Studies of simulations of two-phase water-air flows using ANSYS CFX

A. M., Garrido Filho[1], M. L., Moreira[2], J. L. H., Faccini[2]
e-mail: anizio@ien.gov.br, faccini@ien.gov.br, malu@ien.gov.br

[1] *DIGIN, IEN*
[2] *SETER, IEN*

Keywords: two-phase flow, computational fluid dynamics, CFD, CFX.

This work presents a study of Computational Dynamic Fluid (CFD), applied to an experiment, currently under construction at the Thermo-Hydraulic Laboratory of the Nuclear Engineering Institute (LTE/IEN) [1]. This experiment will test two-phase flows in vertical and inclined positions
This experiment consists of columns of transparent acrylic tubes with an extension of 8 meters in which there will be forced circulation of water and compressed air. A frame receives these columns and counts at the bottom with a swivel, so that the entire assembly can assume any degree of inclination from 0 ° to 90 °. This feature will allow testing the flows in any degree of inclination (Fig.1).

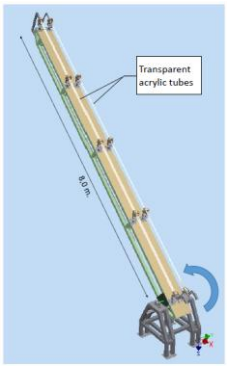

Figure 1. Experiment in inclined position.

This experiment will provide great freedom of configuration, especially at the entry point of the columns. Figure 2 shows an example of entry point under study. The use of CFX will also allow defining new conceptions that bring advantages to this Experiment.
CFD simulations are now a great tool for analyzing fluid dynamics. Commercial CFD software used in this study, ANSYS CFX [2], has a record of good performance for more than two decades. It uses the Navier-Stokes equations as the basis for solving the flow simulations together with turbulence analysis models.

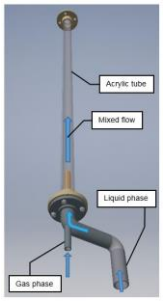

Figure 2. Proposed system of entry point for the experiment.

This study used in the simulations, the k-ε model, which is a good turbulence analysis models presented in the program. It also used the well-established vertical flow pattern map developed by *Taitel et al* [3] as a guide. Some points of this map were then selected, corresponding to the known flow patterns and these data transferred to the CFX to simulate the flow. Then, the program simulate the influence on the flows with the same data under slope conditions.
CFX has a wide range of visualization tools for analyzing the results of the simulations. The contour view of the variation profile of a given parameter is the most used and allows evaluating well the influence of the angle of inclination on a flow, as can be seen in figure 3. This form of analysis was the basis of this study for all types of selected flows.

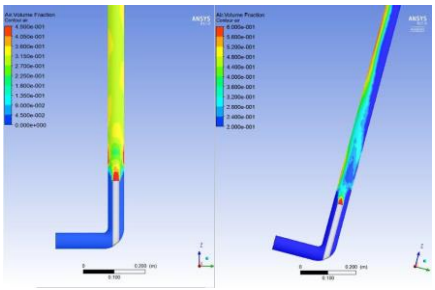

Figure 3. Simulation of the influence of the angle of inclination on a two-phase flow.

References

[1] FACCINI, J. L. H**. Estudos de escoamentos bifásicos gás-líquido em tubos verticais e.inclinados**. Divisão de Reatores, Rio de Janeiro:IEN/CNEN, 2011.
[2] ANSYS. ANSYS WORKBENCH 17.0. Pttsburgh: PRNewswire, 2016 Disponível em: <www.ansys.com>. Acesso em: 23 mar. 2018.
[3] TAITEL, Y.; BORNEA, D.; DUCKLER. A. E. Modelling flow pattern transitions for stead upward gas-liquid flow in vertical tubes. **Journal of Advances in Chemical Engineering**, Amsterdam, v. 26, n. 3, p 345-354, 1980.