NUMERICAL SIMULATION OF TWO-PHASE FLOW IN HORIZONTAL CYLINDRICAL TUBES USING CFX

Rejane De C. Oliveski,
decesaro@unisinos.br
Maria Luiza S. Indrusiak,
sperbindrusiak@via-rs.net
Marcelo Dalla Corte,
marcelodallacorte@gmail.com
Universidade do Vale do Rio dos Sinos - UNISINOS
Av. Unisinos, 950, CEP: 93020-000, São Leopoldo, RS - Brazil

Abstract. *This work presents the slug formation and evolution study of an air-water two-phase flow in horizontal pipes by means of a numerical simulation model developed using the CFX software. The work of Frank, 2005, was reproduced and variations on some parameters were introduced in order to verify their influence on the results. The parameters analyzed were the imposed inlet fluctuation of the interface and the damping of turbulent diffusion at the interface between the two phases. Also a slip velocity was introduced in order to give more realistic characteristics to the model.*

Keywords: *Two phase flow, slug flow, numerical simulation, CFX.*

1. INTRODUCTION

The slug pattern is frequently found in multi-phase flows in horizontal tubes, especially in hydrocarbon transport, liquid-vapor flow in thermo-electric plants and refrigeration systems. That kind of flow may be greatly dangerous for the tube structure and even for all the system, due to the great oscillations of pressure caused by the obstruction of the tube when the slug occurs. (Frank, 2005).

Experimental analysis spend time, resources and are often of difficult control, Meanwhile numerical analysis allow to simulate different processes, operational conditions, fluids and geometry in reduced time and saving resources.

The determination of flow pattern map and correlations for heat transfer and pressure losses are being studied from an experimental approach for a long time as, e.g. Tong (1965). Nevertheless there is still more to be developed in the field of the numerical simulation since the use of numerical models for those processes is very recent, along with the availability of compatible computational resources. Usually two-phase flow numerical studies are limited to only one kind of flow, since the processes of heat transfer and pressure loss are different for each pattern, from the bubbles formation to the film vaporization (Carey, 2008). Because of that, studies are necessary particularly for obtaining experimental data and numerical modeling of those processes.

Although the final goal of the authors is to perform the numerical simulation of the entire process of phase change flow in tubes many steps still must be overcome. In the present work the formation of slugs in a stratified flow air-water was studied. The numerical results of Frank (2005) were reproduced and additional studies on the influence of the variation of some parameters like the amplitude of the interface fluctuation imposed at the inlet and the introduction of a slip velocity between phases were conducted. Also the influence of the source term introduced in Frank's model with the specific function of turbulent diffusion damping in the air-water interface was studied.

2. THE SYSTEM IN ANALYSIS

The system in analysis consists in a 54 mm diameter and 8 m long horizontal tube, with 50% of the volume occupied by water flow and the remainder 50% by air flow. It was imposed a sinusoidal function to the water level at the inlet of the tube to simulate the fluctuation commonly occurring in real systems. The system is isothermal and the fluids are at 25°C.

3. METHODOLOGY

Commercial software CFX11 was used to simulate the two-phase air-water flow. This software uses mass, energy and momentum conservation equations to represent multiphase flow. From this basic set of equations various models can be chosen depending upon the process conditions and the simulation objectives.

The first great difference is related to the number of conservation equations utilized in the process. In this case two models are allowed: the Homogeneous and the Non-homogeneous. In the Homogeneous Model only one set of conservation equations is used and the phases are characterized by the mass quantity. In the Non-homogeneous model a set of conservation equations (energy and momentum) is used, enhanced by closure models for the problems both thermal and dynamical. In the case of turbulent flow the turbulence model can be duplicated too - one for the vapor phase and other for the liquid phase - according to the user's choice and depends upon flow characteristics.

There are great differences between simulations with Homogeneous and Non-homogeneous models. The last one is much more expensive in terms of processing time (CPU) due to number of equations be duplicated. By the other hand, it is much more flexible since it permits to utilize different closure models for energy and momentum equations. Some of those models are available in the basic version of the software but others must be incorporated via external programming. Besides, thanks to this degree of detailing, the Non-homogeneous model permits a better sensibility of the physical problem. Conversely that number of closure conditions demands the user a much greater effort to set the model.

A tri-dimensional model used comprises the energy and momentum conservation equations in three directions, mass conservation equation and turbulence model $\kappa-\omega$. A transient model was used with time step of 0.005s and a convergence criterion of 10^{-5}. Tests investigating the influence of the number of elements in the hexahedral mesh were performed; refinements above those used in this work did not result in significant variations. The mesh was built with 215000 elements, 430 at the cross section and 500 along the tube. Also a refinement was applied at the tube wall vicinity. Fig. 1 shows the volume distribution in the computational domain.

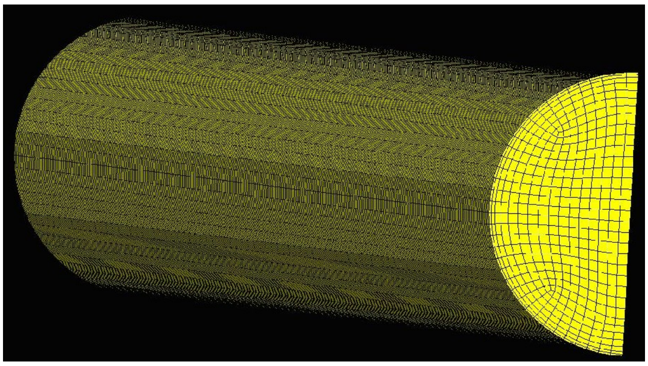


Figure 1. Volume distribution in the tube - hexahedral mesh.

For both fluids - water and air - the continuum hypothesis was assumed. In the Non-homogeneous model a set of conservation equations is used for each phase, coupled to additional closure models that describe change of mass, momentum and energy at the interface and the turbulence in each phase. In this work the mixture model for coupling of momentum between the two phases was used, with a scale of 1 (one) mm, a drag coefficient of 0.44 and surface stress of 0.07 N/m. A static pressure of 0 (zero) Pa was prescribed at the tube outlet. At the inlet the velocities of the two phases were prescribed, taking into account a uniform profile.

The equations that describe the proposed problem are:

$$\frac{\partial}{\partial t}\left(r_\alpha \rho_\alpha\right) + \nabla \cdot \left(r_\alpha \rho_\alpha \vec{U}_\alpha\right) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(r_\alpha \rho_\alpha \vec{U}_\alpha\right) + \nabla \cdot \left(r_\alpha \rho_\alpha \vec{U}_\alpha \otimes \vec{U}_\alpha\right) = \nabla \cdot \left(r_\alpha \mu_\alpha \left(\vec{U}_\alpha + \left(\vec{U}_\alpha\right)^T\right)\right) - r_\alpha \nabla p + r_\alpha \rho_\alpha \vec{g} + \vec{F}_D \tag{2}$$

$$F_{D,\alpha} = C_D \rho_{\alpha\beta} A_{\alpha\beta}\left(U_\beta - U_\alpha\right) \tag{3}$$

where r represents the volume fraction, ρ the density, the velocity field, p the pressure, g the gravity and F_D the drag force. The subscripts α and β represent the liquid and vapor phase, respectively. C_D is the drag coefficient and A is the interface area, that are presented as follows:

$$C_D = 0.44 \qquad \rho_{\alpha\beta} = r_\alpha \rho_\alpha + r_\beta \rho_\beta \qquad A_{\alpha\beta} = \frac{6 r_\alpha r_\beta}{d_{\alpha\beta}} \qquad d_{\alpha\beta} = r_\alpha d_\beta + r_\beta d_\alpha \tag{4}$$

$$S_{W,\alpha} = r_\alpha r_\beta 0.075 \rho_\alpha \omega_{D\alpha}^2 \qquad \mu_\alpha = \rho_\alpha \frac{\kappa_\alpha}{\omega_\alpha} \qquad \omega_{D\alpha} = C_{D\omega} \frac{6\nu_\alpha}{d_{\alpha\beta}^2} \tag{5}$$

where μ is the dynamic viscosity, S_W is the source term of ϖ equations and $C_{D\varpi}$ is a constant that will be object of investigation in this work.

The sinusoidal expressions presented by Frank (2005) to represent the volumetric fraction initial and boundary conditions were used. Eq. 6 is applied in the all domain as initial condition. Equation (7) defines the time dependent volumetric fraction imposed at the tube inlet as boundary condition.

$$y_1 = y_0 + A_1 sen\left(2\pi \frac{x}{0.5L}\right) \tag{6}$$

$$y_1 = y_0 + A_1 sen\left(2\pi \frac{tV_1}{0.5L}\right) \tag{7}$$

where y_1 is the height of the interface liquid-gas from the reference $y_0 = 0.0m$; L is the tube length, x is the axial position, t is the time, V_1 is the characteristic velocity at the interface, considered equal to the initial velocity of both fluids and A_1 is the liquid level fluctuation amplitude. That amplitude and the liquid and gas initial velocities were object of study in this work too. Only at comparative stage the same values used by Frank (2005) were considered for these variables. They were 0.25D for the amplitude and 2 m/s both for liquid and gas initial velocity. As boundary condition at the tube outlet, relative pressure of 0 (zero) Pa was used. The prescribed time interval was 0.005 s in all analyzed cases.

4. RESULTS

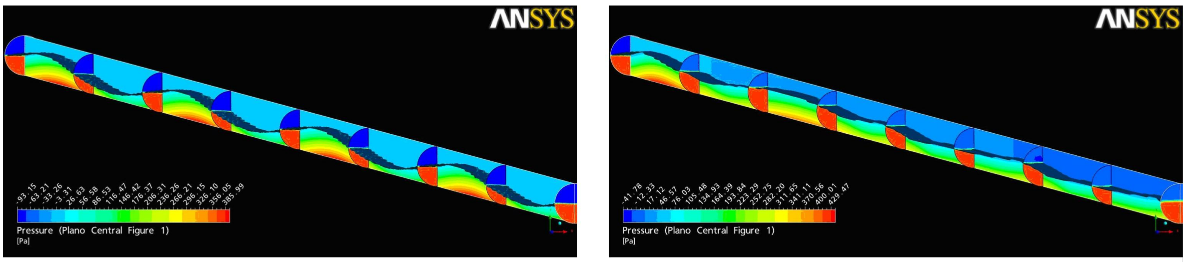
The work was performed in two steps: modeling results validation and investigation of some problem parameters variation. The results are presented in the following sections.

4.1 Validation

Results presented in this section were obtained with the same boundary and initial conditions presented by Frank (2005), i.e., air and water initial velocities of 2 m/s, A_1 = 0.25D, $C_{D\varpi}$ = 1000 and the same number of mesh elements (196000). It must be emphasized that only the number of elements was reproducible since its distribution in the control volume was neither commented nor showed in referred work.

As previously mentioned the volumetric fraction of the fluids was prescribed as a sinusoidal function of time for the initial condition and of the axial coordinate for the boundary condition. That kind of boundary condition in the tube inlet aims to represent a real situation of fluids at device's inlet. By the other hand, sinusoidal initial condition intends to reproduce a steady state flow condition, hoping that the condition keeps sinusoidal, but it does not verify. At the initial period of the simulation, nonetheless the boundary condition keeps sinusoidal, the volumetric fraction profile experiments a transformation from sinusoidal to quasi plane. That process can be partially observed at Fig. 2a and 2b, the first showing the volumetric fraction profile at the starting time and the second at 1.5 s. Those figures show also the pressure field in the symmetry plane. It can be observed that the greatest pressure values are located at the tube base, where maximum pressure points are present, associated to the maximum water level at radial direction.

After that period, the utilized reference authors observed that the first slug formation start occurred after 3.34 s, at 3.8 m from the tube inlet and developed completely after 4.04 s at 4 m from the inlet.



(a) (b)

Figure 2. The 0.5 volumetric fraction isosurface: (a) initial condition; (b) t = 1.5 s.

Although the wave damping characteristic was declared in the reference work, the time intervals are not exactly the same verified in the present work. Frank describes that until a time of 3 s the simulation was characterized by gravitational forces in relation to the initial condition sinusoidal profile of the volumetric fraction, making that profile disappear, or at least reduce substantially its amplitude. In the present work there was also a strong attenuation of the sinusoidal profile, but the first slug appeared at 2.4 s, the second at 3.5 s and the third at 4.65 s. All the three slugs formed practically at the same position: 3.3 m from the tube inlet. The formation sequence can be observed at Fig. 4a-c. Almost simultaneously, at 4.7 s, occur the third and fourth slugs, the last one at 6.5 m (Fig. 3d).

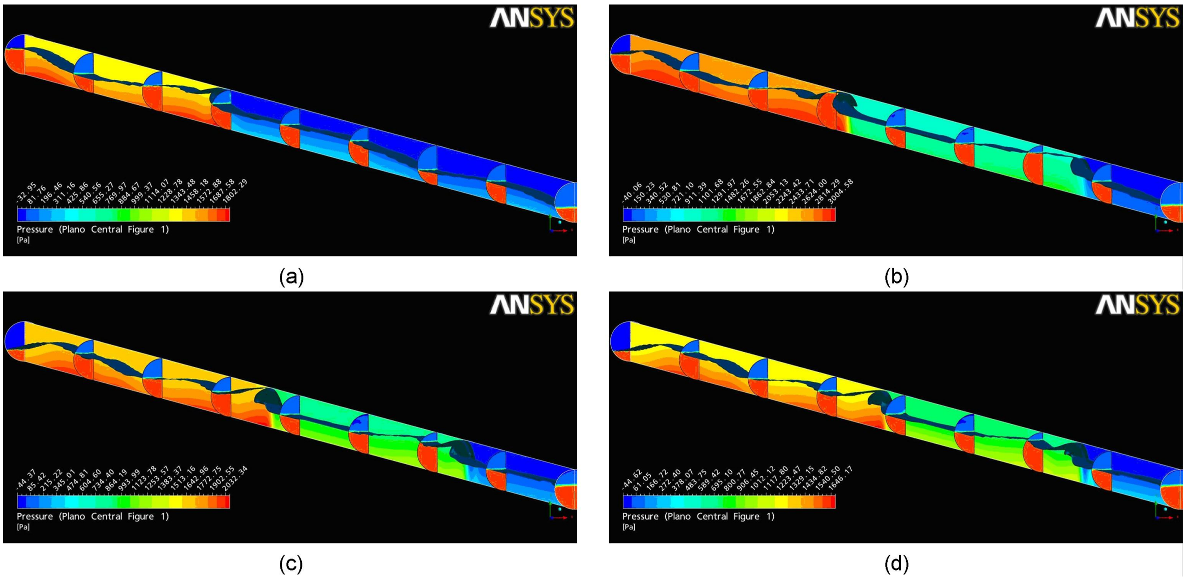


Figure 3. The 0.5 volumetric fraction isosurface: (a) t = 2.4 s; (b) t = 3.5 s, (c) t = 4.65 s and (d) t = 4.7 s.

In Fig.3 may be observed that the complete formation of the slugs occurs at the final of the tube's first half at times of 2.4 s, 3.5 s and 4.65 s. Besides, we can see the formation of another slug at approximately 3/4 of the total tube length, in 3.5 s, but it does not develop like the slugs verified closer to the tube inlet, even at subsequent times not showed in the figure. The authors believe the fact is related to the boundary condition at the tube inlet that prescribes a sinusoidal transient profile for the water level. In that case, the formation of the slugs would be related to the characteristics of the fluid entering at the analyzed region and not to the microscopic instabilities due to the hydrodynamic conditions.

The field pressure saw in the symmetry plane, in the same figure, shows the total tube blocking by the water volume at the slugs closer to the tube inlet formation (Fig. 3a-c). Closer to the tube outlet blocking is not complete, so the pressure does not increase so much.

According to previous comments, the two works differs with relation to the moment of the first slug formation, although the other characteristics are maintained. This can be seen in Figs. 4 and 5, where the water surface velocity and pressure fields are presented respectively.

Observing the evolution of the slugs we can see that their propagation velocity may be estimated at about 3.3 m/s. This value is near the 2.7 to 3.1 m/s obtained in the reference work. However in both works those values are not reliable since it looks that velocity varies along the tube, being higher at its initial part. The little number of slugs present in the flow simultaneously makes less precise the evaluation of the average distance between them.

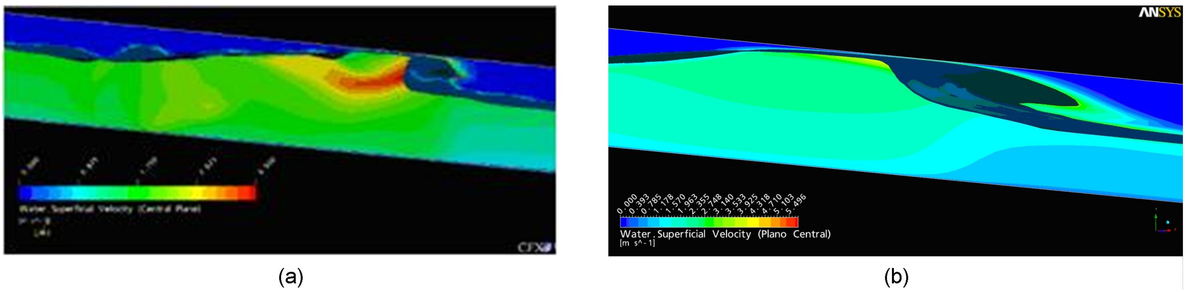
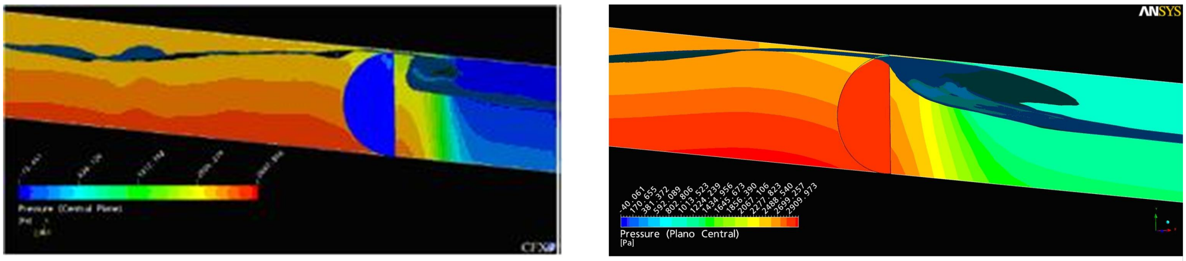


Figure 4. Water superficial velocity. (a) reference, at t = 4.04 s; (b) this work, at 3.5 s.



(a) (b)

Figure 5. Pressure field. (a) reference, at t = 4.04 s; (b) this work, at 3.5 s.

4.2 Case study

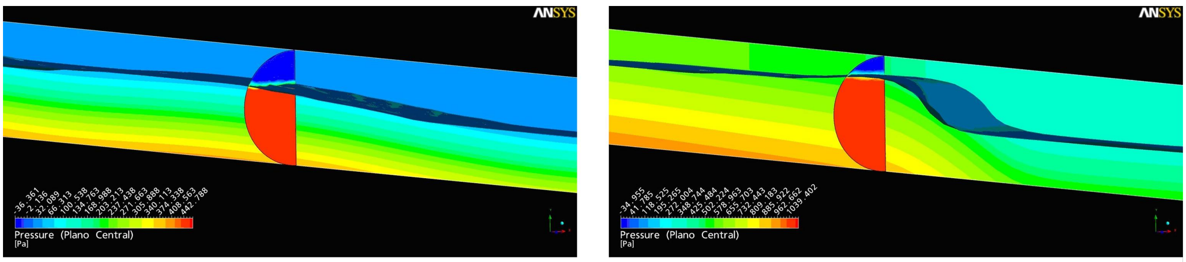
This case study consists on investigating the influence on the flow of initial and boundary amplitudes imposed to the liquid level and of the coefficient $C_{D\varpi}$, especially with relation to slug formation and pressure field. In this case, the tested modifications are presented at Tab. 1.

Table 1. Studied cases

Amplitude	0.25D	0.35D	0.25D	0.25D	0.25D
$C_{D\varpi}$	1000	1000	10	1000	10
V_{air} (m/s)	2	2	2	2	2
V_{water} (m/s)	2	2	2	1.5	1.5

Helmholz instability occurs when the relative velocity at the separation surface between the two immiscible fluids reaches such a value, beyond it a little perturbation of the interface will amplify, grow and distort the flow (Tong and Tang, 1997). In flow patterns maps like those presented by Tong and Tang (for Dukler's theoretical and Mandhane et al. experimental results) for water-air flow at 25°C and 1 atm, Helmholz instability is represented by a dividing line between regions of stratified and intermittent flow.

In the reference work water and air velocity at flow inlet equals 2 m/s, corresponding to initial superficial velocities of 1 m/s. In flow patterns maps that correspond to the region of slug flow. In order to verify the numerical model behavior with respect to the relative velocity variations, results were obtained for inlet velocity of 1.5 m/s for water and 2 m/s for air, corresponding to superficial velocities of 0.75 m/s and 1 m/s respectively. Similarly to the originals, that pair of velocity values is located in the region of intermittent flow. Nevertheless, when reducing the water velocity, the numerical model was not able to reproduce the slugs formation; the oscillations imposed at the inlet extinguished gradual and successively along the flow, replaced by a smooth interface, as showed at Fig. 6a. Probably that fact is due to the insufficient production of turbulence in the interface, caused by the reduction of liquid velocity, caused by the excess of the source term responsible for turbulent diffusion damping, nullifying the perturbations instead of amplify them until slugs formation.



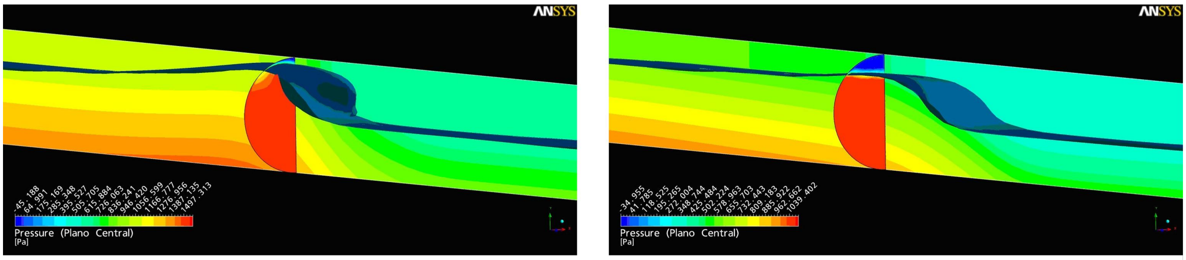
(a) (b)

Figure 6. Slugs formation for water velocity of 1.5 m/s: (a) $C_{D\varpi}$ = 1000, (b) $C_{D\varpi}$ = 10.

Correct flow modeling requires that the turbulent diffusion in the interface be damped by the dissipation term, to avoid yielding of unreal velocities (Valée et al., 2008). That is obtained by inclusion of a source term, set to reinforce turbulence (ϖ in $\kappa - \varpi$ model) destruction at respective equation, at interface region. Nevertheless, turbulent diffusion excessive damping can hinder the perturbation amplifying mechanism responsible for slugs formation.

In order to investigate turbulence diffusion attenuation, the coefficient $C_{D\varpi}$ of source term included in dissipation equation (set on 1000 at the reference work simulation) was varied to 10. Fig. 6b shows the result of reduction of both coefficient $C_{D\varpi}$ to 10 and water velocity to 1.5 m/s. In this case damping, that rested excessive, facilitates slug onset. Perhaps a more detailed study of relations among damping and the other variables provides a more general criterion for correct coefficient $C_{D\varpi}$ determination.

Fig. 7 shows the influence of coefficient $C_{D\varpi}$ = 10, in case of equal (2 m/s; Fig. 7a) and different (Fig.7b) velocities for both fluids.



(a) (b)

Figure 7. Slugs formed with coefficient $C_{D\varpi}$ =10, for initial water velocity of (a) 2 m/s and (b) 1.5 m/s.

Those results clearly show that smaller values of $C_{D\varpi}$, that implies less turbulence dissipation, produces pseudo-slugs, well defined but almost punctual, being rapidly damped at upstream and downstream regions. That does not correspond to physical reality as seen, e.g., in Valée et al. (2008) and Jabardo and Bandarra Filho (2000)

5. CONCLUSIONS

Numerical simulation water-air two-phase flow with slug formation presented by Frank (2005) was reproduced in this work. Results were compared with the reference work and some variations on original proposal parameters were tested.

Results obtained on this work present qualitative and quantitative characteristics like pressure and velocity fields similar to the reference work as well as slugs formation characteristics, except their formation times: in this work slugs appear in lower times than at the reference work.

Parameters variations presented coherent results with physics and numerical fundaments of the problem. Nevertheless, the theme is not exhausted, the authors suggesting the analysis of other parameter variations, namely fluid velocities and its relation with turbulent diffusion damping coefficient.

6. REFERENCES

Carey, Van P., 2008, "‘Liquid-Vapor Phase-Change Phenomena", Taylor e Francis.

CFX 11.0, "Solver Manual".

Frank, T., 2005, Numerical Simulation of Slug Flow Regime for an Air-Water Two-Phase Flow in horizontal Pipes, "International Tropical Meeting on Nuclear Reactor Thermal-Hidraulics" (NURETH-11).

Tong, L. S., Tang, Y. S., 1997, "Boiling Heat Transfer and Two-Phase Flow", Taylor e Francis.

Soviero, P.A.O. and Lavagna, L.G.M.,1997, "A Numerical Model for Thin Airfoils in Unsteady Motion", RBCM- J. of the Brazilian Soc. Mechanical Sciences, Vol.19, No. 3, pp. 332-340.

Valée, C., Höhne, T., Prasser, H. M., Sühnel, T., 2008, Experimental Investigation and CFD simulation of horizontal stratified two-phase flow phenomena, "Nuclear Engineering and Design", vol. 238, pp. 637-646.