

NUMERICAL SIMULATION OF ISOTHERMAL UPWARD TWO-PHASE FLOW IN A VERTICAL TUBE OF ANNULAR SECTION

Flávio Eduardo Ceravolo

Nuclear and Energy Research Institute, IPEN-CNEN/SP
Nuclear Engineering Center, CEN
Av. Prof. Lineu Prestes, 2242, 05508-000, São Paulo/SP
flavioceravolo@gmail.com

Marcelo da Silva Rocha

Nuclear and Energy Research Institute, IPEN-CNEN/SP
Nuclear Engineering Center, CEN
Av. Prof. Lineu Prestes, 2242, 05508-000, São Paulo/SP
msrocha@ipen.br

Fábio Branco Vaz de Oliveira

Nuclear and Energy Research Institute, IPEN-CNEN/SP
Nuclear Engineering Center, CEN
Av. Prof. Lineu Prestes, 2242, 05508-000, São Paulo/SP
fabio@ipen.br

Delvonei Alves de Andrade

Nuclear and Energy Research Institute, IPEN-CNEN/SP
Nuclear Engineering Center, CEN
Av. Prof. Lineu Prestes, 2242, 05508-000, São Paulo/SP
delvonei@ipen.br

A numerical simulation of a vertical, upward, isothermal two-phase flow of air bubbles and water in an annular channel applying Computational Fluid Dynamics code (CFD) was carried out. The simulation considers an Eulerian frame, with two-fluid model, specific correlations for turbulence model considering the dispersion and bubble induction turbulence. The work intends to assess whether the code represents the physical phenomenon accurately by comparing the simulation results with experimental data obtained from literature. The annular channel adopted has equivalent hydraulic diameter of 19.1 mm, where the outer pipe has an internal diameter of 38.1 mm and inner rod 19.1 mm. To represent this geometry, a three-dimensional mesh was generated with 960000 elements, after a mesh independence study. The void fraction distribution, taken radially to the flow section is the main parameter analyzed besides interfacial area concentration, interfacial gas velocity, diameter and distribution of bubbles.

Keywords: Bubble column, two-phase flow, annular flow, CFD.

1. INTRODUCTION

Isothermal gas-liquid flow in annular channels can provide a research basis for more complex systems such as a reboiler in a distillation column or the flow channel in a fuel rod of a BWR reactor. Several experimental studies using annular flow have been conducted to study interfacial area transport mechanisms and void fraction in liquid-gas flow in order to validate models representing such a phenomenon (Hibiki & Ishii, 2002).

Numerical modeling is an important tool in thermal-hydraulic analysis of multiphase systems. The main interest is related to prediction of parameters representing the physical phenomena as: void fraction, flow velocity of the phases, pressure drop, heat transfer coefficients, interfacial area, etc.

In the nuclear area, typically highly reliable one dimensional computational code is used to describe two-phase flow. Examples of these codes are RELAP5, TRACE and CATHARE that have been used since the 1980s in thermo-hydraulic design of nuclear reactors and accident analyses during the licensing process of installations.

Currently, due to the increased processing power of personal computers, commercial tools of CFD (Computational Fluid Dynamics) have gained evidence in the nuclear area (Yadigaroglu, 2005).

This tool allows working with complex three dimensional geometries, simulating local effects that could not be analyzed by conventional tools, which often perform one-dimensional analysis. However, for the CFD technique achieve a degree of reliability comparable to traditional thermo-hydraulic code, a long phase of testing and validation of the tool must be conducted under various conditions.

Multiphase flow field involving the application of CFD is, currently, a promising development area. Mainly due to the great complexity of physical phenomena, which involves mathematical models of equally complex numerical solution (van Wachem & Almsedt, 2003).

This work aims to apply a CFD code, Ansys-Fluent 12, to the representation of an isothermal rising air-water flow in an annulus channel. For this, experimental data from work (Hibiki et al., 2003) are used as comparison parameters.

2. MATHEMATICAL MODELING FOR TWO-PHASE FLOWS

The mathematical model is divided into a set of equations describing the conservation of mass, conservation of momentum, turbulence phenomena and interaction between the phases.

2.1 Governing Equations

2.1.1 Mass Conservation Equation

The Equation (1) represents the continuity of mass for the phase k , where it disregards the mass transfer at the interface, this is assumed valid for the air-water which has no phase change.

$$\frac{\partial}{\partial t}(\alpha_k \rho_k) + \nabla(\alpha_k \rho_k \vec{v}_k) = 0 \quad (1)$$

In the above equation α_k represents the volume fraction of phase k (given by the Equation (2)), ρ_k its density and $\vec{v}_k$ corresponds to the phase velocity.

$$\sum_{k=1}^n \alpha_k = 1 \quad (2)$$

2.1.2 Momentum Conservation Equation

The momentum balance of each phase is given by the Equation (3).

$$\frac{\partial}{\partial t}(\alpha_k \rho_k \vec{v}_k) + \nabla(\alpha_k \rho_k \vec{v}_k \vec{v}_k) = -\alpha_k \nabla P - \nabla(\alpha_k \tau_k) + \alpha_k \rho_k \vec{g} + F \quad (3)$$

Among the Equation (3) terms, P , is the pressure at which the phases are subjected, F represents the forces acting over the phases. These forces are composed by drag forces, lift forces, wall lubrication force, turbulent dispersion force.

In the Equation (3) the tensor τ_k is given by the Equation (4):

$$\tau_k = -\mu_{eff}[(\nabla \vec{v}_k + \nabla \vec{v}_k^T) - \frac{2}{3}(\nabla \cdot \vec{v}_k) I] \quad (4)$$

Where μ_{eff} is the effective viscosity of the liquid according the Equation (5):

$$\mu_{eff,L} = \mu_L + \mu_{T,L} + \mu_{BIT,L} \quad (5)$$

The terms μ_L , $\mu_{T,L}$ and $\mu_{BIT,L}$ correspond respectively to molecular viscosity, resulting viscosity of turbulence and viscosity due to the bubble-induced turbulence.

For the gas viscosity the following ratio is used:

$$\mu_{eff,G} = \mu_{eff,L} \frac{\rho_G}{\rho_L} \quad (6)$$

Among the forces acting on the phases only the drag force parameter is taken into account in Equation (4) due to the availability of models in the software version used (Ansys Fluent, 2009).

2.2 Turbulence Models

In this paper the model to account turbulence was an extension of standard k- ε model, proposed by (Launder and Spalding, 1972) which is applicable to fully turbulent flow. This model is based on the following conservation equations for the kinetic energy k and ε turbulence dissipation.

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla(\rho_m k \vec{v}_m) = \nabla \left[\left(\mu_m + \frac{\mu_{t,m}}{\sigma_k} \right) \nabla k \right] + G_{k,m} - \rho_m \varepsilon + S_{k,m} \quad (7)$$

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla(\rho_m \varepsilon \vec{v}_m) = \nabla \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} G_{k,m} - C_{\varepsilon 2} \rho_m \varepsilon) + S_{\varepsilon,m} \quad (8)$$

In the Equations above, ρ_m is the density, μ_m the molecular viscosity, and $\vec{v}_m$ velocity for the mixture. The turbulence viscosity $\mu_{t,m}$ is described by:

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon} \quad (9)$$

And $G_{k,m}$ is the production of kinetic energy, and is given by:

$$G_{k,m} = \mu_{t,m} [\nabla \vec{v}_m + (\nabla \vec{v}_m)^T] : \nabla \vec{v}_m \quad (10)$$

S_k and S_ε are the source terms due to the kinetic energy generation and dissipation induced by the bubbles. Furthermore, the values of the constants given by this model are $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$ and $C_\mu = 0.09$.

2.3 Interfacial Area Concentration Model

The interfacial area concentration (IAC) is defined as the area of interface between the two phases by mixture unit volume. The Equation (11) defines the transport of IAC as follows:

$$\frac{\partial}{\partial t}(\rho_g X_k) + \nabla(\rho_g \vec{u}_g X_k) = \frac{1}{3} \frac{D\rho_g}{Dt} X_k + \frac{2}{3} \frac{\dot{m}_g}{\alpha_g} X_k + \rho_g (S_{RC} + S_{WE} + S_{TI}) \quad (11)$$

X_k represents the concentration of interfacial area, ρ_g is the gas density, α_g the gas volume fraction. The first two terms on the right hand of the Equation (11) stand for the expansion due to the compressibility and mass transfer respectively. The term $\dot{m}_g$ corresponds to the mass transfer rate in the gas phase by volume of mixture. The parameters S_{RC} and S_{WE} correspond to source and sink terms due to random collision and wake entrainment respectively. The S_{TI} is breakage source term due to turbulence.

Since the flow regime studied is typically a bubbly flow (10% void fraction), only the effects of coalescence and breakage due to turbulence collisions and eddies are relevant. Therefore to represent the source and sink terms for the interfacial area concentration are considered the models based on the work of (Hibiki & Ishii, 2000).

3. RESULTS AND DISCUSSIONS

3.1 Geometry and Mesh Discretization

The geometry used in the study is an annulus channel with hydraulic diameter equivalent to 19.1 mm, where the outer pipe internal diameter is 38.1 mm, the inner rod is 19.1 mm, and the channel length is 800 mm. The mesh discretization proposed to represent this geometry is shown in Figure 1 (a) and (b).

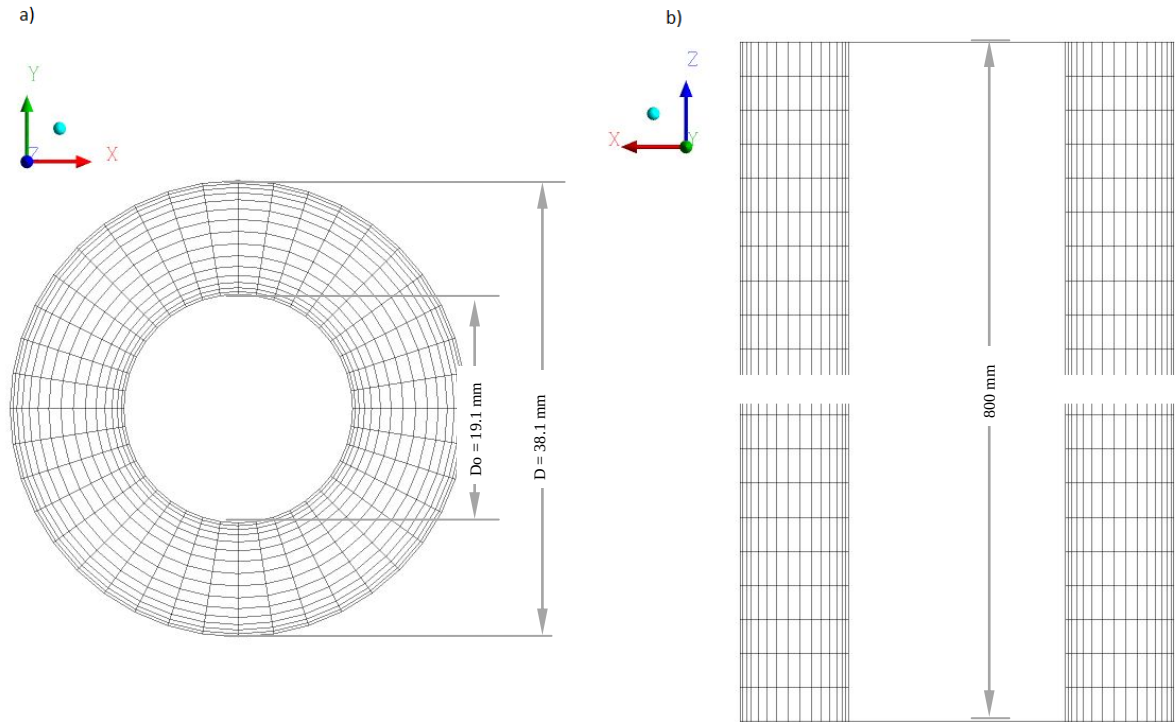


Figure 1. Mesh discretization: (a) cross section view, and (b) side view.

For the mesh construction, it was used predominantly hexahedral control volumes of regular dimensions for the greater uniformity of the mesh and, consequently, a reduction in processing time. Next to the inner and outer walls of the duct was printed a refinement of the mesh in order to seek a more appropriate reproduction of velocity gradients near these walls.

To establish the degree of refinement of the mesh, a mesh independence study was carried out considering 170,000, 480,000 and 960,000 elements (here called 170K, 480K, and 960K). The Figure 2 shows the comparison of mesh tests for each variable of interest.

The mesh 170K showed slight deviation from results of mesh 480K and 960K which indicates that, the results could be independent of the mesh in this last range. Despite the 480K mesh already presented satisfactory results, it was adopted the mesh 960K due to the possibility of extracting up more detailed results from it and do not represent a large computational effort.

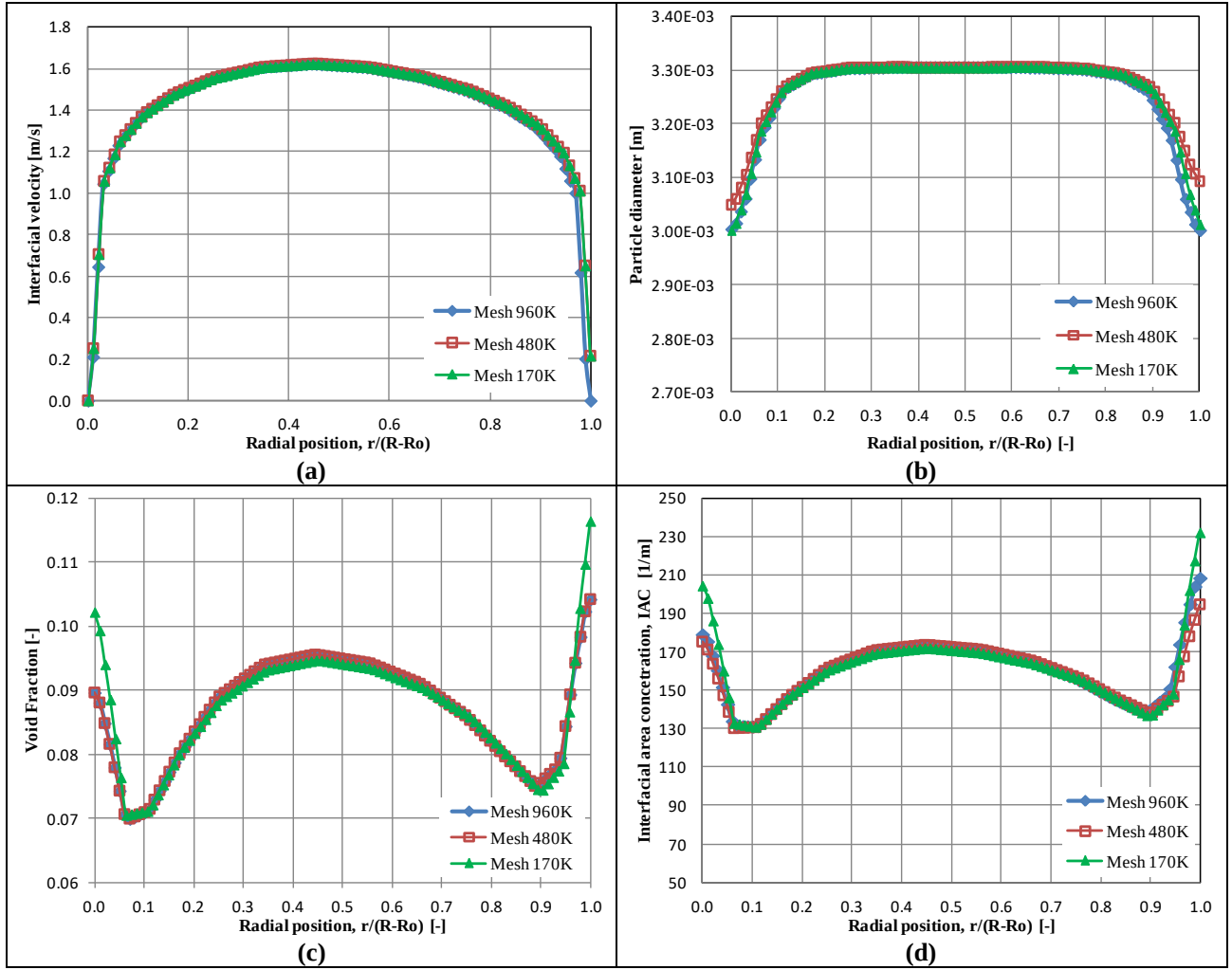


Figure 2. Mesh independence study results: (a) Interfacial velocity; (b) Particle Diameter; (c) Void Fraction; (d) IAC.

3.2 Simulation Results

The simulation results were compared with experimental data from (Hibiki et al., 2003), to do so, they are adopted as reference of this work the following conditions:

- Fluid: air-water at 20°C;
- Surface velocity: Liquid: 1.03 m/s; Gas 0.13 m/s;
- Void fraction: 0.1;
- Axial position: 0.770 m (from the inlet);
- Bubble diameter at the inlet: 2 to 3 mm;
- Interfacial area concentration at the inlet: 200 m⁻¹;
- Turbulence intensity at the inlet: Case 1: 10%; Case 2: 1%.

Considering that the available experimental data do not allow to model the geometric characteristics of the annular channel inlet, and that the effects of turbulence and mixture at the inlet can influence the distribution profile of bubbles in the flow, two cases of evaluation were investigated considering turbulence intensities at inlet of 10% In Case 1 and 1% in Case 2.

The Figure 3 presents a comparison of the results of interfacial velocity profile between the simulations performed for both cases and literature experimental data. Qualitatively, it is observed similarity between the experimental data and simulation, besides the simulation results are on average 20% higher the literature data.

The velocity profile obtained for the annular section resembles that observed in circular geometries where the gas phase is drawn into the liquid phase and the center of the channel suffers less influence of wall effects, resulting in higher velocities in this region. It is observed a small difference between the results obtained from the two cases, the velocity in the Case 1 is slightly higher than in the Case 2.

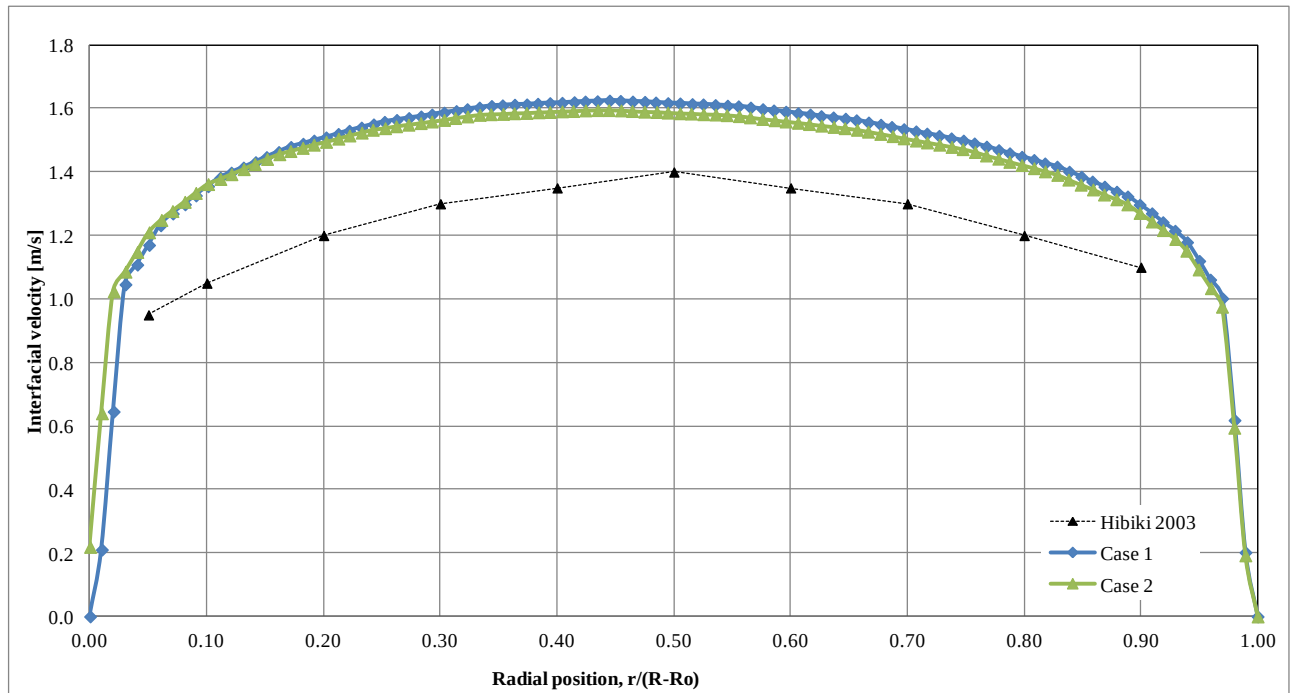


Figure 3. Interfacial velocity profile.

The results obtained for the distribution of bubbles are shown in Figure 4, where the diameters are in the range of approximately 3 mm and 3.3 mm predominantly in the central region of the flow. There was no significant variation in bubble diameter profile as observed in the literature data.

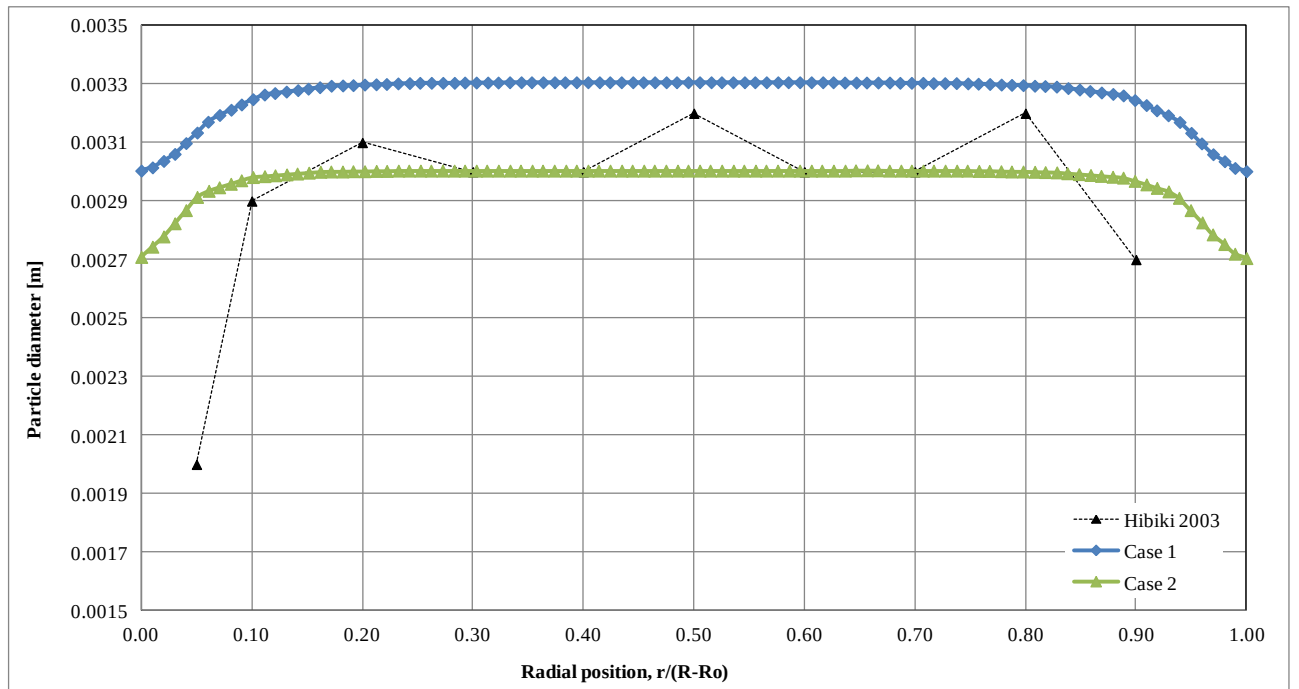


Figure 4. Particle diameter profile.

The void fraction profile shown in Figure 5 obtained by the simulation showed significant deviation from the literature data, which feature a wide range of void fraction, between 0.04 and 0.18, where the maximum values are close to the wall and the minimum values at center of channel. This profile is classified by (Serizawa & Kataoka, 1988) as "wall peak" pattern.

The simulation results showed relatively higher values close to the walls, however, with values ranging from 0.07 to 0.11 for the Case 1 and 0.09 to 0.12 for the Case 2. For both cases this flat behavior of the void fraction profiles can be related to the overestimation of the turbulence effects which tends to increase the bubble mixing.

The lower turbulence intensity applied in Case 2 presented better qualitative results compared to Case 1, since the central parabolic region was reduced and the distribution of void fractions tended to have a higher concentration near the walls and lower concentration in the center, tending to a "wall peak" pattern.

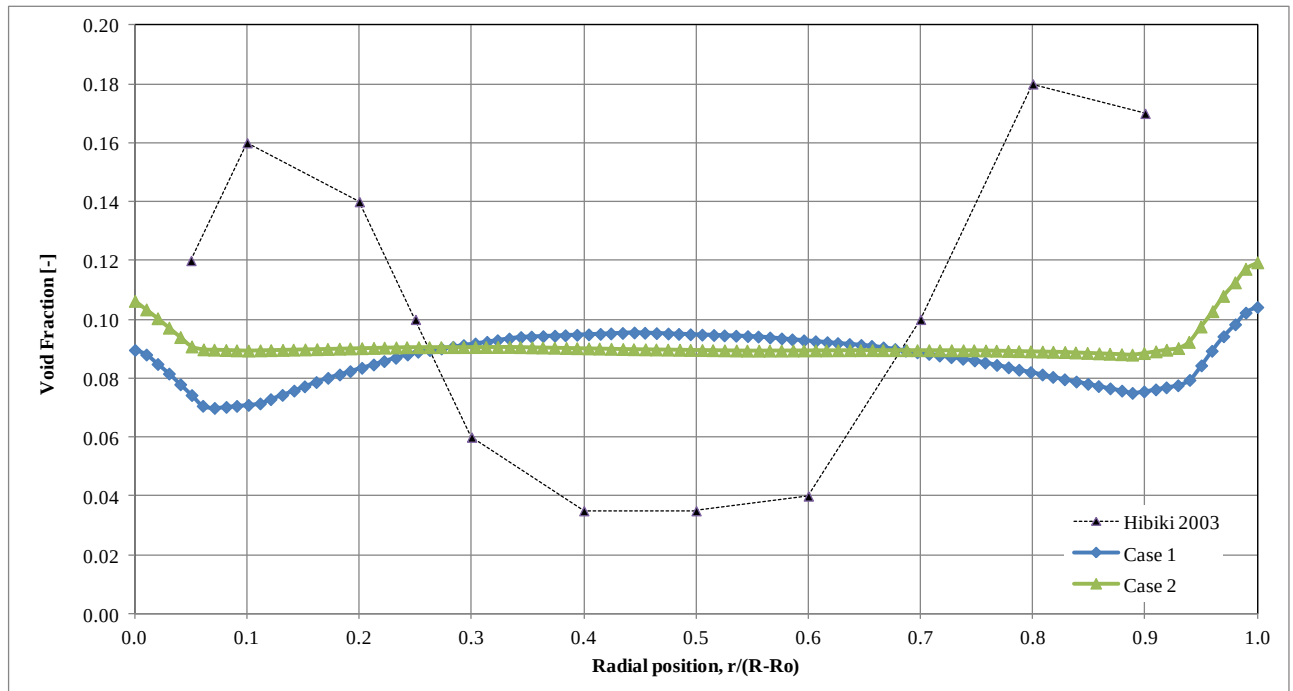


Figure 5. Void fraction profile.

The above results indicate the need to further study bubble coalescing and breakup models, including analyzing the influence of the turbulence models employed in the results obtained, as well as to analyze the influence of interaction forces between the phases such as lubrication force and turbulent dispersion force that were disregarded due to the software limitations.

Proper representation of the flow turbulence has significant influence on the bubble flow. Studies considering this influence were performed at the work of (Masood et al., 2014). Input parameters should be investigated as well, since IAC values and void fraction were not controlled by the author of the experiments (Hibiki et al., 2003).

As the IAC parameter is proportional to the void fraction, the results presented similar behavior as those shown above. Figure 6 presents the comparison of results between simulation and literature data. Likewise, they show significant deviations with the experimental results.

Here the results of the Case 2 presented better behavior in comparison with Case 1, probably due to the reduction of the turbulence intensity in the boundary conditions.

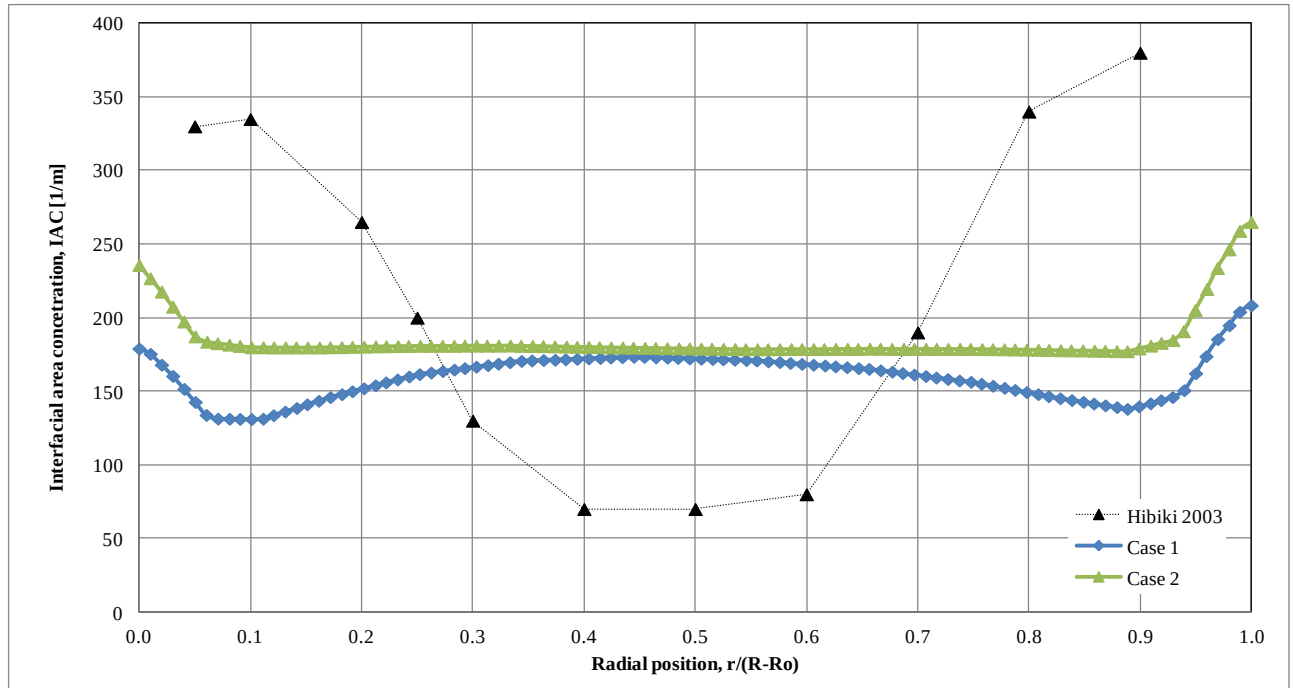


Figure 6. Interfacial area concentration (IAC) profile.

4. CONCLUSIONS

The work carried out the simulation of an ascending isothermal of two-phase flow in a annulus channel using a CFD code. The results were compared with experimental data available in the literature. Is observed good proximity results for the velocity profile and distribution of bubbles, however, regarding to the void fraction distribution and concentration of the interfacial areas, the results were not satisfactory. It indicates the need for investigating possible influences caused by models which describe the interaction forces between phases and the flowing turbulence.

5. REFERENCES

- Ansyes Fluent, 2009. ANSYS Fluent Theory Guide, Release 12. ANSYS Inc.
- Hibiki, T. et al., 2003. Local flow measurements of vertical upward bubbly flow in an annulus. *International Journal of Heat and Mass Transfer*, 46(2003), pp.1479–1496.
- Hibiki, T. & Ishii, M., 2002. Development of one-group interfacial area transport equation in bubbly flow systems. *International Journal of Heat and Mass Transfer*, 45, pp.2351–2372.
- Hibiki, T. & Ishii, M., 2000. One-group interfacial area transport of bubbly flows in vertical round tubes. *International Journal of Heat and Mass Transfer*, 43(15), pp.2711–2726.
- Launder, B.E. & Spalding, D.B., 1972. *Lectures in mathematical models of turbulence* London, ed., New York.
- Masood, R.M.A., Rauh, C. & Delgado, A., 2014. International Journal of Multiphase Flow CFD simulation of bubble column flows: An explicit algebraic Reynolds stress model approach. *International Journal of Multiphase Flow*, 66, pp.11–25.
- Serizawa, A. & Kataoka, I., 1988. Phase distribution in two-phase flow, Transient Phenomena in Multiphase Flow. In *Transient Phenomena in Multiphase Flow*. Washington DC, pp. 179–224.
- van Wachem, B.G.M. & Almstedt, A.E., 2003. Methods for multiphase computational fluid dynamics. *Chemical Engineering Journal*, 96(1–3), pp.81–98.
- Yadigaroglu, G., 2005. Computational Fluid Dynamics for nuclear applications: from CFD to multi-scale CMFD. *Nuclear Engineering and Design*, 235, pp.153–164.

6. RESPONSIBILITY NOTICE

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