8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

CFD SIMULATION OF A STRATIFIED GAS-LIQUID FLOW WITH AND WITHOUT THE EFFECT OF GRAVITY

***Kamel. Sidi-Ali[1] and Renée. Gatignol[2]**

[1]Institut Jean Le Rond D'Alembert
Université Pierre et Marie Curie & CNRS
4, Place Jussieu Case courrier 162
75252 Paris Cedex 05 France.
sidiali@lmm.jussieu.fr

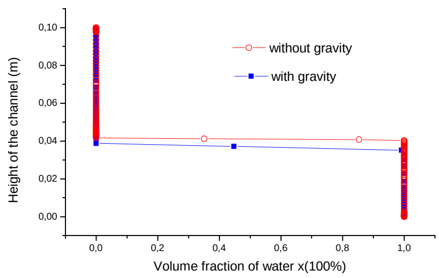
[2] Institut Jean Le Rond D'Alembert
Université Pierre et Marie Curie & CNRS
4, Place Jussieu Case courrier 162
75252 Paris Cedex 05 France.
renee.gatignol@upmc.fr

Key Words: *Two-phase flow, Stratified flow,VOF Model, Gravity, Unsteady state .*

ABSTRACT

Separated two-phase flows are generally met in the chemical, nuclear, aerospace and oil industries. Many theoretical, numerical or experimental works were developped and exploited. The CFD use many techniques to track the interface between the two phases, which make them a performant tool for obtaining good results. The aim of this work is to study a stratified two-phase flow first with the effect of gravity and second without it. The flow to study is stratified and the two phases are separated. Water is in laminar flow in the lower part of the channel, while air, considered as an ideal gas, is in turbulent flow in the upper part. The purpose of the following work is to evaluate the phase distribution, the profile of velocity phases, pressures and turbulent quantities inside a channel with a height of 0.1m and a length of 10m. Unsteady state is investigated for the case that the gas velocity is bigger than the liquid one for a time of 170 seconds.

The boundary conditions are set for non moving walls, pressure at the exit and velocity at the inlet of the channel. The VOF model is used to track the interface as the $k-\varepsilon$ model for turbulence in the industrial CFD code Fluent. For the discretisation, a PRESTO scheme has been chosen for the pressure field computations and a PISO scheme to treat the coupling between pressure and velocity. For the momentum, the turbulent kinetic energy and its dissipation rate, a second order UPWIND scheme has been used. The residuals have been fixed at order 10^{-7} for all quantities. Being an unsteady computation, a time step size of 10^{-4} s is chosen. The number of time steps is 1.700.000 for a maximum number of 1000 iterations for each time step. The obtained results are shown below :



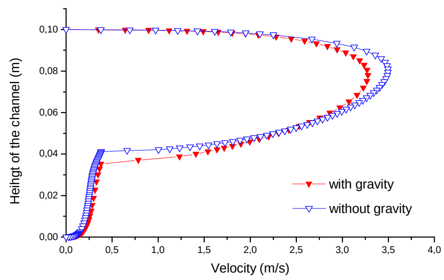
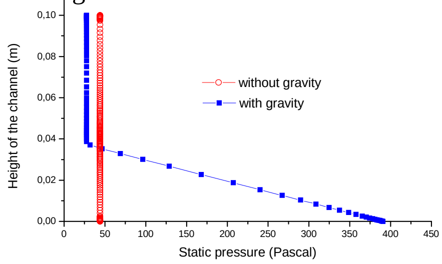


Fig. 1 Volume fraction of water

Fig. 2 Velocity profile



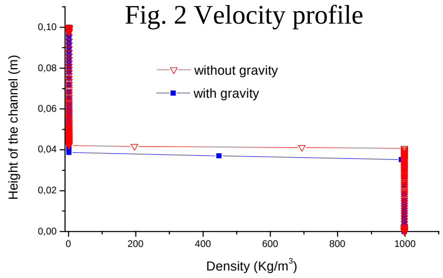
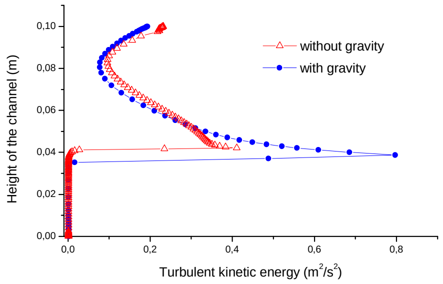


Fig. 3 Profile of the pressure

Fig. 4 Profile of density



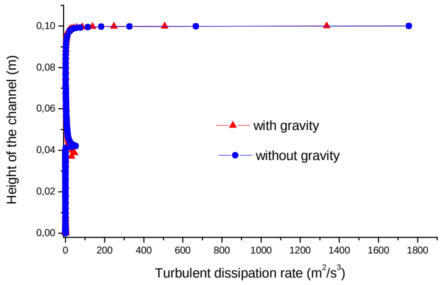


Fig. 5 Turbulent kinetic energy

Fig. 6 Turbulent dissipation rate

The results obtained show the effect of gravity on the principal quantities. For the two cases: the profiles of phases, densities, velocities, turbulent kinetic energies and their dissipation rates show a clean separation between the two phases by an interface which affects all these quantities. The case of pressure is different, in the case of flow with gravity the pressure increases in the liquid until the bottom of the channel while in the case of flow without gravity the pressure is constant in the air as in the liquid and then it is not hydostatic. The flow without gravity shows a thin interface for a height of the liquid more important than this of the flow with gravity. In this last case the interface is more larger. Concerning the velocities profile, one observes that the velocity of water in contact with the interface is the biggest one and is the same for the two flow cases. For the turbulent kinetic energy, one sees that the maximum for the two cases is at the interface, and the maximum is for the case of flow with gravity and the inverse at the upper wall. For the turbulence dissipation rate the maximum is obtained at the upper wall and the minimum at the interface.

REFERENCES

[1] Y. Taitel, A.E. Dukler, “A theoretical approach to the Lockhart–Martinelli correlation for stratified flow”, *Int. J. Multiphase Flow* 2 (1976) 591–595.

[2] Ø. Strand , “An experimental investigation of stratified two-phase flow in horizontal pipes”. PhD thesis, University of Oslo, Norway, 1993

[3] S. Ghorai, K.D.P. Nigam, “CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes”, *Chemical Engineering and Processing* **45** (2006) 55–65

[4] K. Sidi Ali, R.Gatignol, “ Horizontal stratified gas-liquid flow simulation with CFD code Fluent”, Regional ANSYS conference, Converge 2007, Paris les 18/19 oct 07.