Third International Conference on Energy, Materials, Applied Energetics and Pollution
ICEMAEP2016, October 30-31, 2016, Constantine,Algeria
M.KADJA, A.ZAATRI, Z.NEMOUCHI, R.BESSAIH, S.BENISSAAD and K. TALBI (Eds.)

ON TWO-PHASE FLOW PRESSURE DROP PREDICTION USING CFD MODELING IN HORIZONTAL PIPE SUBJECTED TO A SUDDEN CONTRACTION: VALIDATION

I. BELGACEM[1, 2], A. ARABI[1], Y. SALHI[1], E-K SI-AHMED[1,3]

[1]LMFTA Faculty of Physique, U.S.T.H.B., Algiers, Algeria, Ibtissem.belgacem@gmail.com
[2]Université Akli Mhend Oulhadj, Institut de Technologie, Bouira, Algeria
[3]GEPEA, CNRS UMR-6144 Université de Nantes France.

ABSTRACT

In this work, pressure drop through sudden contraction in mini circular pipe have been numerically investigated, using air and water as the working fluids at room temperature and near atmospheric pressure. Numerical methods include the use of two-phase computational dynamics CFD calculations based on VOF method were used to predict the pressure. The calculated pressure drop is determined by extrapolating the computed pressure profiles upstream and downstream the contraction. The tubes inside diameters are respectively 40 mm and 30 mm with a contraction ratio of 0,567. The superficial velocities, investigated in this work, ranged for the gas from 0.54 to 5.5 m/s and for the liquid from 0.011 up to 0.24 m/s. The numerical results are validated against experimental data from the literature and are found to be in good agreement.

Key Words: *Two-phase flow, sudden contraction, pressure drop, CFD, VOF, validation.*

NOMENCLATURE

Symbols :

P,p Pressure (Pa).

ΔP Pressure drop (Pa).

Greek Letters:

α_j Volume fraction of phase j.

α Void fraction.

μ Dynamic viscosity (Pa.s)

ρ Density ((kg/m^3)

σ_A Passage cross section area ratio d_2^2/d_1^2 .

$\vec{v}$ Velocity(m/s)

ε dissipation rate

λ dimensionless parameter

ψ dimensionless parameter

δ Surface tension(N/m)

Indices / Exponents :

j=l for liquid

j=g for gas

1. INTRODUCTION

The calculation of pressure drop due to gas-liquid two-phase flow through an abrupt contraction is a problem yet to be solved in engineering design. Knowledge of pressure drop for two-phase flow is important for the control and operation of industrial devices, such as chemical processes, petroleum engineering and energy manufacturing units systems. This important subject has attracted several investigations particularly for applications involving design, safety and economical operations.

Although single phase flow through singularities has been largely studied, great uncertainties exist as far as the multiphase flow is concerned. In subsequent years, some studies were conducted in order to propose new experimental data and prediction correlations. It was shown that current two-phase pressure drop correlations are applicable to a limited range of experimental conditions, and large errors occur when these correlations are applied outside the intended range(Belgacem 2015). It is worth noticing that in some cases the homogeneous model seems to be the most appropriate for modeling two-phase flow pressure drop, in a pipe subjected to a sudden contraction, while in other cases this is not necessarily true as discussed hereafter. To our knowledge, the evolution of air-water two-phase flow through a horizontal pipe subject to a sudden contraction has not been sufficiently investigated both experimentally and numerically. The present work is devoted to investigate air-water co-current two-phase flow behavior resulting from the existence of a sudden contraction in horizontal pipe. Computational fluid dynamic CFD calculation using VOF techniques are employed to generate the profile of pressure, The numerical results are validated against experimental data from the literature and are found to be in good agreement.

2. NUMERICAL PROCEDURE

2.1. Mathematical model

For the mathematical model, Eulerian based volume of fluid VOF technique for two phase modeling were employed to investigate the two phase pattern in horizontal pipe. In this model, liquid is considered to be the continuous and primary phase, and gas considered to be the dispersed and secondary phase. The fluid in both phases is Newtonian, viscous and incompressible. The uniform pressure field is assumed to be shared by both phases, the flow is considered isothermal so the energy equations are not needed.

The VOF method has the advantages of high precision, and traces the volume of fluid in the grid, not the motion of fluid particles. In the VOF model, a single set of momentum equations is shared by the fluids, and the fluid volume fraction in each computational cell is tracked throughout the domain. This model has been found to be suitable for simulating interface among two or more fluids (Ghorai et al, 2006).

The VOF method utilizes the volume fraction α, which means the fraction of the filled fluid volume in the grid to achieve the goal. The indicator function α is defined as 0 for a cell with pure gas, 1 for a cell with pure liquid, and $0 < \alpha < 1$for a cell with a mixture of gas and liquid. An interface exists in those cells that give a volume of fluid value of neither 0 nor 1. Since the indicator function is not explicitly associated with a particular front grid, an algorithm is needed to reconstruct the interface (Hirt and Nichols, 1981):

$$\alpha = \begin{cases} 0 & \text{in pur gas} \\ 0 < \alpha < 1 & \text{gas} - \text{liquid interface} \\ 1 & \text{in pur liquid} \end{cases} \quad [1]$$

2.2.Governing equations

Numerical simulation of any flow problem is based on solving the basic flow equations describing turbulence, continuity and momentum. The principal equations are solved for each phase and can be written as follow:

Continuity equation

$$\frac{\partial(\alpha\rho)}{\partial t} + \nabla.(\alpha\rho\vec{v}) = 0 \qquad [2]$$

Momentum equation

$$\frac{\partial(\alpha\rho\vec{v})}{\partial t} + \nabla.(\alpha\rho\vec{v}\vec{v}) = -\alpha\nabla p + \alpha\nabla.[\mu(\nabla\vec{v} + \nabla\vec{v}^T)] + \alpha\rho\vec{g} + \alpha\vec{F} \; [3]$$

The void fraction α is the void fraction of water or liquid phase.

Turbulent model

The Reynolds Stress Model (RSM) is a higher level, elaborate turbulence model. It is usually called a Second Order Closure. This modeling approach originates from the work by Launder 1975, in RSM, the eddy viscosity approach has been discarded and the Reynolds stress is directly computed. The model can be used to predict the turbulent anisotropic level in the flow. Given that the two-phase flows are very unstable and highly anisotropic.

$$\frac{\partial(\rho\bar{\alpha}\tilde{R}_{ij})}{\partial t} + \frac{\partial}{\partial x_k}(\rho\bar{\alpha}\widetilde{\boldsymbol{U}}_k\tilde{R}_{ij}) = -\rho\bar{\alpha}[\tilde{R}_{ij}(\nabla\widetilde{\boldsymbol{U}}_k)^T + (\nabla\widetilde{\boldsymbol{U}}_k)\tilde{R}_{ij}] + \frac{\partial}{\partial x_k}[\bar{\alpha}\mu\frac{\partial\tilde{R}_{ij}}{\partial x_k}] - \frac{\partial}{\partial x_k}[\rho\bar{\alpha}\overline{u'_i u'_j u'_k}] + \bar{\alpha}p\overline{\left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i}\right)} - \bar{\alpha}\rho\,\tilde{\varepsilon}_{ij} + \frac{2}{3}\delta_{ik}\Pi_k \qquad [4]$$

2.3. Numerical procedure

The experimental geometry with and without contraction has been modeled using an axi-symmetric 2D geometry. The simulation was performed using the commercial CFD code Fluent 6.3.26 at double precision solver mode, with an implicit scheme for all variables and a fixed time step t= 0.001 s for computation. To solve the momentum transport equation the Quick (quadratic upwind interpolation) scheme was used, for pressure the PRESTO (PREssure STaggering Option) scheme increases stability in the solution. The phase – coupled PISO (Issa (1986)) algorithm is used for the pressure –velocity coupling. RSM model has been used for turbulent two phase-flows. These schemes ensured, in general, satisfactory accuracy, stability and convergence. In addition, the steady-state solution strategy was employed. Meshing the geometry was achieved by using a software GAMBIT (2.4.6). We used the quadratic elements and the dimension of each cell is 0.004 making the number of cells equal to 842 205. The convergence criterion is decided based on the residual value of the calculated variables, namely mass, velocity components and pressure. In the present study, the numerical computation is considered converged when the residuals of the different variables are lowered by five orders of magnitude.

Inlet boundary: For both geometries the velocity of the fluids is specified at the inlet.

Outlet boundary condition: At the outlet, pressure outlet boundary is used.

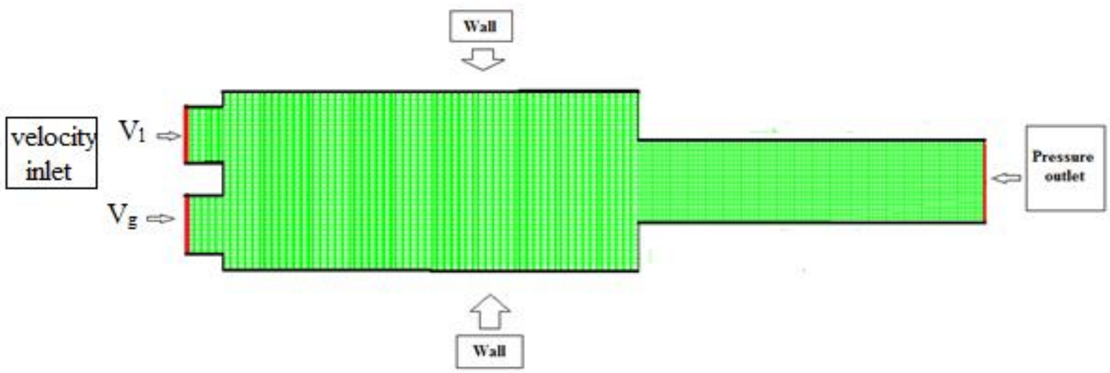
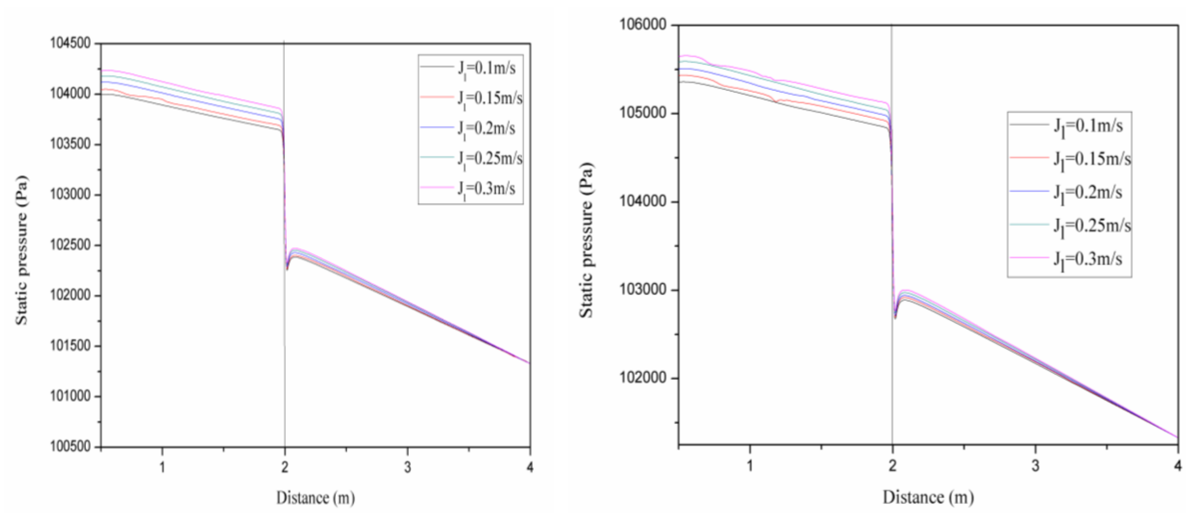


Figure 1. Computational domain and boundary conditions for contraction section.

3. RESULTS AND DESCUSSION

Figures 2a and 2b depict the predicted two-phase pressure profiles subjected to a sudden contraction for different liquid velocities keeping the gas velocity gas. Similarly to the experimental results it is observed that the pressure drop through sudden contractions increases with increasing the liquid velocities. The figure show the pressure change upstream and downstream the contraction, the static pressure decreases more rapidly than in the region of fully developed flow, It attains the (locally) smallest value at a distance of about L/D = 3.33 after the contraction section. , the results agree well with the experimental data Schmidt et Friedel et (1997) and Belgacem (2015).



(a) (b)

Figure 2. Numerical pressure profiles for two-phase air –water flow through sudden contraction

(a) Jg=3m/s (b) Jg=4m/s

Figure 3 compares the computed values of the two-phase pressure drop with the experimental data. The agreement is found to be quite good. The proposed numerical model shows acceptable accuracy against the experimental prediction. The prediction of pressure drops lies within ±25%.

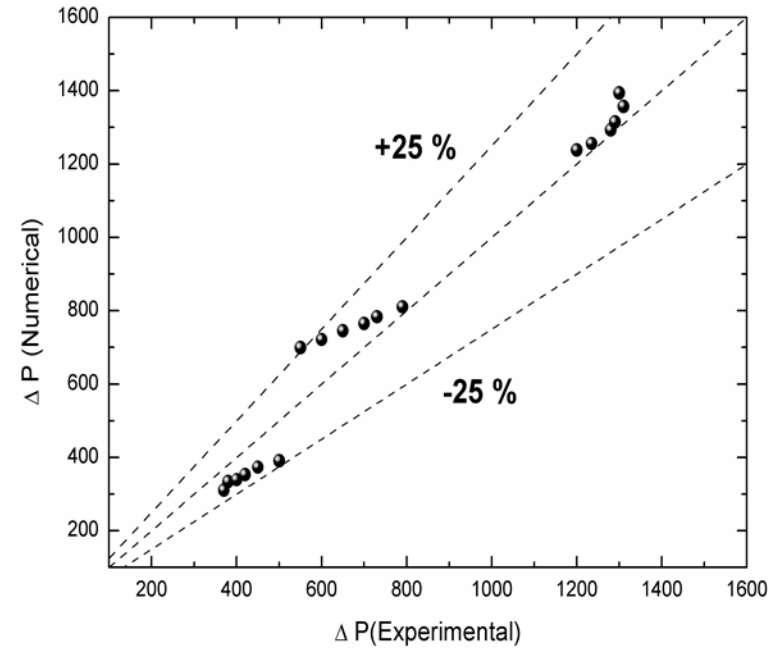


Figure 3.Comparison between numerical prediction and experimental data of Schmidt et Friedel (1997).

Simulated pressure contour are illustrated in Fig. 4, which clearly shows the pressure change upstream and downstream the contraction, the static pressure decreases more rapidly than in the region of fully developed flow, On the other hand, the pressure contours as depicted in this figure, clearly shows that the two-phase flow does not contract behind the edge of transition , the zone of recirculation is not observed, and the vena contracta phenomena is not detectable, the results agree well with the experimental data.

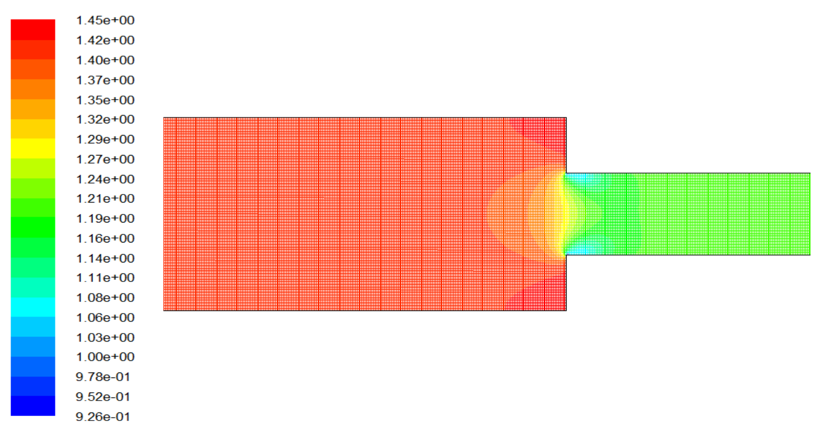


Figure 4. Snapshots of pressure contours (Jg= 5 m/s Jl=0.3m/s at t=5.5 s)

4. CONCLUSION

The numerical simulations performed with the CFD code Fluent 6.3.26 revealed that the static pressure is predicted around75%. The analysis was encouraging on showing that computational fluid dynamics model can be used for the prediction of pressure evolution and flow pattern in horizontal two-phase flow through sudden contraction. The latter can be of practical importance in the design confidence.

REFERENCES

[1] Belgacem, I. (2015). Contributions à l'étude de l'écoulement diphasique gaz-liquide co-courant en milieu confiné dans une conduite horizontale avec ou sans singularité. Thèse de Doctorat, U.S.T.H.B.

[2] Ghorai, S., Nigam ,K.D.P.(2006). CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes, Chemical engineering and processing journal, 45 (1), 55-65.

[3] Hirt, C. W., and Nichols, B. D. (1981).Volume of fluid (VOF) method for the dynamics of free boundaries.Journal of computational physics, 39(1), 201-225.

[4]Issa, R.I., 1986. Solution of implicitly discretized fluid flow equations by operator splitting.J.Comput.Phys.62, 40-65.

[5]Launder, B.E, Reece, G.J. and Rodi, W. 1975. Progress in the development of a Reynolds-Stress Turbulent closure.Journal of fluid Mechanics, 68(3), 537-566.

[6]Schmidt, J., Friedel, L., 1997. Two–phase flow pressure drop across sudden contractions in duct areas. Int.J.Multiphase flow 23.2, 283-299.