial differential equations used to describe the flow of fluids, heat transfer, and turbulence, since these equations, by the characteristic non-linearity, have no known analytical solution. Multiphase flow has been one of the most challenging CFD problems and intense research has been devoted to it over the last decades. The formidable complexity of multiphase flows usually translates into a high time-consuming and computational expensive problem. Multiphase flows can consist of two, three or even more components, with astonishingly high interfacial interactions, together with turbulence, chemical reactions, as combustion, and phase change. Therefore, the modelling and the development of closure relations are still important research issues and they should be for a very long time, even considering the continuing growth of computing capability.

In this work, we decided for the use of a commercial code and followed the steps below:

- creation of geometry using the software Modeler Desing 12.0®, the mesh generation software Ansys Icem CFD® and definition of the physical model, i.e., initial and boundary conditions and models for the numerical solution at CFX-Pre 12.1®;

- the solution of the problem of CFD, which corresponds to the process of calculations involved to solve the governing equations, using the ANSYS CFX-Solver®;

- analysis and visualization of results with CFX-Post 12.1®.

The simulations were performed at the cluster Inepar of the company Andritz Hydro, which has provided official licenses of the commercial software mentioned above. It was used a computer operating system Linux - Red Hat, 44 cores, 3.0 GHz and 96 GB RAM through 11 CPU's. Two different meshes were tested and used in the simulations. The processing time for the simulations with the coarse mesh, with maximum distance of 0.75mm between nodes, with approximately 400,000 nodes for a pipe length of 1m, was approximately 48 hours; and with the fine mesh, with maximum distance between nodes 0.25mm, with approximately 1.5 million nodes for a 1m length of pipe, it was 300 hours.

As hypothesis: Newtonian fluids, two-fluid model, being the oil flow in the laminar regime and the water flow in the turbulent regime, transient, incompressible, isothermal, three-dimensional, symmetrical, no mass transfer between the phases and gravity acting.

Due to the symmetry condition, it was chosen a hybrid mesh refined in the wall with seven layers of prisms calculated using the criterion Y^+ around 10-30, value range acceptable for turbulent flow when using the $k-\epsilon$ model, the extent of which varies from 0 to 150.

The physical properties of oil and water used in the simulation were: density of oil $\rho_o = 824.32 \text{ kg/m}^3$, dynamic viscosity of the oil $\mu_o = 214.622 \text{ mPa.s}$, density of water $\rho_w = 997 \text{ kg/m}^3$ and dynamic viscosity of water $\mu_w = 0.8899 \text{ mPa.s}$.

As boundary conditions: mass flow in both the pipe entrance and exit, no-slip condition in the pipe wall and Symmetry.

The basic configuration for the simulations were: oil was set as the reference buoyance density fluid and both fluids being continuous, the fluid model with the option of *Free Surface* without heat exchange, thermal radiation or combustion. For the turbulence models, the option *Fluid Dependent* was used, using *Laminar* model for the oil and $k-\epsilon$ model for the water. A constant surface tension equal to 0.0275 N/m and a drag coefficient of 0.44 were used.

For transient flow it was set for the coarse mesh a time step of 0.001s and for the fine mesh a time step of 0.00005s. These values were selected so as to obtain, during the simulation, a considerably small value for the Courant number, in order to obtain a solution as precise as possible.

Based on the initial hypothesis described above, it was adopted an Eulerian mathematical model, in which the fluid is considered continuous and interpenetrating, with the volume fraction being the variable which quantifies each phase. As it is an isothermal problem, the governing equations are expressed by the equations of mass and conservation of momentum.

The closure relations used in the simulations, as shear stresses of each phase, interfacial shear and drag, virtual mass force, etc., were those currently available in ANSYS Solver-CFX®.

RESULTS

Quantitative comparisons of *in-situ* volumetric fraction and two-phase pressure gradient

Figure 1 shows a comparison between the experimental data of *in-situ* oil volume fraction and the following models predictions: the model proposed by [9], homogeneous model, the model proposed by [11] and the results obtained by CFD for inclination (a) horizontal, (b) 10° downward and (c) 20° downward, in which U_{ws} corresponds to the water superficial velocity, U_{os} the oil superficial velocity and ϵ_o the *in-situ* oil volume fraction.

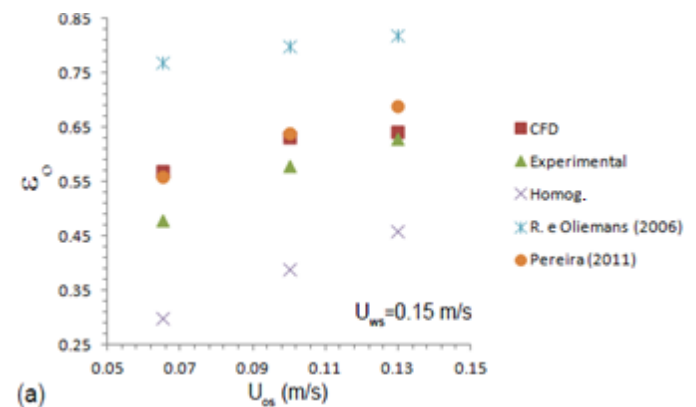
We note in Fig.1 that the data were underestimated by the homogeneous model and overestimated by Rodriguez and Oliemans' model [9]. In all cases, the model proposed by [11] gave better predictions and the values obtained by CFD simulation showed the best results.

As can be seen in a direct comparison between the data and the predictions of the model proposed by [11], there was an overall average relative error of 14.2%, whereas for the Rodriguez and Oliemans' model [9] it was of 40.4% and for the homogeneous model of 39%. The results of CFD simulations presented an overall average relative error of only 7.43%. The agreement between Pereira's model [11] and CFD's are quite satisfactory, and therefore considered a good result in view of the complexity of wavy stratified two-phase flow. Nevertheless, the model of [9] offered rather unsatisfactory predictions. The overall average relative error was calculated using the expression:

$$err_{holdup o} = \frac{1}{N} \sum_i^N \sqrt{[(\epsilon_{theo} - \epsilon_{exp})/\epsilon_{exp}]^2} \cdot 100 \quad (3)$$

ϵ_{theo} represents the *in-situ* volume fraction predicted by the models and ϵ_{exp} the *in-situ* volume fraction measured experimentally.

Figure 2 shows comparisons between the pressure gradient obtained experimentally and predicted by: the model of [9], the homogeneous model, the model of [11] and CFD results for (a) horizontal, (b) 10° downward and (c) 20° downwards as a functions of the oil superficial velocity. Overall, the experimental data were systematically overpredicted by all models.



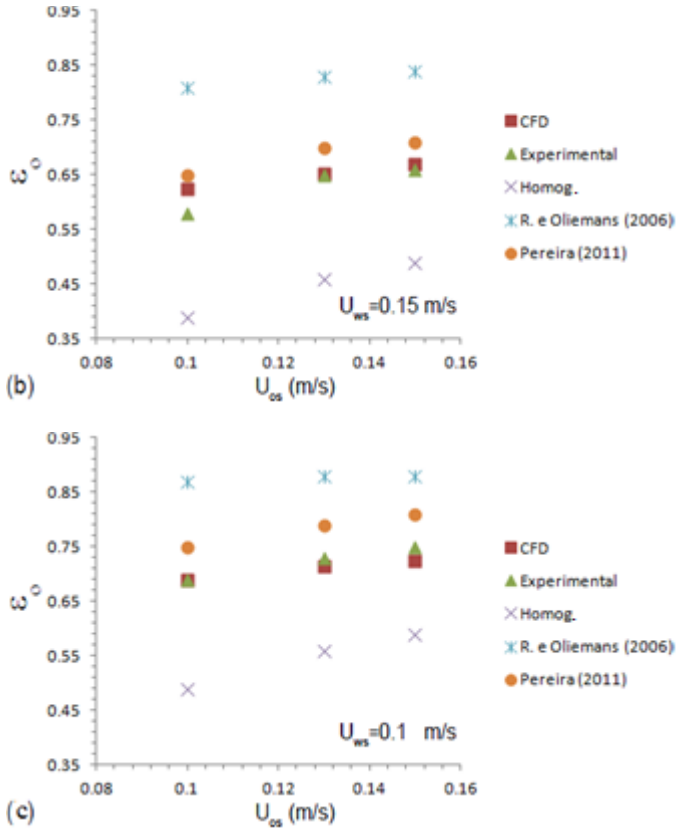


Fig.1 – *In-situ* oil volumetric fraction as a function of superficial velocity of the oil: experimental data, homogeneous model, model of Rodriguez and Oliemans (2006), model of Pereira (2011) and CFD results for (a) horizontal, (b) -10° , (c) -20° inclined flow.

The model proposed by [11] approached better in most cases, when compared with the theoretical models. The CFD results presented the best agreement, although rather unsatisfactory. The pressure drop predicted by models of [11], [9], homogeneous and CFD showed an overall average relative error of 70.6%, 82.4%, 106.0% and 66.0% respectively. To calculate the overall average relative error was used the following expression:

$$err_{pressure} = \frac{1}{N} \sum_i^N \sqrt{[(DP_{theo} - DP_{exp})/DP_{exp}]^2} \cdot 100 \quad (4)$$

where DP_{theo} represents the pressure drop predicted by the models and DP_{exp} the pressure drop measured experimentally.

The comparisons suggest an investigation focused on the closure relations. Although the volume fraction data were satisfactorily predicted by one of the theoretical models and CFD, the same cannot be said about the pressure drop predictions. Even the predictions of numerical simulation were not really satisfactory, suggesting the need for a more detailed investigation of two-phase flow models used in ANSYS CFX-Solver. The results, according to Fig.2, suggest that the error could be associated to the frictional pressure gradient term.

Figure 3 shows the evolution of the two-phase flow in time, at 10s and 60s. In both figures it can be observed that close to the pipe inlet there is a smooth interface, while after a few diameters the wavy interface is fully developed. This behavior of the flow in the tube was experimentally observed and recorded on video, with pairs of flow rates similar to those used in the numerical simulation.

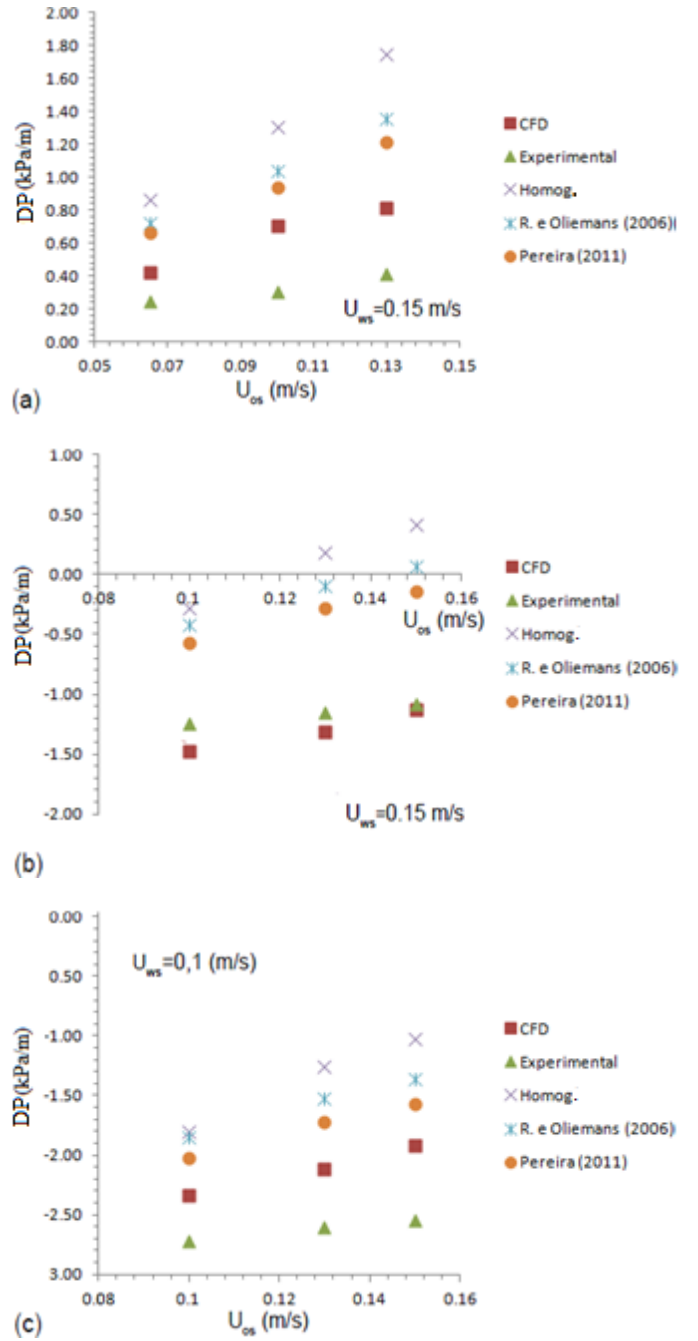


Fig.2 - Two-phase pressure gradient as a function of superficial velocity of oil: experimental data, homogeneous model, Rodriguez and Oliemans' model (2006), model of Pereira (2011) and CFD results for (a) horizontal, (b) -10° (c) -20° inclined flow.

To calculate the pressure drop, two numerical probes were positioned after the wavy interface was developed and spaced 0.25 m from each other, the first was positioned 0.62 m from the pipe inlet. The numerical results were also used to estimate the value of the in-situ volumetric fraction.

Thus, using the numerical results of volume fraction, it was possible to calculate the total pressure drop or pressure gradient from the data collected via the differential pressure transducer. Eq. 5 and 6 were used for that purpose.

$$\Delta P = [\alpha \rho_o + (1 - \alpha) \rho_w] g L \sin(\beta) + \Delta P_{F+A} \quad (5)$$

Qualitative analysis of the geometrical properties of interfacial waves

A qualitative comparison of the results obtained by CFD simulations with data obtained experimentally by [11] regarding the interfacial characteristics of the wave can be seen in Fig.5. The wave profiles observed experimentally are quite similar to those obtained by numerical simulation. The result is encouraging.

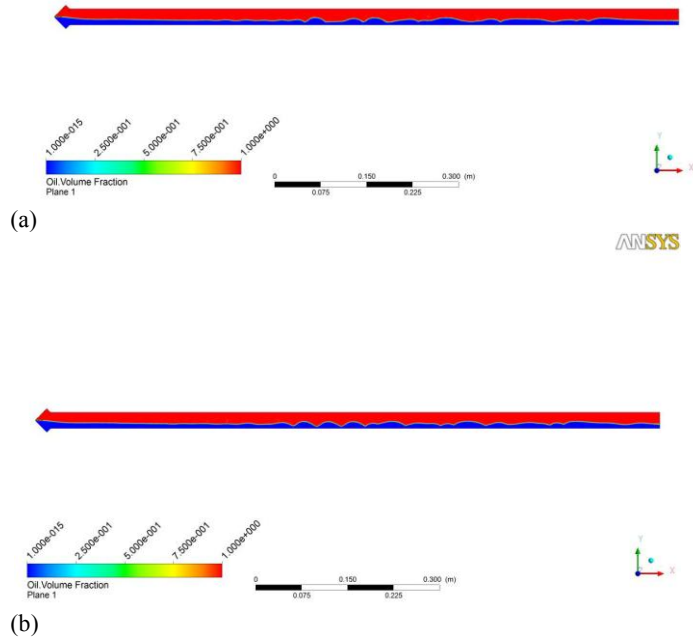


Fig.3 - Results of the simulation for (a) t = 10s, and (b) t = 15s.

$$\Delta P' = [\alpha(\rho_o - \rho_w)]g L \sin(\beta) + \Delta P_{F+A} , \quad (6)$$

where ΔP stands for the total pressure drop provided by numerical simulation [Pa], α the fraction of oil supplied by numerical simulation, ρ_o the density of the oil phase [kg/m³], ρ_w the density of the water phase [kg/m³], g the gravitational acceleration [m/s²], L the length of the test section [m], β the angle of inclination of the test section [°], ΔP_{F+A} the portion of the pressure drop represented by the friction and acceleration [Pa], and $\Delta P'$ the total pressure drop given by the Validyne® differential pressure transducer [Pa].

In Fig.4 one can see, for both virtual pressure taps, the development of the numerical pressure signal over time. The numerical data were computed only after reaching steady state.

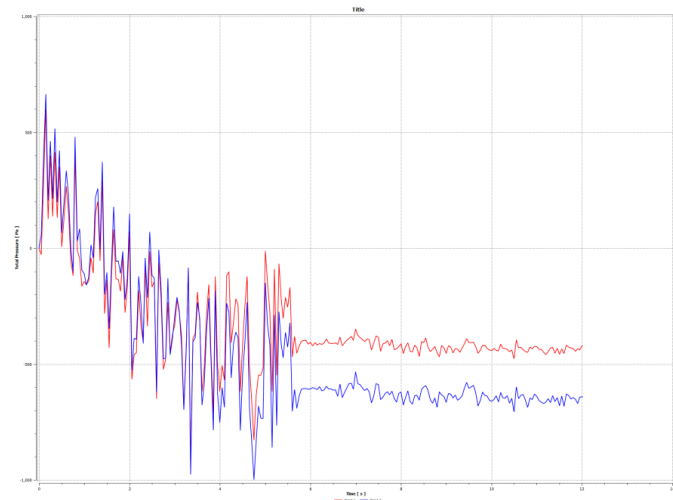
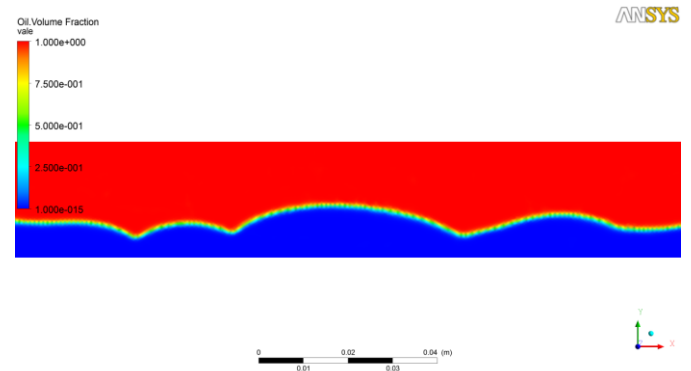


Fig.4 - Pressure versus time for both virtual pressure taps at horizontal flow and $U_o = 0.1m/s$ e $U_w = 0.15m/s$.



(a)



(b)

Fig.5 - Wavy profile (a) observed experimentally and (b) obtained via CFD.

A concave interface was predicted by [11] and the adoption of such interfacial geometry gave better results compared to other phenomenological models available in the literature. As one can see in Figs.6 and 7, the CFD simulations predicted a concave interface. Even more interesting is the observation of the variation of the cross-sectional interfacial geometry with the position, between crest and valley of the wave, Figs.6 and 7. In the valley, the concave interface is very similar to that suggested by [11], Fig.7 (a). However, it presents a different geometry at the crest, with a pronounced increase of the water height in the center of the tube, as shown in Fig.7 (b). Such information should still be verified experimentally. Once confirmed, may be used to improve the closure relations used in the two-fluid model, more particularly, to improve the interfacial shear stress term.

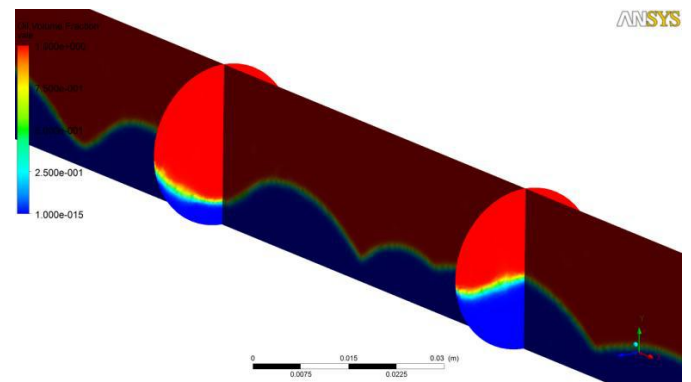


Fig.6 - Cutting planes in the simulated domain.

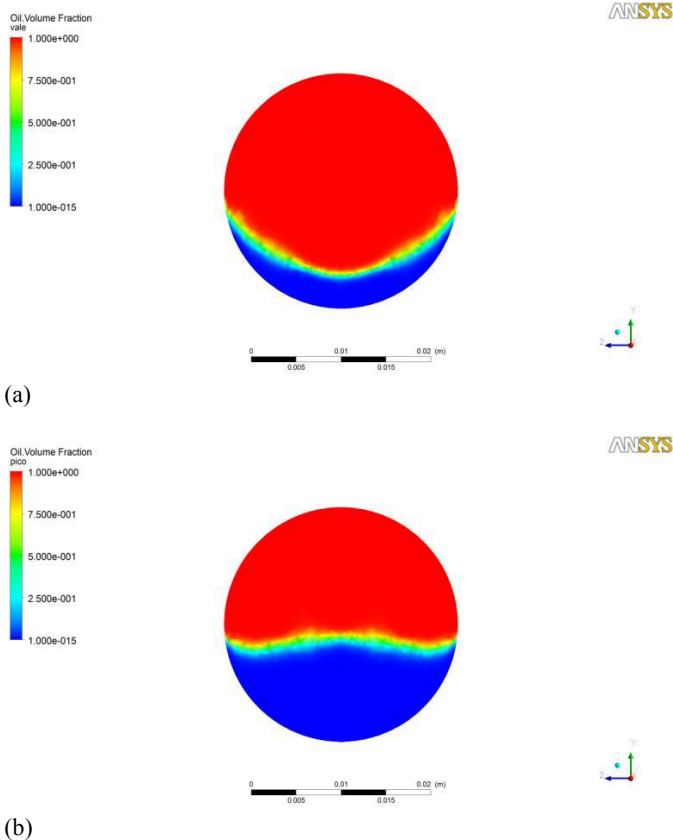


Fig. 7 - Wavy profile in the cross-sectional plane related to (a) wave valley and (b) wave crest.

CONCLUSIONS

Computational Fluid Dynamics (CFD) was used for the analysis of wavy stratified liquid-liquid pipe flow. Comparisons of simulation results with experimental and theoretical predictions were carried out. The following findings can be drawn:

1. Regarding the analysis of mesh refinement, we could verify the importance of the refinement at the interface region, to predict the shape and behavior of the interface.
2. The CFD results of *in-situ* volume fraction were compared with predictions of three phenomenological models available in the literature, showing the best agreement with experimental data.
3. The CFD results of two-phase pressure gradient were also compared with predictions of three phenomenological models available in the literature; although it performed the best, the agreement between CFD's and data were rather unsatisfactory (66%), probably due to the simulation of the frictional energy dissipation.
4. The qualitative results suggest that the interface has mainly a cross-sectional concave geometry and that it varies along the wavelength.

NOMENCLATURE

Symbol	Quantity	SI unity
θ	Angle of inclination	$^{\circ}$
ρ_o	Density of oil	g/cm^3
ρ_w	Density of water	g/cm^3
μ_o	Dynamic viscosity of the oil	N s/m^2
μ_w	Dynamic viscosity of the water	N s/m^2
δ	Liquid film thickness of the annular flow	cm

α	Amplitude of the wave interfacial	cm
f_i	Interfacial friction factor	
f_f	Friction factor of each phase, f_w or f_o , respectively, of the water and oil	
D	Diameter of the tube	cm
λ	Wavelength	cm
U_{ws}	Speed water inlet	m/s
U_{os}	Speed oil inlet	m/s
ϵ_o	Volume fraction <i>in situ</i> of the oil	m^3
ϵ_{theo}	Volume fractions <i>in situ</i> of the models	m^3
ϵ_{exp}	Volume fraction <i>in situ</i> measured experimentally	m^3
DP_{theo}	Pressure drop of the models	Pa/m
DP_{exp}	Pressure drop measured experimentally	Pa/m
ΔP	Total pressure drop by numerical simulation	Pa
$\Delta P'$	Total pressure drop handles the differential pressure transducer	Pa
ΔP_{F+A}	Portion of the pressure drop represented by the friction and acceleration	Pa
g	Gravitational acceleration	m/s^2
L	Length of the test section	m
β	Angle of inclination of the test section	$^{\circ}$

REFERENCES

- [1] A.A. Mouza, S.V. Paras, e A.J. Karabelas, CFD code application to wavy stratified gas-liquid flow, *Trans IChemE*, vol.79, pp.561-568, 2001.
- [2] C. Valleé, T. Höhne, H.M. Prasser e T. Sühnel, Experimental investigation and CFD simulation of horizontal stratified two-phase flow phenomena, *Nuclear Engineering and Design*, vol.238, pp.637-646, 2008.
- [3] F.A.A. Mendes, O.M.H. Rodriguez, V. Estevam, e D. Lopes, Flow patterns in inclined gas-liquid annular duct flow, In: A.A. Mammoli; C.A. Brebbia. (Org.), *Computational Methods in Multiphase Flow VI*, 1 ed., Southampton: WITpress, vol.6, pp.271-283, 2011.
- [4] M.S. de Castro, C.C. Pereira, J.N. dos Santos, e O.M.H. Rodriguez, Holdup, pressure drop and objective classification of inclined oil-water, *15th International Conference on Multiphase production Technology - BHR Group*, University of São Paulo, Brazil, 2011.
- [5] R. Belt et al., Comparison of commercial multiphase flow simulators with experimental and field databases, *15th International Conference on Multiphase production Technology - BHR Group*, Total EP, France-Norway, 2011
- [6] M.S. de Castro, C.C. Pereira, J.N. dos Santos, e O.M.H. Rodriguez, Geometrical and kinematic properties of interfacial waves in stratified oil-water flow in inclined pipe, *Experimental Thermal and Fluid Science*, 2011, DOI: doi:10.1016/j.expthermflusci.2011.11.003.
- [7] N.A. Vlachos, Studies of wavy stratified and stratified/atomization gas-liquid flow, *Journal of Energy Resources Technology*, vol.125, pp.131-136, 2003.
- [8] N. Petalas and K. Aziz, A mechanistic model for multiphase flow in pipes, *Journal of Canadian Petroleum Technology*, vol.39, n.6, pp.43-55, 2000.
- [9] O.M.H. Rodriguez and R.V.A. Oliemans, Experimental Study on Oil-Water Flow in Horizontal and Slightly Inclined Pipes, *International Journal of Multiphase Flow*, vol.32, pp.323-343, 2006.
- [10] S.E. Haaland, Simple and Explicit Formulas for the Friction Factor in Turbulent Pipe Flow, *Journal of Fluids Engineering*, vol.105, March, 1983.
- [11] J.L. Trallero, Oil-water flow patterns in horizontal pipes, PhD Thesis, The University of Tulsa, Tulsa, 1995.
- [12] C.C. Pereira, Estudo experimental e modelagem do escoamento estratificado ondulado óleo-água, Master Thesis, Engineering School of São Carlos, University of São Paulo, São Carlos, Brazil, 148p, 2011.
- [13] M.M. Santos, Simulação Numérica do Escoamento Bifásico Óleo-Água em Tubos, Master Thesis, UNIFEI, Itajubá, Minas Gerais, Brazil, 99 p, 2010.
- [14] G.B. Wallis, One dimensional Two Phase Flow, McGraw Hill, USA, 1969.
- [15] ANSYS CFX (2009): ANSYS CFX Reference Guide. Release 12.0. ANSYS, Inc. April 2009. Southpointe. 275 Technology Drive.
- [16] L.S. Baldani, Estudo Numérico do escoamento estratificado ondulado óleo-água, Conclusion Course Work, Engineering School of São Carlos, University of São Paulo, São Carlos, Brazil, 78p, 2011.

