encit 2020

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

18th Brazilian Congress of Thermal Sciences and Engineering
November 16–20, 2020 (Online)

ENC-2020-0458
COMPARISON OF NUMERICAL SIMULATIONS WITH EXPERIMENTAL DATA FOR VERTICAL ANNULAR AND CHURN GAS-LIQUID FLOWS

Larissa Steiger de Freitas
Marcus Vinícius Canhoto Alves
Mechanical Engineering Department, State University of Santa Catarina, Paulo Malshitzki, 200, Joinville - SC - Brazil
larissasteigerf@gmail.com; marcus.alves@udesc.br

Rafael Rodrigues Francisco
Petroleum Engineering Department, State University of Santa Catarina, Lourival Cesário, Balneário Camboriú - SC - Brazil
rafael.francisco@udesc.br

Abstract. *This work presents transient and three-dimensional investigations of the annular and churn liquid-gas flow patterns. The continuity and Navier-Stokes equations for each phase are solved together with an advection equation for gas volume fraction. A hybrid algorithm that combines the characteristics of the Eulerian-Eulerian and VOF models, called multiphaseEulerFoam, available within the OpenFOAM source code, is used for the simulations. The simulations are based in the experiments by Govan et al. (1991) that, are used as a basis to introduce the initial and boundary conditions. The results for the pressure gradient and the void fraction in the annular and churn flows are presented, along with its isosurfaces. The simulation results show a reasonable agreement with the literature.*

Keywords: *Annular Flow, Churn Flow, OpenFOAM, multiphaseEulerFoam*

1. INTRODUCTION

Multiphase flows have several applications in areas of engineering interest. They appear in distillers in the chemical industry, in cooling equipment in the thermal and nuclear power generation industry, and in the oil industry. Transient analysis of two-phase flows is particularly important in the nuclear industry. The geometry assumed by the fluids, and their behavior are directly related to the Pressurized Water Reactors – PWR operational safety. In cases where there is a failure in the cooling of the reactors, (Loss-of-Coolant Accidents - LOCAs), the liquid film adjacent to the tube wall can be depleted, generating dryout flows. This type of flow is capable of rapidly increasing the temperature of the pipe wall, if the heat flow is kept constant, which causes large-scale accidents.

The phenomenon mentioned above is associated with the flooding and flow reversal mechanisms, described by Hewitt and Hall-Taylor (1970). This work aims to investigate the annular and churn liquid-gas flow patterns in vertical pipes, through a transient and three-dimensional simulation. The annular flow is characterized by the presence of a stable liquid film with a large gas core at high speed, which has liquid droplets pulled from the film by shear forces. Its parameters have been extensively investigated since the 1960s (see, *e.g* Hewitt (1961), Hutchinson *et al.* (1973), Tatterson (1975), Whalley (1974), Whalley and Hewitt (1978), Hewitt and Govan (1990), Okawa and Kataoka (2005)). On the other hand, the churn flow is highly unstable and disturbed and presents large interfacial waves where the liquid film adjacent to the walls can show changes in direction. In the churn flow, the elongated bubbles, known as Taylor bubbles, are not present since they collapse due to the high gas fraction in the two-phase mixture. Compared to the annular flow, the churn pattern has less studies and is less understood, since its unstable nature has often made it considered to be only a transitional flow pattern.

There are different ways to identify the churn-annular transition. According to Barbosa (2001), flow reversal is the most used parameter to characterize the transition between churn and annular patterns, because it is an easily visible phenomenon (see, *e.g.* the flow reversal criterion proposed by Wallis (1969), based on the correlation of Hewitt and Wallis (1963) for the flooding phenomenon). Another way to identify the churn-annular transition is by analyzing the pressure gradient. According to Owen (1986), in the churn flow pattern, the flooding waves are intensely present, and as the gas speed increases, they tend to disappear. The disappearance of these waves causes a drop in the pressure gradient, which passes through a point of minimum in the transition region and rises gradually when approaching the annular flow. The average interfacial shear stress on the wall can also be used as a transition criterion, since it assumes positive values when the film flow is entirely upward, and negative values when part of the film flow is downward. In this way, the churn-annular transition occurs when the average stress in the wall is equal to zero, which is positive in the annular flow, and negative in the churn flow.

The topology of the churn and annular flow patterns, present characteristics of segregated and dispersed flows at the same time. For this reason, it is even more difficult to simulate these flow patterns, since the codes indicated to simulate segregated flows are different from those indicated for dispersed flows. For flows with segregated characteristics, the ideal is to use the VOF (Volume of Fluid) method, while in dispersed flows, the use of the two-fluid method is more appropriate. OpenFOAM (Open-source Field OperationAnd Manipulation) is a free commercial software, developed by OpenCFD Ltd in 2004. It has in its pre-programmed libraries a hybrid algorithm, called multiphaseEulerFoam, capable of coupling the characteristics of the VOF method with the two-fluid model.

Despite the range of works available in the literature regarding two-phase vertical liquid-gas flows, most present simplified models, and empirical correlations for calculating pressure drop in only one dimension, which does not describe the behavior of fluids in depth. In addition, in cases where simulations are conducted more robustly, the most used commercial software is ANSYS-FLUENT, which besides having a high license cost, does not allow access to the source code (see *e.g* Da Riva and Del Col (2009), Liu *et al.* (2011), Karami *et al.* (2014), Qiu *et al.* (2014), Parsi *et al.* (2016)). Therefore, the present proposal aims to investigate the two-phase liquid-gas flow in the vertical configuration, during the transient churn-annular transition process. For this, the open access commercial program OpenFOAM (source code available in C ++) will be used for the transient three-dimensional flow simulation.

2. NUMERICAL METHODOLOGY

The topology of the annular and churn flow patterns has characteristics of segregated and dispersed flows at the same time. Thus, the best way to perform the simulations proposed here is to use the hybrid code developed by Wardle and Weller (2013), which contemplates the characteristics of the Eulerian-Eulerian and VOF models. This algorithm is called multiphaseEulerFoam and is available within the OpenFOAM libraries.

OpenFOAM is an open source commercial software developed by OpenCFD Ltd in 2004. It is a simulator widely used in industries and academia, being able to solve the mechanics of the continuum equations, including the Computational Fluid Dynamics (CFD). Its great differential in relation to many commercial programs available, is that its source code is open, which allows changes to the user. Among the OpenFOAM pre-program libraries, the software multiphaseEulerFoam couples the VOF method with the two-fluid model. VOF is used to represent the large interfacial scales, while the two-fluid method is responsible for including information about the dispersion phase in the model, without a large computational effort.

2.1 Mathematical equations

To derive the system government equations, it is necessary to identify the phases present in the computational cells studied. According to Hill (1998), it is possible to condition local equations through the indicator function $I_k(x,t)$, described by Equation 1. If the phase of interest is inside the observed cell, its average conservation equations are computed, multiplying them by 1. On the other hand, if the phase of interest is not present in the cell, its effects are disregarded, and its equations are multiplied by 0. Still, according to Cerne *et al.* (2001), if there is more than one phase present in the cell analyzed, an average process based on the volume fraction of each phase is used to compute the effects of the two-phase mixture.

$$I_k(x,t) = \begin{cases} 1, if\ (x,t)\ is\ in\ phase\ k \\ 0, if\ (x,t)\ isn't\ in\ phase\ k \end{cases} \tag{1}$$

According to Tocci (2016), the volume fraction of the k phase is defined by the Equation 2

$$\alpha_k = \overline{I_k\,(x,t)} \tag{2}$$

where the dash represents the averaging process.

The mathematical modeling of the problem studied is composed of the continuity equation (Equation 3), and the Navier-Stokes equation (Equation 4). They are presented below, considering the hypothesis of a flow in three dimensions, incompressible, with immiscible fluids and with the properties of each fluid constant.

$$\frac{\partial \alpha_k}{\partial t} + \vec{U}_k \cdot \nabla \alpha_k = 0 \tag{3}$$

$$\frac{\partial \left(\rho_k \alpha_k \vec{U}_k\right)}{\partial t} + \left(\rho_k \alpha_k \vec{U}_k \cdot \nabla\right) \vec{U}_k = \nabla \cdot \left(U_k \alpha_k \nabla \vec{U}_k\right) - \alpha_k \nabla P + \rho_k \alpha_k \vec{g} + \vec{F}_{D,k} + \vec{F}_{S,k} \tag{4}$$

where ρ_k is the density of the k phase, α_k indicates the volume fraction of the k phase, $\vec{U}_k$ represents the speed of the k phase, and $\vec{g}$ is the acceleration due to gravity. Drag forces and surface tension are represented by $\vec{F}_{D,k}$ and $\vec{F}_{S,k}$, respectively.

The surface tension can be found through the Equation 5, proposed by Brackbill *et al.* (1992):

$$\vec{F}_{S,k} = \sigma \kappa \nabla \alpha_k \tag{5}$$

where σ is the coefficient of surface tension between the phases, and κ is the local surface curvature. The Schiller and Naumann model, available in the OpenFOAM library, can be used to find the overall drag coefficient by averaging the coefficients for each phase, and after obtained, it is possible to find the drag force term (TOCCI, 2016).

In addition, the mathematical description has the equation 6, which is purely advective, and has the function of including interface information in the model:

$$\frac{\partial \alpha_k}{\partial t} + \vec{U}_k \cdot \nabla \alpha_k + \nabla \cdot \left(\vec{U}_c \alpha_k (1 - \alpha_k) \right) = 0 \tag{6}$$

where $\vec{U}_c$ is the artificial interface compression speed, given by Equation 7

$$\vec{U}_c = C_\alpha \left| \vec{U} \right| \frac{\nabla \alpha_k}{|\nabla \alpha_k|} \tag{7}$$

where C_α is the interface compression ratio, and the term $\frac{\nabla \alpha_k}{|\nabla \alpha_k|}$ indicates the interface normal vector. To include the effects of turbulence, OpenFOAM has a library with several turbulence models. The model used here, is the $k - \omega\, SST$, which is a combination of the $k - \varepsilon$ and $k - \omega$ models, responsible for calculating the turbulent kinetic energy and the turbulent dissipation rate, in situations far away and close to the tube wall, respectively.

According to Wardle and Weller (2013), to ensure the convergence and stability of the simulations performed with the multiphaseEulerFoam code, the Courant number described by Equation 8, must be less or equal to 0.25.

$$C_o = \Delta t \sum_{i=1}^{n} \frac{\vec{U}_i}{\Delta x_i} \tag{8}$$

where Δt represents the time step, and Δx is the space between the mesh cells.

To numerically solve Equations 3, 4, and 6, it is necessary to discretize the transient, gradients, and Laplacian terms. To do this, the implicit Euler method, the least squares method, and the Gauss with limited linear interpolation scheme has been used, respectively. As in Tocci (2016), the divergent terms were separated in turbulent, phase fraction, convection and other terms, which were discretized using the limited linear Gauss, VanLeer, and limited linear GaussV methods. All these methods have limiting functions to avoid non-physical values in the simulations. In solving incompressible and transient flows, it is recommended to use the PISO algorithm to solve the pressure-velocity coupling. In this type of code, for each iteration, the momentum conservation equation and the correction equation for the pressure are solved twice.

2.2 Problem Geometry

The simulations were conducted in a 1.682 [m] tube with 0.0318 [m] of internal diameter. The liquid and gaseous phases are introduced at the bottom of the pipe and leave the duct through the upper outlet. The liquid is introduced to induce an annular pattern, with a film thickness of 0.0015 [m]. Figure 1 illustrates the geometry used.

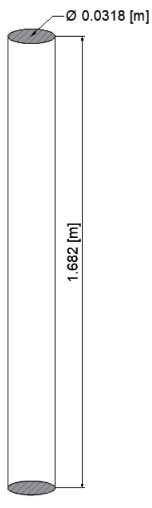


Figure 1: Geometry Used in Simulations.

2.3 Fluids Properties

Table 1 presents the properties of air and water, obtained at a temperature of approximately $T = 293.15\,[K]$, at a pressure of $P = 133000\,[Pa]$.

Table 1: Fluids Properties.

	Air	**Water**
$\rho\,[kg/m^3]$	1.58	997
$\mu\,[Pa.s]$	$1.84\,x\,10^{-5}$	0.001

2.4 Boundary Conditions

To analyze the annular flow and the churn flow, two points of the experiment conducted by Govan *et al.* (1991) were selected. Each point was taken from one of the aforementioned regions in order to establish the flow conditions for entering the simulation. In addition, some parameters required by the $k-\omega\,SST$ turbulence model, such as turbulent kinetic energy (k), and the turbulent dissipation rate (ω), were calculated. These boundary conditions are presented in Table 2.

Table 2: Boundary Conditions.

	ANNULAR		**CHURN**	
	Air	**Water**	**Air**	**Water**
$\dot{m}[kg/s]$	$1.73\,x\,10^{-2}$	$2.53\,x\,10^{-2}$	$1.56\,x\,10^{-2}$	$2.43\,x\,10^{-2}$
$k[m^2/s^2]$	$5.26\,x\,10^{-1}$	$6.93\,x\,10^{-6}$	$4.35\,x\,10^{-1}$	$6.48\,x\,10^{-6}$
ω	$5.95\,x\,10^{+2}$	2.16	$5.41\,x\,10^{+2}$	2.09

3. RESULTS

The simulations were performed on the computers available at the Center for Technological Sciences at the University of the State of Santa Catarina. Two-phase flows require, in general, a great computational effort. For this reason, and considering the limitations of the processors employed, it was possible to create three different meshes: one with 15264 volumes, another with 48760, and finally, one with 186560 volumes. For the post-processing step of the data found, an OpenFOAM tool called ParaView was used. ParaView allows the visualization of flow geometry, in addition to having a series of data processing features. The data were exported to the Origin graphic software, which enabled the generation of the images that will be presented in this work.

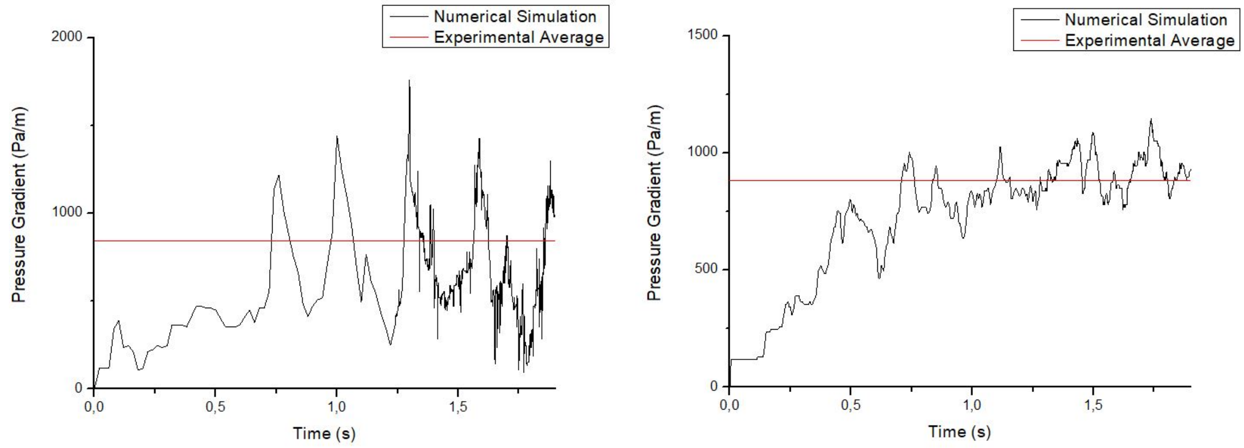
Table 3 shows the influence of the mesh volumes on the results of absolute pressure and void fraction, for a plane located at 0.858 [m] from the pipe inlet. This point is the same used by Govan *et al.* (1991) for bench calibration. In the experiments conducted by the authors, the pressure was maintained at 133000 [Pa] through valve 2 (see Govan *et al.* (1991)).

Table 3: Mesh Analysis.

	ANNULAR		**CHURN**	
Mesh	**Pressure [Pa]**	**Void Fraction [-]**	**Pressure [Pa]**	**Void Fraction [-]**
15264	134765.124	0.926	134254.577	0.924
48760	132971.094	0.989	132910.388	0.984
186560	132603.510	0.985	132860.071	0.989

For the finer mesh, pressure gradients were obtained in the annular and churn flows, which are represented by Figures 2.a and 2.b, respectively. Analyzing the results found numerically, it is possible to observe a large oscillation of the pressure gradient value in both flow patterns, which can be attributed, in part, to the liquid filaments that pass through the pipeline causing a gas phase strangulation, and consequently they generate an oscillation in the pressure. In addition, another factor that contributes to a greater pressure fluctuation at this point is that the liquid filaments have a lower velocity than the gas, which causes a greater consumption of flow energy when they are accelerated towards the end of the tube. For the annular flow, the relative error between the simulated pressure gradient and the experimental one was 16.3%, while for the churn flow, the relative error was 3.46%. Probably, the simulation of the annular flow is suffering numerical diffusion, since in practice, the pressure oscillations should be greater in the churn flow. In addition, it is important to note that the flow has not yet reached the steady state, so the oscillations caused by the presence of liquid filaments in the gas core are still significant.

Figure 2: Pressure gradient at 0.858 [m] from the entrance.

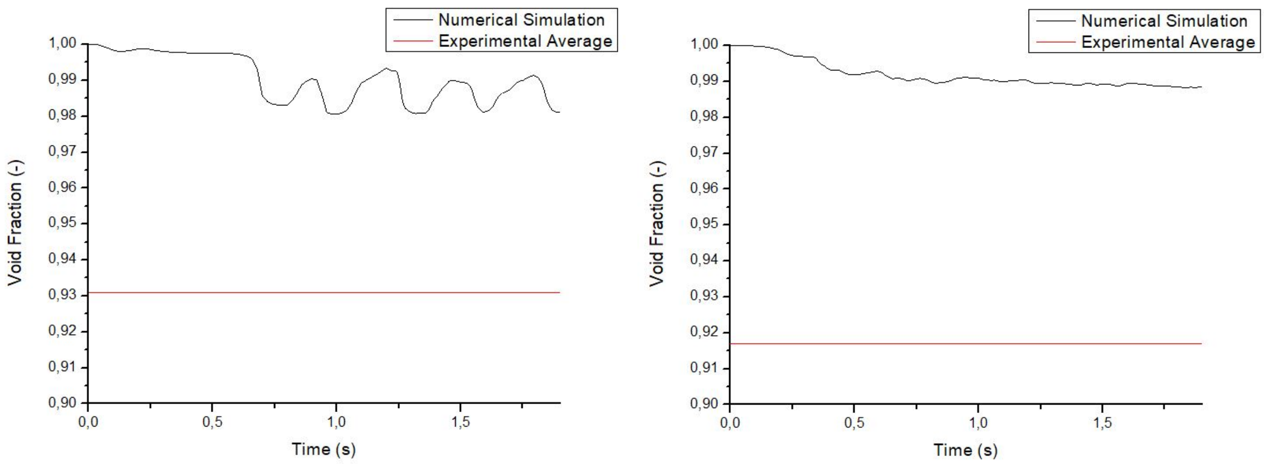


(a) Pressure gradient for annular flow.

(b) Pressure gradient for churn flow.

Another important analysis parameter is the void fraction. Figure 3 illustrates the temporal variation of the average volumetric fraction of the gas in the annular and churn flow patterns, measured between the 0.296 [m] and 1.195 [m] planes. In the annular flow (Figure 3.a), oscillations in the void fraction are more accentuated, and the moments when the void fraction suffers a more sudden drop indicate the passage of small filaments or drops of liquid in the gas, which increases the volumetric fraction of the liquid phase and decreases the fraction of the gas phase, since the sum between the void fractions of gas and liquid retention must be about 1. On the other hand, in the churn flow (Figure 3.a), the fraction remains with few oscillations and a value very close to 1, which shows the majority presence of the gas phase. The relative error between the simulated void fraction and the experimental one was 5.90% for the annular flow, and 7.86% for the churn flow, which indicates a good agreement with the reference values.

Figure 3: Void fraction between the 0.296 [m] and 1.195 [m] planes.



(a) Void fraction for annular flow.

(b) Void fraction for churn flow.

For a better visualization and understanding of the simulations and flow, Figures 4 and 5 were obtained, which presents the isosurfaces for the annular and churn flows at two different times, being t = 1.290 [s] and t = 1.520 [s] respectively. In each of the figures, cross sections of the pipe are shown at two different points: the first cut is made at a position 0.296 [m] away from the pipe inlet, and the second cut is located at 1.195 [m] from the pipe inlet. In addition, the influence of the mesh refinement can also be seen through the figures, since the columns on the left represent a mesh of 48760 volumes, and the columns on the right have the finest mesh, with 186560 volumes. Although the flow has not yet reached a steady state, the flow behaves similarly in the two time periods presented.

Through the isosurfaces, it is possible to verify that the hybrid numerical method used to develop the simulation, is able to capture the effects of the liquid film, the filaments, and the droplets, and that, the finer mesh captures the small drops in more detail. The computationally obtained figures are very similar with the description of the annular flow and churn patterns present in the literature, in the case of annular flow, the large gas core and the film adjacent to the walls are observed with some filaments that tend to detach from the film. In the case of churn flow, it is also possible to identify the gas core, and a liquid film with a much more chaotic aspect, together with some liquid droplets.

Figure 4: Isosurfaces for annular and churn flows at t = 1.290 s.

(a) Annular flow at 0.296 m in a 48760 mesh.

(b) Annular flow at 0.296 m in a 186560 mesh.

(c) Annular flow at 1.195 m in a 48760 mesh.

(d) Annular flow at 1.195 m in a 186560 mesh.

(e) Churn flow at 0.296 m in a 48760 mesh.

(f) Churn flow at 0.296 m in a 186560 mesh.

(g) Churn flow at 1.195 m in a 48760 mesh.

(h) Churn flow at 1.195 m in a 186560 mesh.

Figure 5: Isosurfaces for annular and churn flows at t = 1.520 s.

(a) Annular flow at 0.296 m in a 48760 mesh.
(b) Annular flow at 0.296 m in a 186560 mesh.
(c) Annular flow at 1.195 m in a 48760 mesh.
(d) Annular flow at 1.195 m in a 186560 mesh.
(e) Churn flow at 0.296 m in a 48760 mesh.
(f) Churn flow at 0.296 m in a 186560 mesh.
(g) Churn flow at 1.195 m in a 48760 mesh.
(h) Churn flow at 1.195 m in a 186560 mesh.

4. FINAL COMMENTS

This work presented transient and three-dimensional investigations of the annular and churn liquid-gas flow patterns through a hybrid model that combines the characteristics of the Eulerian-Eulerian and VOF models present in OpenFoam libraries, called multiphaseEulerFoam. The simulations were based on the experiments by Govan *et al.* (1991), which served as a basis for the boundary conditions of the simulation.

The results found for the pressure gradient showed a relative error between the simulated and the experimental value of 16.3% for the annular flow, and 3.46% for the churn flow, which indicates a possible numerical diffusion for the case of the annular flow. Another factor analyzed was the volumetric fraction of the gas, which showed a relative error between the simulated and the experimental value of 5.90% for the annular flow, and 7.86% for the churn flow, which indicates a good agreement with the reference values. Finally, the isosurfaces of the annular and churn flows were presented, which made it possible to visualize the flow, and confirmed the efficiency of the method of capturing the characteristic phenomena of these types of flows, such as liquid film, filaments and droplets. In general, the results found are reasonably in line with what is available in the literature.

5. ACKNOWLEDGEMENTS

The authors would like to thank FAPESC for the financial support to the project through the FAPESC 2019TR000783 and FAPESC 2019TR000779 concessions. To CNPq for the financial support granted through grant 433820 / 2018-7. Larissa Steiger de Freitas would like to individully thank the Coordination for the Improvement of Higher Education Personal (CAPES) for the Master's scholarship during the execution of this work (Code 001).

6. REFERENCES

Barbosa, J.R., 2001. *Phase change of single component fluids and mixtures in annular flow*. Ph.D. thesis, University of London-Imperial College London.

Brackbill, J.U., Kothe, D.B. and Zemach, C., 1992. "A continuum method for modeling surface tension". *Journal of computational physics*, Vol. 100, No. 2, pp. 335–354.

Cerne, G., Petelin, S. and Tiselj, I., 2001. "Coupling of the interface tracking and the two-fluid models for the simulation of incompressible two-phase flow". *Journal of computational physics*, Vol. 171, No. 2, pp. 776–804.

Da Riva, E. and Del Col, D., 2009. "Numerical simulation of churn flow in a vertical pipe". *Chemical Engineering Science*, Vol. 64, No. 17, pp. 3753–3765.

Govan, A., Hewitt, G., Richter, H. and Scott, A., 1991. "Flooding and churn flow in vertical pipes". *International journal of multiphase flow*, Vol. 17, No. 1, pp. 27–44.

Hewitt, G.F., 1961. "Analysis of annular two-phase flow: Application of the dukler analysis to vertical upward flow in a tube". Technical report, United Kingdom Atomic Energy Authority. Research Group. Atomic Energy ....

Hewitt, G.F. and Wallis, G.B., 1963. "Flooding and associated phenomena in falling film flow in a tube". Technical report, United Kingdom Atomic Energy Authority. Research Group. Atomic Energy ....

Hewitt, G. and Govan, A., 1990. "Phenomenological modelling of non-equilibrium flows with phase change". *International journal of heat and mass transfer*, Vol. 33, No. 2, pp. 229–242.

Hewitt, G. and Hall-Taylor, N., 1970. "Annular two-phase flow". *Pergamon Press, Oxford, UK*.

Hill, D.P., 1998. *The computer simulation of dispersed two-phase flow*. Ph.D. thesis, Citeseer.

Hutchinson, P., PB, W. *et al.*, 1973. "A possible characterization of entrainment in annular flow".

Karami, H., Torres, C.F., Parsi, M., Pereyra, E. and Sarica, C., 2014. "Cfd simulations of low liquid loading multiphase flow in horizontal pipelines". In *Fluids Engineering Division Summer Meeting*. American Society of Mechanical Engineers, Vol. 46261, p. V002T06A011.

Liu, Y., Li, W. and Quan, S., 2011. "A self-standing two-fluid cfd model for vertical upward two-phase annular flow". *Nuclear Engineering and Design*, Vol. 241, No. 5, pp. 1636–1642.

Okawa, T. and Kataoka, I., 2005. "Correlations for the mass transfer rate of droplets in vertical upward annular flow". *International journal of heat and mass transfer*, Vol. 48, No. 23-24, pp. 4766–4778.

Owen, D.G., 1986. *An experimental and theoretical analysis of equilibrium annular flows*. Ph.D. thesis, University of Birmingham.

Parsi, M., Agrawal, M., Srinivasan, V., Vieira, R.E., Torres, C.F., McLaury, B.S., Shirazi, S.A., Schleicher, E. and Hampel, U., 2016. "Assessment of a hybrid cfd model for simulation of complex vertical upward gas–liquid churn flow". *Chemical Engineering Research and Design*, Vol. 105, pp. 71–84.

Qiu, G., Wu, Z., Jiang, Y., Li, S. and Cai, W., 2014. "Numerical simulation of condensation of upward flow in a vertical pipe". In *Fluids Engineering Division Summer Meeting*. American Society of Mechanical Engineers, Vol. 46247, p. V01DT32A003.

Tatterson, D.F., 1975. "Rates of automization and drop size in annular two phase flow". *PhDT*.

Tocci, F., 2016. "Assessment of a hybrid vof two-fluid cfd solver for simulation of gas-liquid flows in vertical pipelines in openfoam".

Wallis, G.B., 1969. "One-dimensional two-phase flow".

Wardle, K.E. and Weller, H.G., 2013. "Hybrid multiphase cfd solver for coupled dispersed/segregated flows in liquid-liquid extraction". *International Journal of Chemical Engineering*, Vol. 2013.

Whalley, P., 1974. "Experimental wave and entrainment measurements in vertical annular two-phase flow". In *Multi-phase*

Flow Systems Symp., Strathcylde., 1974.

Whalley, P. and Hewitt, G.F., 1978. *The correlation of liquid entrainment fraction and entrainment rate in annular two-phase flow*. UKAEA Atomic Energy Research Establishment.

7. RESPONSIBILITY NOTICE

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