6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

COMPUTATIONAL FLUID DYNAMICS APPLIED TO THE STUDY OF FALLING LIQUID FILMS IN A WAVE FILM GENERATION FACILITY

SUSANA M. IGLESIAS[1,2], DANY S. DOMINGUEZ[1,2], ALBERTO ESCRIVÁ[2], JOSÉ L. MUÑOZ-COBO[2], CÉSAR BERNA[2], JOSÉ L. CUADROS[2] AND YAGO RIVERA[2]

[1]Universidade Estadual de Santa Cruz, Postgraduate Program of Computational Modeling in Science and Technology, Rodovia Jorge Amado, km 16, Salobrinho, 45662-900, Ilhéus, BA, Brazil,
smiglesias@uesc.br, dsdominguez@gmail.com

[2]Universitat Politècnica de València, Institute for Energy Engineering, Camino de Vera 14, 46022 Valencia, Spain, aescriva@iqn.upv.es, jlcobos@iqn.upv.es, ceberes@iie.upv.es, jocuaor@upv.es, yagriver@gmail.com

Key words: CFD, Multiphase Simulation, Falling Films, Volume of Fluid.

Abstract. Two phase gas-liquid flow appears in several engineering applications, such as boiling nuclear reactors, oil and gas industry, chemical plants, among others. For this phenomenon's study the experimental facility Wave Film Generator (GEPELON, *Generador de Película Ondulatoria*) was constructed. It uses conductivity sensors for the multi-phase flow analysis in vertical pipes. In this work, we propose the use of Computational Fluid Dynamics for modelling the falling liquid films regime in this facility. Specifically, this air-water flow regime consists of a water film that falls down the pipe wall and a central region with stagnant air. We build up a simplified installation's geometrical model, considering a mass flow boundary condition in the inlet and constant pressure in the outlet. To capture the water-air interface a non-uniform structured mesh with fine elements close to the wall was constructed. Using a Volume of Fluid (VOF) model and the FLUENT solver, we simulate different flow rates and from the water volumetric fraction we estimate the film thickness in all the domain. The simulations are stationary and due to the hydrodynamic complexity, a pseudo-transient approach was used to stabilize the iterative process. The simulation values were compared with the experimental results obtained in the GEPELON facility. A mesh independence analysis was performed and several mass flows were simulated, showed good agreement with the experimental values for lower flows and reasonable for higher flows.

1 INTRODUCTION

Two phase flow regimes appear in several engineering applications such as boiling water nuclear reactors (BWR), crude-oil distillation towers, chemical reactors, cooling towers, etc. Within this gas-liquid flows, the annular flow generates interest among researchers who dedicates a lot of efforts to simulate this phenomenon in experimental facilities [1,2,3]. The droplet entrainment and the wave formation in the gas-liquid interface are a complex phenomenon that are being studied using analytical models, simulation tools and experimental techniques [4,5,6]. To perform this phenomenon's study the experimental facility Wave Film Generator (GEPELON, *Generador de Película Ondulatoria*) was constructed at Universitat Politècnica de València. The main objective of this experimental facility is to study the water-air and water-steam annular flows in vertical channels, reproducing in this way, the normal operation and the accident scenarios in boiling water nuclear reactors [7]. The GEPELON facility uses conductivity sensors to measure the wall water film thickness and to characterize the interface between the phases [8].

Recently, the computational capabilities’ increase and the improvement of models and simulation techniques have allowed the use of Computational Fluid Dynamics (CFD) in the multiphase fluid study. Among the goals associated with the development of the GEPELON experimental facility is the use of CFD techniques to simulate the studied phenomena. In this work we perform a first approach in the CFD simulation for the GEPELON facility. Specifically, we model the falling liquid films regime for the air-water flow, in a simplified geometry of this facility. Using different water mass fluxes, a mesh independence analysis was performed. After that, the mean film thickness was calculated and the simulation results were compared with experimental values.

The experimental study of falling liquid films has been addressed by several authors being the main goal the film thickness determination and the waves' amplitude and frequency calculations [9,10,11,12]. Kalliadasis et al. describe in [13] the main theoretical foundations and the several models that can be used in the falling liquid film regimes studies. Additionally, several simulation techniques have been applied to this phenomenon in different scenarios [14,15,16,17].

In the next section we describe the GEPELON facility and the experimental parameters. In section 3 we offer the simulation details, including the geometry, models, boundary conditions and other simulation parameters. The numerical results are shown in section 4 and finally, in section 5 we present the conclusions and suggestions for future works.

2 GEPELON EXPERIMENTAL FACILITY

The wave film generator experimental facility was constructed in the thermohydraulic laboratory of the Institute for Energy Engineering at the Universitat Politècnica de València, with various researcher projects funds. The facility is versatile and can be configured for several air-water regimens with dimensions and operational parameters similar to the industrial facilities. In this work we consider a falling water film regime represented in Figure 1.

In the GEPELON facility, we have a water tank at atmospheric pressure coupled to a set of flow controllers (valves, pumps, and instrumentation). The pump sends the water to a small pressurized tank located in the upper extremity of the test pipeline. In this tank the water crosses transversely a sintered porous pipeline, entering inside the test pipeline with an annular distribution, which is occupied by the stagnant air in the central section.

The test section is a methacrylate pipeline with 42E-03 m of inner radius and 3.80 m length. Along the pipeline are four (4) conductivity sensors, in 0.25, 1.10, 1.58 and 3.50 m from the inlet. The conductivity sensors collect the water film thickness over time, allowing the mean thickness determination. In this work we consider flow rates varying from 20 L/m (0.3319 kg/s) to 10 L/m (0.1660 kg/s), the temperatures were 30 ^{0}C for the water and 23 ^{0}C for the air.

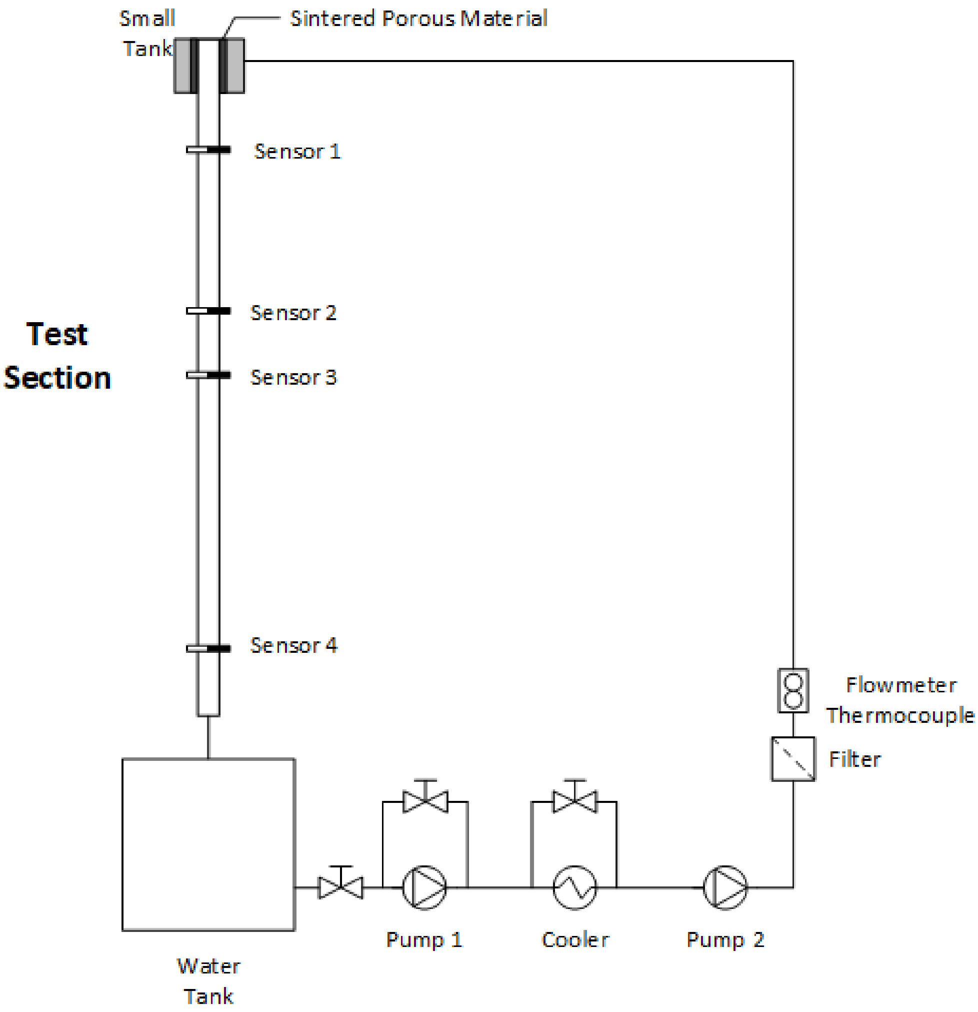

Figure 1: GEPELON experimental facility (not scaled)

3 CFD SIMULATION CONDITIONS

For the GEPELON facility simulation using CFD techniques, we considered as a domain, a two-dimensional section of the pipeline initial segment with 0.55 m length. To model the water inlet through the sintered pipeline, we used a geometric approximation introducing an annular “crown” at the domain’s beginning, with 0.25 m length and 2.0E-03 m thickness. A representation of this geometry is shown in the Figure 2a. The boundary conditions used were water mass flow rate in the inlet, mixture constant pressure in the outlet and wall conditions in the other domain boundaries.

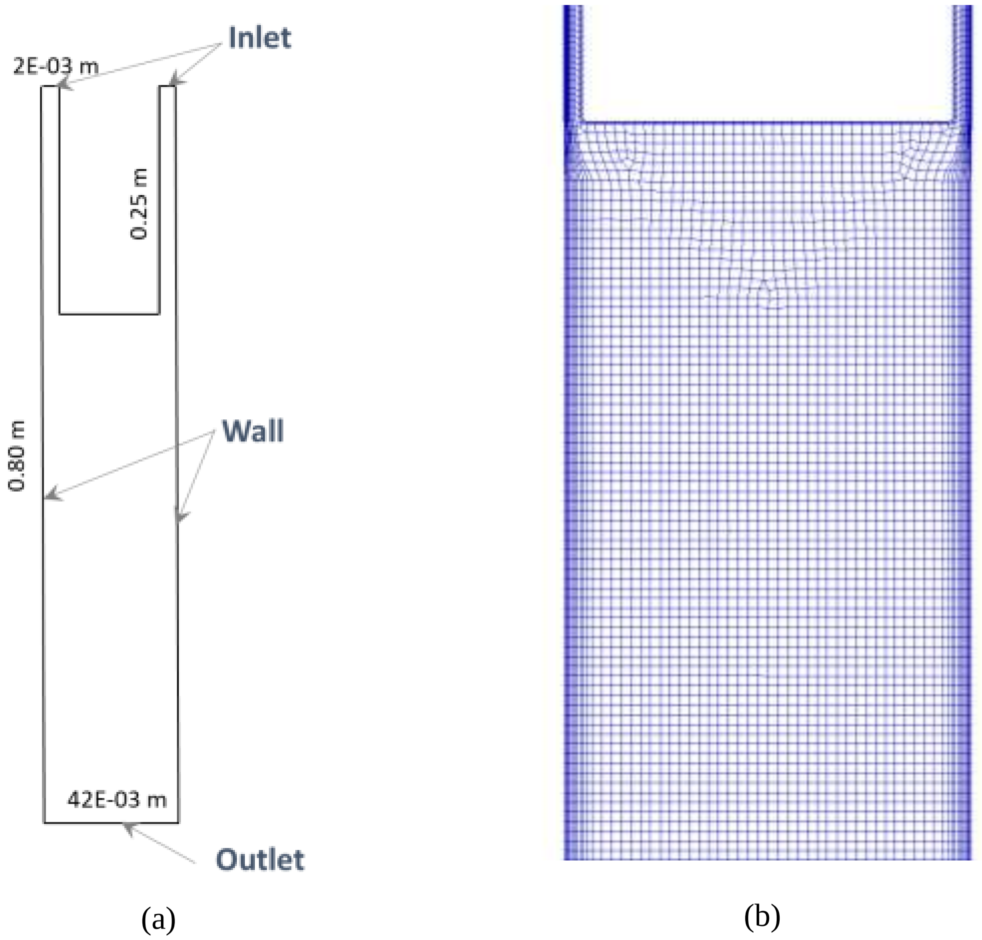

Figure 2: (a) Simulation geometry (not scaled), (b) Partial mesh representation

We built a structured mesh in all the domain, including only, some triangular elements to adjust the mesh to the geometry. In the falling water film regime, the water phase slides down the domain external wall, being the interior occupied by the air phase. As our main goal is to obtain the water film thickness, we should pay special attention to the region close to the external wall where we have the phase interaction and where appears the strongest gradients. For that reason, we use a mesh inflation with finer cells in this area, allowing a simulation with a better physical problem reproduction. A partial representation of the constructed mesh is presented in Figure 2b. To perform the mesh independence analysis were considered three (3) meshes with different characteristics that are summarized in Table 1.

Table 1: Different meshes parameters

Mesh	Elements number	Maximum face size [m]	First inflation layer [m]	Inflation layers
1	45152	1E-03	1E-04	12
2	23747	2E-03	1E-04	12
3	46824	1E-03	5E-05	15

In this work we use the version 18.1 [18] of the ANSYS Fluent. For our multiphase regime, we selected the Volume of Fluid Model (VOF) and a compressive interface scheme for the volume fraction calculations. We considered also, the Continuum Surface Force model to describe the phases interaction and the Wall Adhesion model for a detailed wall treatment. We didn’t consider the turbulence phenomenon because our simulation domain is the beginning of the test section, where we have a low-speed flow and we are far from the developed flow condition (L>32D, where L and D are longitude and diameter pipe respectively, [2]). The solver selection and the models for our simulation are based on previous works that solves similar problems [14], [19], and also in the ANSYS Fluent manual recommendations [20].

We execute stationary simulations for several water phase mass flow rates. The convergence criteria for the iterative process was: a residue less that 1E-03 in all the unknowns or stable residues with inlet-outlet mass imbalance lower than 1E-05. Due to the studied phenomenon instability we used the Fluent's pseudo-transient simulation mechanism with low relaxation coefficients. We highlight that to reach the convergence criteria for the lower mass flows rate, we had to use very low relaxation coefficient and also an elevated number of iterations.

The water film thickness is not a CFD simulation's output variable. To obtain this value we used the phases' volumetric fraction and geometric criteria resulting in this expression

$$\delta_x = \sum_i v_{f_{xi}} h_{xi}, \quad v_{f_{xi}} > 0, \tag{1}$$

where δ_x is the film thickness at x position, $v_{f_{xi}}$ is the water volume fraction at position x in i-cell, and h_{xi} is cell height.

4 RESULTS AND DISCUSSION

We perform several CFD simulations for the conditions described in section 3, with mass flow rates varying from 20 to 10 L/m. Initially we analyze the mesh independence, for that we execute simulations for mass flows values 20, 15 and 10 L/m for the meshes described in Table 1. In each simulation, we evaluate the mixture mean pressure in a radial plane and also the film thickness in the sensor 1 position (x = 0.25 m). These results are shown in Table 2.

Table 2: Results for the mesh indepence analysis

Flow rate [L/m - kg/s]	Mesh	P [Pa]	δ [mμ]	Δ_{max} P [%]	Δ_{max} δ [%]
20 - 3.32E-01	1	101498	362.851	0.00099	1.43347
	2	101497	363.004		
	3	101497	368.128		
15 - 2.49E-01	1	101493	328.750	0.00099	1.60220
	2	101492	328.846		
	3	101492	334.103		
10 - 1.66E-01	1	101487	286.102	0.00099	1.83833
	2	101486	286.103		
	3	101486	291.460		

In Table 2 is possible to notice that the maximum relative deviations among the results in the pressure's case, do not surpass the 0.1% and for the film thickness, are less than 2%. Then, we can say that the results generated by the simulation are independent of the mesh. Considering the efficiency, the results precision, and the film's profile reconstruction accuracy, we select the mesh 1 for the next simulations.

Using the mesh 1 we perform simulations in the mass flows' considered interval with an increment of 2 L/m. The water volume fraction contour for Q = 20 L/m appears in Figure 3. Is possible to observe in the figure a small area of instability where the "crown" is coupled with the test section entrance. Then, we notice the fine water film down the wall with a thickness approximately constant, while the central region of the pipeline is occupied by air. This qualitative behavior is consistent with the physical phenomenon and is repeated for all the simulated mass flows.

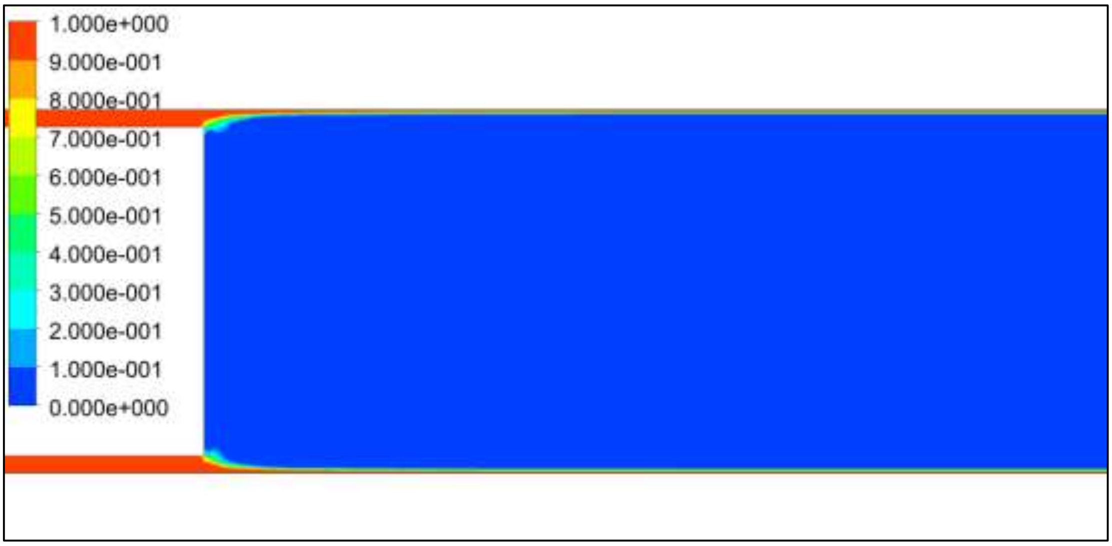

Figure 3: Water volume fraction contour for 20 L/m flow mass rate in partial domain (rotated)

In Table 3 we show the simulation results and the corresponding experimental values. The water film profiles along the test section $(0.25\ \mathrm{m} \le x \le 0.80\ \mathrm{m})$ for the analyzed flows is shown in Figure 4. Finally, in Figure 5 we compared the thickness simulation results with those measured experimentally.

Table 3: Simulation and experimental results at sensor 1 position

Flow rate Q [L/m - kg/s]	$\delta_{\mathrm{exp.}}$ [mμ]	$\delta_{\mathrm{sim.}}$ [mμ]	$\Delta\delta$ [%]
20 - 3.31900E-01	448.094	362.851	19.02356
18 - 2.98710E-01	435.794	349.950	19.69825
16 - 2.65520E-01	420.263	336.069	20.03368
14 - 2.32330E-01	340.901	321.041	5.82587
12 - 1.99140E-01	293.259	304.495	3.83156
10 - 1.65950E-01	282.183	286.102	1.38878

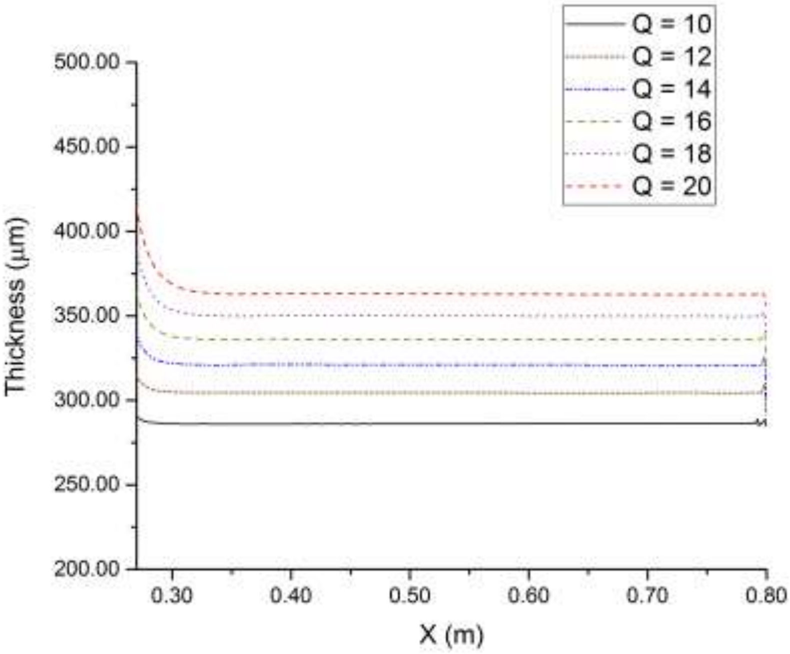

Figure 4: Film thickness profiles along the test section for different mass flow rates.

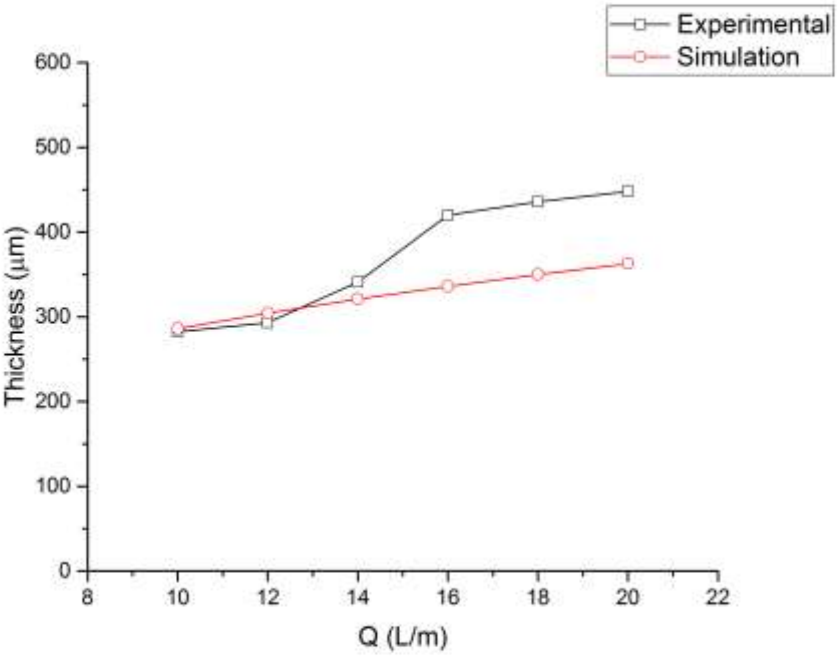

Figure 5: Comparison between simulated and experimental film thickness at sensor 1 position

Analyzing Table 3 and Figure 4 we can verify that when the water mass flow increases the film thickness also increase. We also observe an instability zone in the outlet caused by the boundary condition. When we compared the simulation results with the experimental values (see Table 3 and Figure 5) we can notice that for mass flow rates under 15 L/m we have a good concordance, with deviations lower than to 5%. In the other hand, for values greater than 15 L/m, the simulation values present a 20% deviation if compared with the experimental values. This behavior could be explained by two factors, first the inaccuracy of experimentally measured value or second, problems in the simulation models. Regarding the experimental inaccuracy, we observe a strong gradient in the film thickness when the flow changes from 14 to 16 L/m and smooth gradients for all the other flow rate changes. This behavior suggests the necessity of the experiment replication to check the results. In the other hand, the simulation models used has several parameters that must be adjusted in order to obtain the best correspondence with the experimental results.

5 CONCLUSIONS AND FUTURE WORKS

In this work, we present experimental and simulated results for falling liquid film regime in a vertical pipeline. This is the first step for the CFD simulation of GEPELON experimental facility. For mass flows between 10 and 15 L/m the experimental and simulation results show good agreement, for higher flows the results are satisfactory, however, the differences are significant (approximately 20%). We highlight that both experiments and simulations are in an initial stage and several elements of both processes have to be improved.

At this point, this research targets its action in two complementary directions: the first, that involves the simulation, comprises the domain expansion to all the test section, the performance of models' parameters sensibility study, the inclusion of turbulence models, the realization of time-dependent simulations and the improvement of the film thickness calculation methods. The second line, associated with the experimental work considers, the measures' improvement, including, error estimation techniques and the validation of the conductivity sensor's measurement, comparing their results with PLIF (Planar Laser-Induced Fluorescence) image measurements.

ACKNOWLEDGMENTS

The authors are indebted to the plan of I+D support of the EXMOTRANSIN project ENE2016-79489-C2-1-P. Author's Susana M. Iglesias and Dany S. Dominguez acknowledge the Universidade Estadual de Santa Cruz, for supporting this research.

REFERENCES

[1] A. Wolf, S. Jayanti, and G. F. Hewitt, “Flow development in vertical annular flow,” *Chem. Eng. Sci.*, (2001) **56**, no. 10, 3221–3235.

[2] I. Zadrazil, O. K. Matar, and C. N. Markides, “An experimental characterization of downwards gas-liquid annular flow by laser-induced fluorescence: Flow regimes and film statistics,” *Int. J. Multiph. Flow*, (2014) **60**:87–102.

[3] S. H. Pham, Z. Kawara, T. Yokomine, and T. Kunugi, “Measurements of liquid film and droplets of annular two-phase flow on a rod-bundle geometry with spacer,” *Int. J. Multiph. Flow*, (2015) **70**:35–57.

[4] B. J. Azzopardi, “Drops in annular two-phase flow,” *Int. J. Multiph. Flow,* (1997) **23**, no. 7, 1–53.

[5] Y. Zhao, C. N. Markides, O. K. Matar, and G. F. Hewitt, “Disturbance wave development in two-phase gas-liquid upwards vertical annular flow,” *Int. J. Multiph. Flow*, (2013) **55**:111–129.

[6] L. Pan, H. He, P. Ju, T. Hibiki, and M. Ishii, “Experimental study and modeling of disturbance wave height of vertical annular flow,” *Heat Mass Transf.*, (2015) **89**:65–175.

[7] J. L. Cuadros, C. Berna, S. Chiva, A. Escrivá, and G. Monrós, “Estudio de la interfase en el flujo anular aire agua co-corriente en tubería vertical ascendente .,” in *43 Reunión Anual de la Sociedad Nuclear Española*, (2017).

[8] J. L. Muñoz-Cobo, S. Chiva, S. Méndez, G. Monrós, A. Escrivá, and J. L. Cuadros, “Development of conductivity sensors for multi-phase flow local measurements at the polytechnic university of valencia (UPV) and university jaume I of castellon (UJI),” *Sensors (Switzerland)*, (2017) **17**, no. 5.

[9] P. L. Kapitza, “Wave flow of thin layers of a viscous fluid: I. Free flow. II. Fluid flow in the presence of continuous gas flow and heat transfer,” in *Collected Papers of P.L. Kapitza*, pp. 662–689, (1965).

[10] P. L. Kapitza and S. P. Kapitza, “Wave flow of thin layers of a viscous fluid: III. Experimental study of undulatory flow conditions,” in *Collected Papers of P.L. Kapitza*, pp. 690–709, (1965).

[11] H. Takahama and S. Kato, “Longitudinal flow characteristics of vertically falling liquid films without concurrent gas flow,” *Int. J. Multiph. Flow*, (1980) **6**, no. 3, 203–215.

[12] T. D. Karapantsios and A. J. Karavelas, “Longitudinal Charactheristics Of Wavy Falling Films,” *Int. J. Multiph. Flow*, (1995) **21**, no. 1, 119–127.

[13] S. Kalliadasis, C. Ruyer-Quil, B. Scheid, and M. G. Velarde, *Falling Liquid Films*, no. 163. Springer, Applied Mathematical Sciences, (2012).

[14] D. Gao, N. B. Morley, and V. Dhir, “Numerical simulation of wavy falling film flow using VOF method,” *J. Comput. Phys.*, (2003) **192**, no. 2, 624–642.

[15] C. D. Ho, H. Chang, H. J. Chen, C. L. Chang, H. H. Li, and Y. Y. Chang, “CFD simulation of the two-phase flow for a falling film microreactor,” *Int. J. Heat Mass Transf.*, (2011) **54**, no. 15–16, pp. 3740–3748.

[16] G. D. Wehinger, J. Peeters, S. Muzaferija, T. Eppinger, and M. Kraume, “Numerical simulation of vertical liquid-film wave dynamics,” *Chem. Eng. Sci.*, (2013) **104**:934–944.

[17] M. Martin, T. Defraeye, D. Derome, and J. Carmeliet, “A film flow model for analysing gravity-driven, thin wavy fluid films,” *Int. J. Multiph. Flow*, (2015) **73**:207–216.

[18] ANSYS, *ANSYS Fluent Theory Guide*, April, (2017).

[19] N. Mascarenhas and I. Mudawar, “Study of the influence of interfacial waves on heat transfer in turbulent falling films,” *Int. J. Heat Mass Transf.*, (2013) **67**:1106–1121.

[20] ANSYS, *ANSYS FLUENT User’s Guide*, April (2017).