2017 International Nuclear Atlantic Conference - INAC 2017
Belo Horizonte, MG, Brazil, October 22-27, 2017
ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR – ABEN

Agência Brasileira do ISBN
ISBN 978-85-99141-07-6

STUDIES OF SIMULATIONS OF TWO-PHASE WATER-AIR FLOWS USING ANSYS CFX.

Anizio M. Garrido Filho[1], Maria de Lourdes Moreira[2] and José L. H. Faccini[3]

[1] Instituto de Engenharia Nuclear (IEN / CNEN – RJ)
Divisão de Gestão e Infraestrutura
Rua Hélio de Almeida, 75
21941-972 Rio de Janeiro, RJ
anizio@ien.gov.br

[2] Instituto de Engenharia Nuclear (IEN / CNEN – RJ)
Divisão de Engenharia de Reatores
Serviço de Engenharia e Tecnologia de Reatores
Rua Hélio de Almeida, 75
21941-972 Rio de Janeiro, RJ
malu@ien.gov.br

[3] Instituto de Engenharia Nuclear (IEN / CNEN – RJ)
Divisão de Engenharia de Reatores
Serviço de Engenharia e Tecnologia de Reatores
Rua Hélio de Almeida, 75
21941-972 Rio de Janeiro, RJ
faccini@ien.gov.br

ABSTRACT

Normally in all simulations of flows in computational fluid dynamics, CFD, it is common to use characteristic planes to visualize the profiles of the parameters of interest, mainly in 3D simulations.

The present work proposes a standard form of visualization that shows, mainly in two-phase flows, in a more realistic way, the dynamics of the development of the phase flow. This visualization is present within the CFX program in the post-processing module, in the option of representing volumes using sub option, isovolumes. Through this representation, the program highlights the volumes of the finite element mesh corresponding to the selected values of the parameter to be analyzed such as pressure, velocity, volumetric fraction, etc.

By means of the volume-isovolume representation, a well representative effect of the current flow pattern is obtained, especially when the volumetric fraction of the air or the gas phase of the flow is emphasized. This form of visualization is being applied to the study of inclined two-phase flows, which will be tested in a new experiment currently under construction at the Laboratory of Experimental Thermal-Hydraulics – LTE of the Institute of Nuclear Engineering - IEN in Rio de Janeiro.

1. INTRODUCTION

The present work presents a computational study of gas-liquid two-phase flows in vertical and inclined pipes. Computational simulations were performed using a commercial CFD software of a two-phase section for the study of vertical and inclined gas-liquid flows currently under construction at the Laboratory of Experimental Thermal-Hydraulics – LTE - of the Institute of Nuclear Engineering - IEN.

Pipes of transparent material that can be inclined form the two-phase section. Working fluids are compressed air and water at ambient temperatures and pressures. The flows that will be generated in the section comprise bubble flow, slug, churn and annular flow.
The present work proposes a form of visualization that shows, mainly in two-phase flows, in a more realistic way, the dynamics of the development of the flow of the phases. This form of visualization, by isovolumes, is present within the ANSYS CFX program in the post-processing module and curiously is not normally employed in most studies. The representation by isovolumes allows a good evaluation of the geometry of the gas phase inside the flow. This facilitates the identification of current standards.

2. THE TWO-PHASE FLOW EXPERIMENT

The LTE Two-Phase Flow Experiment (FACCINI 2011), consists of a piping circuit in which water driven by hydraulic pump will circulate along with a compressed pressurized air flow. Once the two mixed streams have been run through the study section, they are depleted in a tank open to the atmosphere, from which the water is drawn by the hydraulic pump (Figure 1).
The study section of the system is composed of two columns in transparent acrylic tubes with internal diameters of 1 and 2 inches divided into 4 segments of 2 meters each (Figure 1). The two sections are mounted side by side in a frame that counts at the bottom with a pivoting support, so the whole assembly can assume any degree of inclination from 0 to 90°. This functionality allows flows to be tested at any degree of slope (Figure 2).
This experiment is designed to provide great freedom of configuration, especially at the entry point of the test sections. For example, the present study used the project proposed in Figure 3. This project of entry of the sections has as premise to provide a better distribution of the gas phase with the liquid one. However, with the use of CFX, it is possible in the future to test other conceptions that can bring advantages to this entry point of the phases.

Agência Brasileira do ISBN
ISBN 978-85-99141-07-6

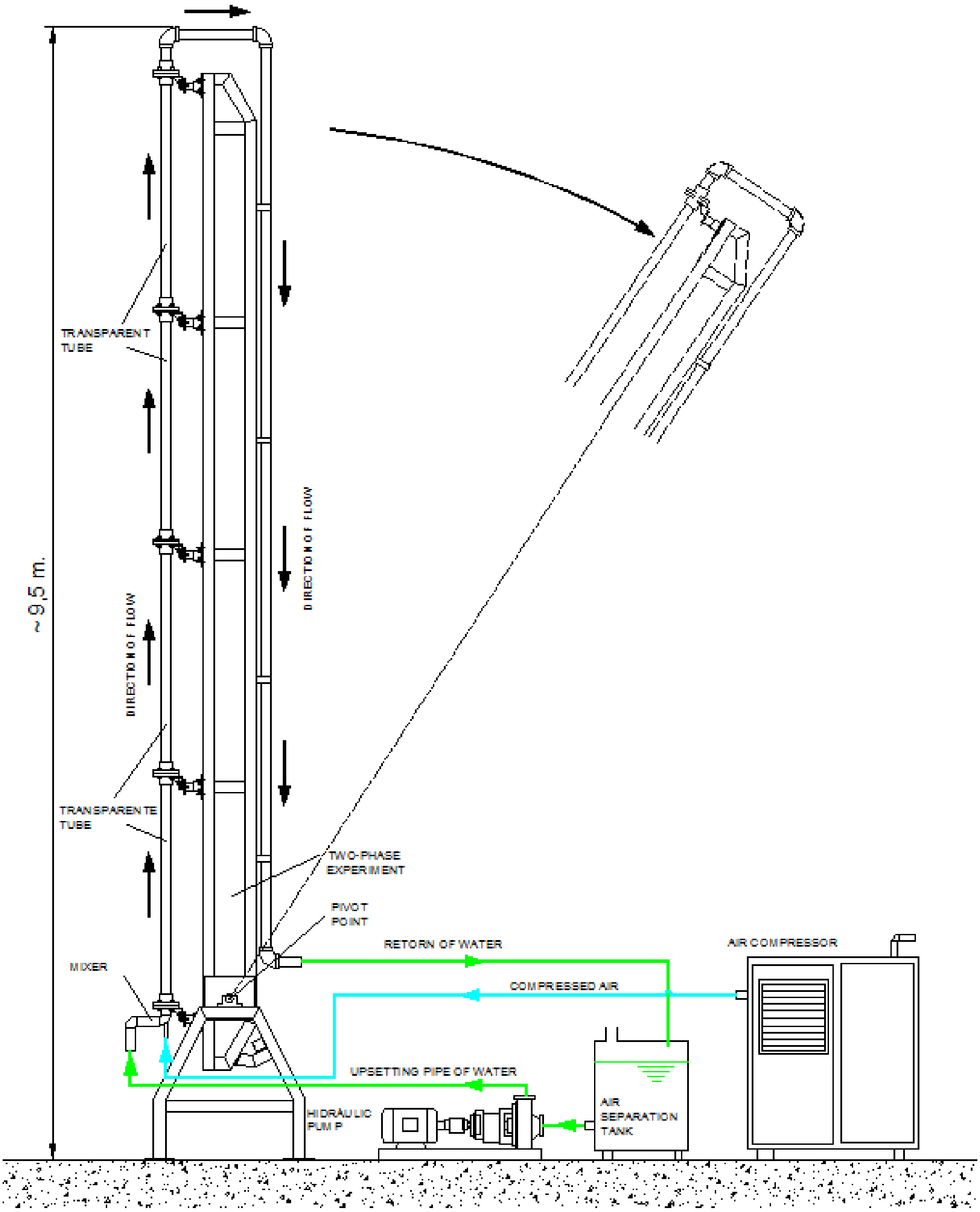


Figure 1 - Scheme for the operation of the Two-Phase Flow Experiment of the IEN Experimental Thermo-Hydraulic Laboratory.

Figure 2 – View of the experiment in inclined position.

Agência Brasileira do ISBN
ISBN 978-85-99141-07-6

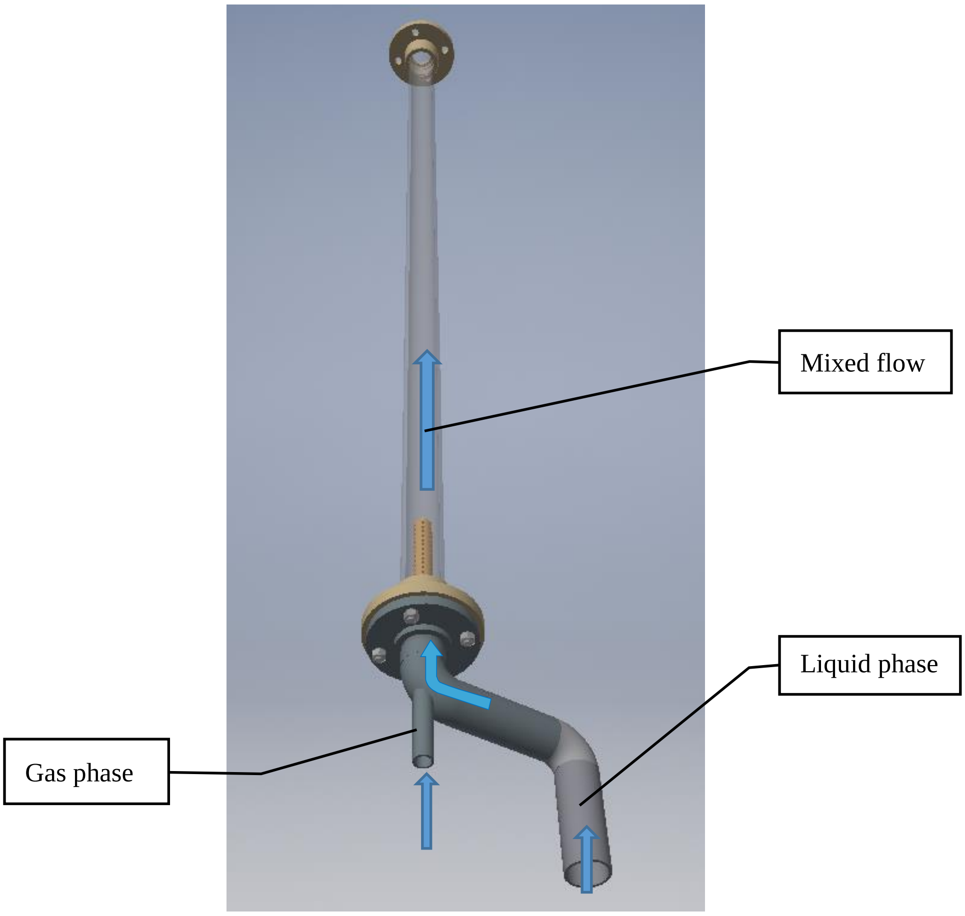


Figure 3 - Proposed system for entering the phases in the experiment.

3. DEVELOPMENT OF THE STUDY IN ANSYS CFX.

CFD simulations are a powerful tool for analyzing processes, products and materials. In the field of fluid dynamics has its space increasingly consolidated by the great ease it provides allowing varying the conditions of geometry, boundary conditions and parameters involved, without the need to elaborate models and experimental models, including the possibility of simulations of extremely conditions Difficult to reproduce and even risky.
This study will be dedicated to the use of the CFX program in version 17.0 of ANSYS Inc. (ANSYS 2016); the Institute of Nuclear Engineering has a license to use this program.
CFX is a Computational Fluid Dynamics (CFD) program, used to simulate the flow of fluids in a variety of applications and situations. It has the advantage of taking advantage of data and information common to many simulations based on 20 years of studies.

3.1. Models and Conditions employed.

ANSYS CFX uses the Navier-Stokes equations as the basis for solutions of all flow types. However, it uses methods that help in the solution that are based on models of turbulence,

that were developed to describe the phenomena without having to resort to a direct numerical simulation that would imply in the use of meshes so thin, that would be prohibitive in terms of computational resources. Experimental statistics helped to develop most of these turbulence models.
Thus, these turbulence models modify the original Navier-Stokes equations by introducing mean and floating quantities to produce what it calls the Reynolds averaged Navier-Stokes equations (RANS).
Among the turbulence models available, the Eddy Viscosity Models of two equations, called Model k-ε, based the present work. Separate transport equations solves the two-equation models of both velocity and length scales, hence the term designating the method.
Like the other models, it was developed over time, and its authorship was credited to Jones and Launder. Launder and Sharma (LAUNDER 1974) introduced improvements in the value of the constants used.
Moreover, the model k-ε, is one of the models of two equations most used, is robust, precise and has stability. It is currently considered the standard among the turbulence models used in industrial simulations; most commercial CFD codes embedded it in.

3.2. Study of Analyzed Flows.

In this work, the flow pattern map developed by Taitel et al. (TAITEL 1980) characterizing a specific flow rate for the combination of surface velocities of the gas and liquid phases (Figure 4) was used as a guide.

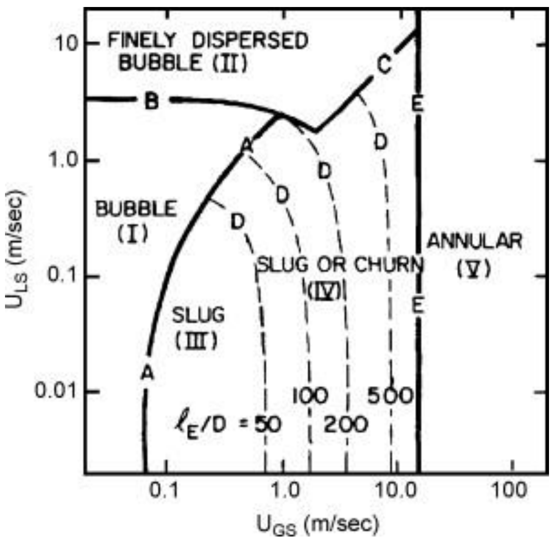


Figure 4 – Traditional two-phase flow regime map of Taitel et al, for air-water at 25 ° C and 0.1 Mpa in a 50-mm diameter tube.

Thus, the ANSYS CFX program, previously adjusted according to the geometry of the Experiment, is fed with boundary conditions corresponding to a combination of surface velocities of the liquid phase and the gas phase identified in the flow pattern map of Taitel et al. Which characterize a particular type of flow. The results provided by the program are compared to the predicted flow type according to the available visualization tools.
As a facilitator for the present work we have that the LTE Two-Phase Experiment presents the same characteristics used in the construction of the Taitel et al., Which are

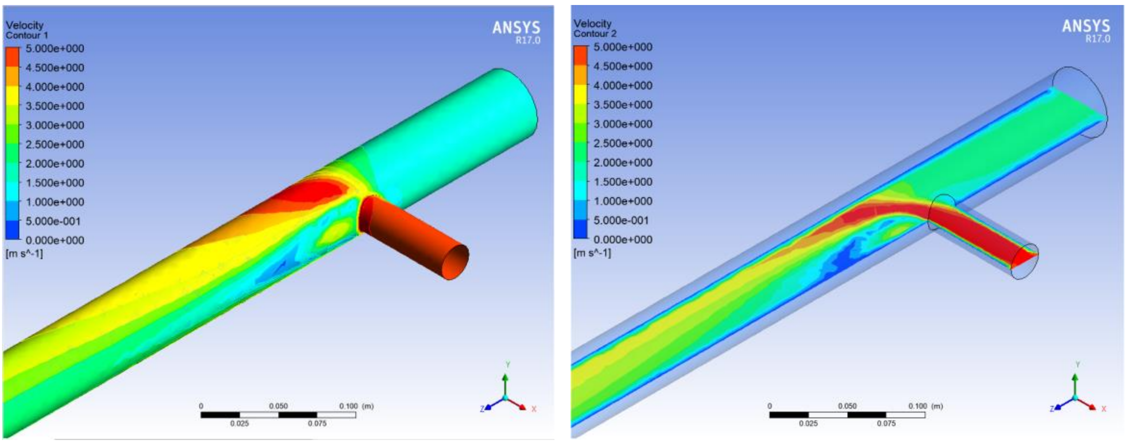
2017 International Nuclear Atlantic Conference - INAC 2017
Belo Horizonte, MG, Brazil, October 22-27, 2017
ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR – ABEN

Agência Brasileira do ISBN
ISBN 978-85-99141-07-6

the water and air fluids, the ambient temperature, the atmospheric pressure besides the tube diameters of 50 and 25 mm (Figure 4). As is known, these maps were constructed for a vertical upward flow and this orientation is used as the starting point for the evaluations with the ANSYS CFX simulations for different angles of inclination, as is the proposal of the experiment, and it will be possible to see the influence of this Angle of flow characteristics.
The proposal of this study is precisely to better understand the behavior of two-phase flows in positions other than vertical, since most of the existing references date back some years as the first studies of Runge and Wallis (RUNGE 1965), and then the work of Zukoski (ZUKOSKI 1966), Maneri and Zuber (MANERI. C. C. 1974) and Spedding and Nguyen (SPEDDING 1978).

3.3. Visualization of the Flow in Isovolumes.

Usually in the results visualization of the simulations executed using the ANSYS CFX program, a contour tool is used. The contours are based on the finite element mesh generated to solve the problem. The regions represent the variation of the values of any parameter, such as pressure, temperature, among others. This tool works by doing a 2D mapping of these regions over a predefined plane or a surface. In cases of flow in pipes or other types of ducts, we can see the variation, for example, of velocities on the inner wall or in any plane that intercepts the flow (Figure 5).



(a) (b)

Figure 5 - Example of mapping by boundary regions of the velocity of a two-phase water / air flow in the ANSYS CFX - (a) on the inner wall of the tube - (b) on a plane XZ.

However, among the many tools available in the ANSYS CFX visualization module, there is volume generation based on the finite element mesh used for program calculations and associating these volumes with a parameter, such as the contour tool. This tool also has the isovolumes option, which highlights volumes according to a desired parameter variation.

In the case of two-phase flows, the visualization of phase appearance is very interesting when, instead of using the contours of the regions, the generation of volumes based on the finite element mesh is used (Figure 6).

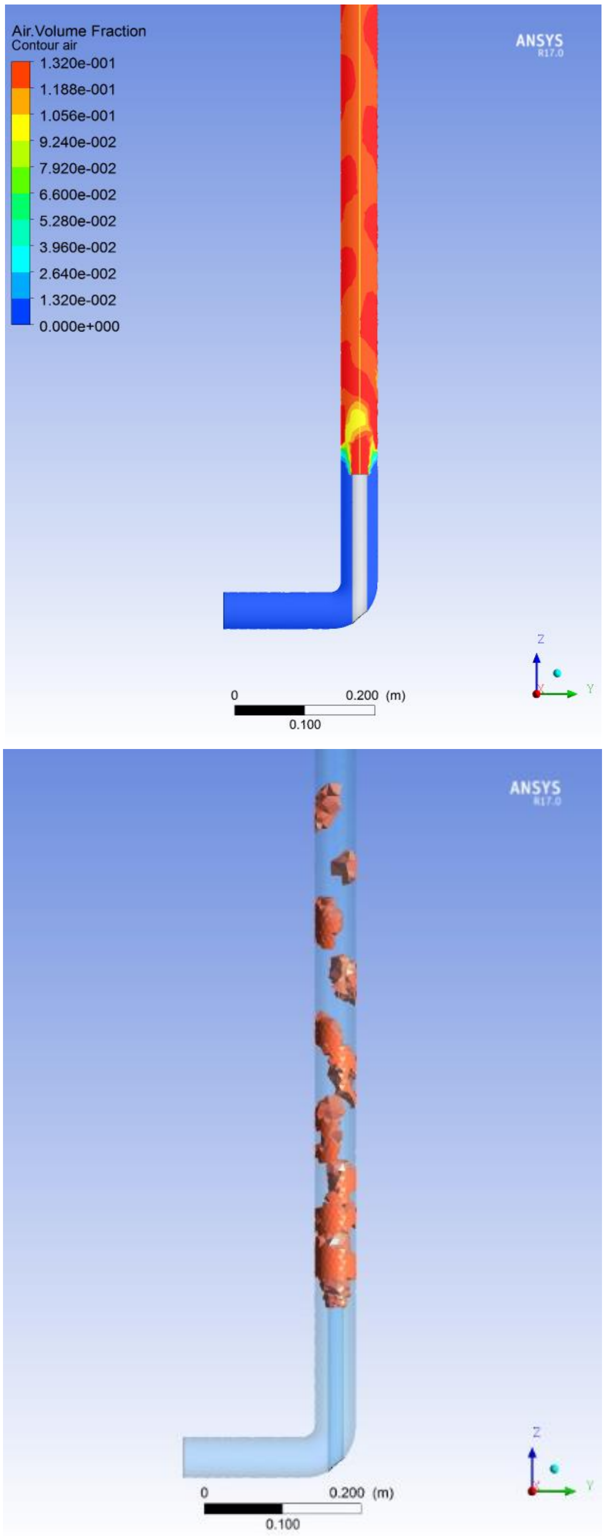
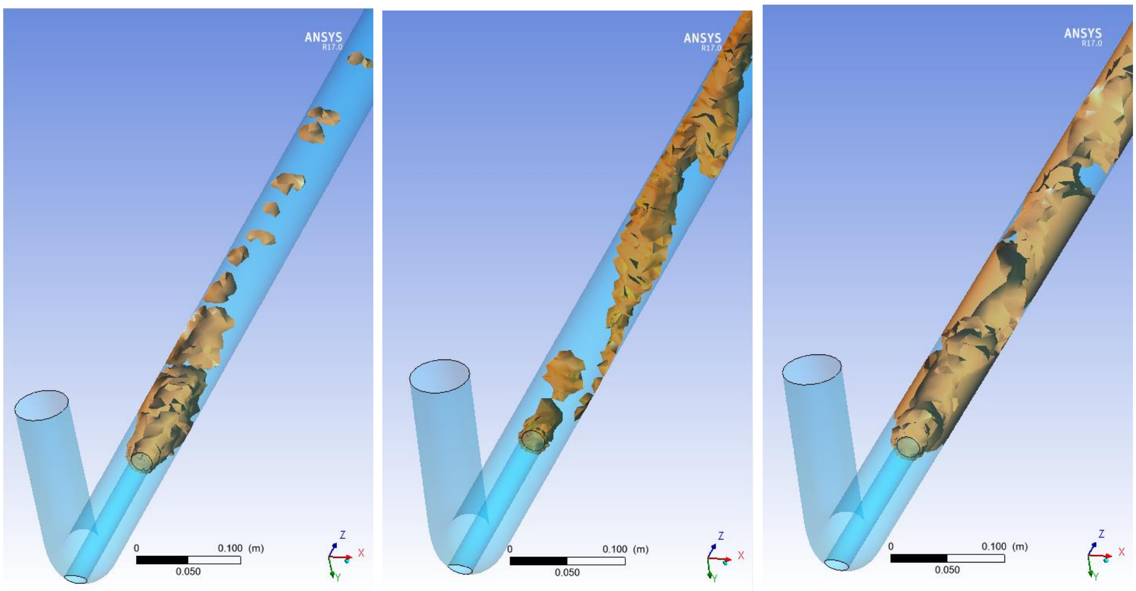


Figure 6 - Comparison of the contour (above) and isovolume (below) visualization of a given volume fraction of air in a two-phase flow of water and air.

Agência Brasileira do ISBN
ISBN 978-85-99141-07-6

The best visualization of the two-phase flow provided by the CFX isovolume option allows evaluating the flow pattern present in a given situation under analysis (Figure 7). Of course, this type of visualization requires a 3D solution, which sometimes requires a high computational effort to solve some problems. However, with due approximations it is possible to achieve satisfactory results without compromising accuracy.



(a) (b) (c)

Figure 7 – Isovolume visualization of the volumetric fraction of air in two-phase water / air flows of the experiment - (a) pattern in bubbles - (b) agitated pattern - (c) almost annular pattern

3. CONCLUSIONS

The proposed study for the future LTE Two-Phase Flow Experiment is directed to the analysis of the behavior of the flows at different angles of vertical and horizontal. This type of study is still little approached and the references already date back several years. Thus, once in operation this experiment will allow a more detailed study around the variations of the parameters of the flows submitted to the most diverse angles of inclination. The present study contributes by giving a prediction of these variations, including bringing a form of visualization, apparently little used in other works, but that adds great visual impact by the proximity of the real physical aspect of the phenomena of air and water flow together.

ACKNOWLEDGMENTS

This study certainly owes much to the encouragement coming from the two advisors who sign in with the author. It is also important to highlight the indispensable support of colleagues from the IEN in the field of computer science and the Thermo-Hydraulic

Laboratory - LTE. In addition, the precise orientation given by technicians of the Brazilian Distributor of ANSYS should be highlighted.

REFERENCES

1. ANSYS, Inc., Ansys Workbench - release 17.0. (2016).
2. FACCINI, JOSÉ LUIZ HORÁCIO. "Estudos de Escoamentos Bifásicos Gás-Líquido em Tubos Verticais e." Divisão de Reatores, IEN/CNEN, (2011).
3. LAUNDER, B. E. , SHARMA, B. I. ""Application of the Energy Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc"." *Letters in Heat and Mass Transfer*, pp. 131-138: (1974).
4. MANERI. C. C., ZUBBER, N. "An Experimental Study of Plane Bubbles Rising at Inclination." *International Journal of Multiphase Flow*, pp. 623-645: (1974).
5. RUNGE, D. E., WALLIS, G.B. "The Rise Velocity of Cylindrical Bubbles in Inclined Tubes." *Report NYO-3114-8,* (1965).
6. SPEDDING, P. L., NGUYEN, V. T. "Bubble Rise and Liquid Content in Horizontal and Inclined Tubes." *Chemical Engineering Science*, pp. 987-994: (1978).
7. TAITEL, Y., BORNEA, D., e DUCKLER, A. E., "Modelling Flow Pattern Transitions for Stead Upward Gas-Liquid Flow in Vertical Tubes*". AIChE J.*, (1980).
8. ZUKOSKI, E. E. ""Influence of Viscosity, Surface Tension, and Inclination Angle on Motion of Long Bubbles in Closed Tubes"." *Journal of Fluids Mechanics*, pp. 821-837: (1966).