DOI: <https://doi.org/10.14311/TPFM.2022.010>

OPENFOAM SIMULATION OF TWO PHASE FLOW IN A HORIZONTAL PIPE

Z. Ghoudi ^{1,3}, F. Benkhaldoun ¹, F. Piscaglia ², N. Hajjaji ³

¹ Department of applied mathematical, Faculty of Galilee institute, University of Sorbonne Paris Nord, umr Villeteuse 7539, France

² Department of aerospace science and technology, Faculty of aerospace science and technology, University of politecnico di Milano, Via La Masa, 34 - 20156 Milano, Italy

³ Department of process engineering, University of Gabes, National engineering of Gabes, Omar Ibn El Khattab, Zrig Eddakhlania 6029, Tunisia

Abstract

The present work consists in the simulation of water-slurry two-phase flow through a horizontal pipeline using OpenFOAM, a computational fluid dynamics (CFD) simulation technique and open source software. In the aim to model the slurry comportment and evaluate the diffusion between the two phases.

Keywords: CFD, slurry pipeline, two-phase.

1 Introduction

Slurry transport through pipeline is common in many industries including foods, pharmaceuticals, chemicals and mining industries.

Slurry flows in horizontal pipes are divided into four flow regimes based on actual particle flow and particle size (1) [1];

- (1) Pseudo homogeneous and/or homogeneous mixtures with all solids in suspension;
- (2) Heterogeneous flow;
- (3) Flow with a moving bed;
- (4) Flow with a stationary bed.

Figure 1: Regimes of slurries in horizontal pipes

Computational fluid dynamics (CFD) is a useful technique to predict two-phase flow behavior of a complex system under any condition [26–28]. Currently in OpenFOAM, there are two available common approaches for computer modeling of multiphase flow. The first is the Eulerian-Eulerian method (Volume of Fluid (VOF), Eulerian and Mixture Model) and the second approach is the Eulerian-Lagrangian technique. The Euler-Euler methods consider the different phases as continua mathematically interpenetrate. The phasic volume fraction approach is proposed in this method, and the volume fractions are taken as a continuous function of time and the space whose sum is equal to one [10]. In the Eulerian-Lagrangian method, the primary phase is assumed to be continuous and the Navier–Stokes equation is solved for the primary phase, where slurry particles

are plotted throughout the domain.

Several studies were conducted using Eulerian-Eulerian CFD-based two-phase models to simulate slurry flow in circular tubes. Hossain et al (2003) have used the multiphase mixture model available in Fluent in order to study the particle deposition and suspension in a horizontal pipe flow. They have considered the diameter of particle, density of fluid and velocity of fluid as the decisive factors for the deposition of particles in the pipe flow. Stamatiou and Kawaji [11] adopted the mixture model to evaluate the particle distribution, velocity, and pressure drop of isothermal ice slurry flow. The mixture multiphase model is applied in multiphase flows wherein various phases move at different velocities. In this study, the two-phases have the same velocities. Therefore, the VOF model available in OpenFOAM [12] is used to evaluate the concentration distribution of phosphate slurry flow.

2 COMPUTATIONAL MODEL

CFD modeling involves three main steps: (1) Creating the model geometry and grid. (2) Defining the appropriate physical models and (3) Defining the boundary and operating conditions.

2.1 Geometry and Mesh

Simulations were setup in three dimensions, horizontal pipe with inner diameter D and sufficiently long length $L > 50D$ to ensure fully developed turbulent flow.

Three meshes with 242,000 (Grid 1), 484,000 (Grid 2) and 864,000 (Grid 3) cells are constructed for the 3D pipeline geometry. The studied meshes were generated using blockMesh, an OpenFOAM utility and were developed to evaluate the best compromise between results accuracy and computational time effort. The computational domain of Grid-1 is shown in Fig. 2.

Figure 2: Geometry of the computational domain-(a) and the axial and cross-section view of computational Grid-1 (237,000 cells)-(b).

2.2 Multiphase Model

The VOF (volume of fluid) model [2] is used here. The validated [7] solver TwoLiquidMixingFoam for two incompressible fluids is used to simulate the transport of water-slurry two-phase flow in the present work. A single set of momentum equations is solved in the volume of fraction model and VOF for all the phases is tracked in the entire domain. The physical properties are shared by all phases and determined by taking averaged values depending on their phasic fraction in the entire control volume [9, 8].

Mass conservation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \quad (1)$$

Momentum equation

$$\frac{\partial(\rho U)}{\partial t} + \nabla \cdot (\rho U U) - \nabla \cdot \tau_{eff} = -\nabla p + \rho g \quad (2)$$

where τ_{eff} is the viscous stress tensor, given by [3]:

$$\tau_{eff} = \mu_{eff} \frac{\nabla U + \nabla U^T}{2} - \frac{2}{3} \mu_{eff} (\nabla \cdot U) I \quad (3)$$

where

$$\begin{aligned} \mu_{eff} &= \alpha(\mu_{eff})_1 + (1 - \alpha)(\mu_{eff})_2 \\ \mu_{eff_i} &= (\mu - \mu_t)_i. \text{ Subscript "i" denotes either fluid 1 or 2.} \end{aligned}$$

Concentration transport equation

Concentration transport equation models a combination of convection and diffusion transport of the slurry fraction α . Equation in standard form can analytically be expressed as:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (U \alpha) = \nabla \cdot \left(\left(D_{AB} + \frac{\nu_t}{Sc_t} \right) \nabla \alpha \right) \quad (4)$$

where $\mathbf{U}$ represents the velocity vector, D_{AB} is the molecular diffusion coefficient, ν_t the eddy (turbulent) viscosity, and Sc_t is the turbulent Schmidt number. From the solved concentration field, a corrected density field is obtained as:

$$\rho = \alpha \rho_1 + (1 - \alpha) \rho_2 \quad (5)$$

where ρ_1 and ρ_2 are the densities for slurry and water respectively (table1), and α is the slurry fraction by volume.

The new density is then used in the transport equations of momentum and turbulence. The results include single velocity, pressure, and concentration fields.

Turbulence Model

With the solver twoLiquidMixingFoam, the k-Epsilon ($k - \epsilon$) [4] and SST k-Omega ($SSTk - \omega$) [5, 6] turbulence models were used. These models are characterized by systems with two equations and assume that the Reynolds stresses are proportional to the average velocity gradients. These models are characterized by two-equations systems and assume that Reynolds stresses are proportional to the mean velocity gradients [4, 5]. $k - \epsilon$ and $SSTk - \omega$ use an Eulerian approach and are characterized by two-phase flow models. These turbulence models present a good approximation of the turbulence kinetic energy and the turbulence dissipation rate for flows with a dominant continuous phase. The turbulent kinetic energy and the turbulent kinetic energy dissipation rate implemented in $k - \epsilon$ model are expressed in Equations (7) and (8).

$$\frac{\partial}{\partial t} \rho k = \nabla \cdot (\rho D_k \nabla k) + T_p - \rho \epsilon \quad (6)$$

$$\frac{\partial}{\partial t} \rho \epsilon = \nabla \cdot (\rho D_k \nabla \epsilon) + \frac{c_1 \epsilon}{k} (T_p + c_3 \frac{2}{3} k \nabla \cdot u) - c_2 \rho \frac{\epsilon^2}{k} \quad (7)$$

where,

k — turbulent kinetic energy;

D_k — effective diffusivity for k ;

T_p — turbulent kinetic energy production rate;

ϵ — turbulent kinetic energy dissipation rate;

D_ϵ — effective diffusivity for ϵ ;

c_1 , c_2 and c_3 — model coefficients, $c_1 = 1.44$, $c_2 = 1.92$, $c_3 = 0.00$.

2.3 Numerical details

The simulations were performed using: Euler implicit time scheme to discretize the time, Gauss linear in the gradient schemes, Gauss linear corrected in the Laplacian schemes, linear for the interpolation scheme, a corrected scheme for the surface-normal gradient scheme. The main divergence schemes used for the solver are present in Table 1.

Table 1: 3. Divergence scheme used in the simulations.

Term	TwoLiquidMixingFoam
convection	Gauss vanLeer
Diffusion	Gauss linear
momentum transport	Gauss linearUpwind grad(U)
turbulence	Gauss upwind

3 Numerical Results

Numerical simulations are performed with the OpenFOAM CFD code, version 7. Numerical results for slurry concentration (slurry fraction) distribution at a benchmark pipe of $D = 54.9mm$ of diameter have been plotted in Figure 4. The main parameters of the simulation are shown in table 2.

Table 2: Parameters of numerical simulation

Simulation parameter	Values
Pipe diameter, $D(mm)$	54.9
Pipe length, $L(m)$	3.3
Mean velocity $U(m/s)$	3
Density of slurry $\rho_1(kg/m^3)$	1600
Density of water $\rho_2(kg/m^3)$	1000
Viscosity of slurry $\mu_1(Pa)$	10^{-2}
Viscosity of water $\mu_2(Pa)$	10^{-3}
Molecular diffusion coefficient D_{AB}	10^{-6}
Turbulent Schmidt number Sc_t	1
Rheological Model	Newtonian

Figure 3: Cross sections averaged spatial concentration for $D = 54.9mm$, $L = 3.3m$, $l = 0.54m$, and $U = 3m/s$ at different meshes 242,000 (a), 484,000 (b), 864,000 (c).

Table 3: Comparison of CPU time (min) at different mesh sizes.

number of cells	CPU (min)
242,000	7.64
484,000	25.38
864,000	71.59

Regarding process times (CPU time), for mesh 3, twoLiquidMixingFoam solver requires over 72 min for one second of simulation, whereas for mesh 2 it requires about 25 min of computation time for one second of simulation. As for mesh 1, the CPU time is over 7 min for one second of simulation. Figure 3 shows that there is no effect on the concentration distribution for the different meshes used. Then, results accuracy does not affected by the mesh size. Therefore, mesh 1 was adopted for the remaining analyses, as it requires the lower computational effort the best compromise between results accuracy and computational effort.

Figure 4: Concentration distribution predicted by VOF extension model along the pipe for $D = 54.9\text{mm}$, $L = 3.3\text{m}$, $l = 0.54\text{m}$, and $U = 3\text{m/s}$.

Figure 4 represents the flow of slurry alongwith water transported by means of gravity through a benchmark pipe at turbulent regime, and capture the interface between the two phases. In fact, there is not too much mixing between water and slurry at the interface. We can conclude that the VOF model is a good model to adopt for this transport of slurry, considering these parameters of simulation.

4 Conclusion

A numerical simulation of the evolution of a two-phase turbulent flow in horizontal pipe was performed with the OpenFOAM code, version v7. The VOF model is computationally effective for predicting the concentration distribution of a two-phase flow, but it relies on a simplified assumption for the slurry phase; it considers the slurry as an homogeneous fluid. But, in reality, the slurry is a mixture of a carrier fluid and particles. Nonetheless, further research is required to evaluate the real capability of the VOF model as either an interpretative or a predictive tool.

Acknowledgment

The financial support for the present project was partly provided by LAGA, Laboratory of Analyse Geometry and Application at university of Paris 13.

References

- [1] RP. King: Introduction to practical fluid flow. *Elsevier*: 2002.
- [2] Giussani, F., Piscaglia, F., Saez-Mischlich, G., & Hèlie, J. : A three-phase VOF solver for the simulation of in-nozzle cavitation effects on liquid atomization. *Journal of Computational Physics*, vol. 406: (2020) pp. 109068.
- [3] Jacobsen, K. Å. : The dead water phenomenon. *A computational fluid dynamics study*: (2018).
- [4] Launder, B. E., & Spalding, D. B. : The numerical computation of turbulent flows, *Numerical prediction of flow, heat transfer, turbulence and combustion*: (1983) pp. 96-116 .
- [5] Menter, F. R. : Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, vol. 32: (1994) pp. 1598-1605.
- [6] Menter, F. R., Kuntz, M., & Langtry, R. : Ten years of industrial experience with the SST turbulence model. *Turbulence, heat and mass transfer*, vol. 4: (2003) pp. 625-632.
- [7] Zhang, S., Jiang, B., Law, A. W. K., & Zhao, B. : Large eddy simulations of 45 inclined dense jets. *Environmental Fluid Mechanics*, vol. 16: (2016) pp. 101-121.
- [8] Li, C., Wu, W., Liu, Y., Hu, C., & Zhou, J. : Analysis of Air–Oil Flow and Heat Transfer inside a Grooved Rotating-Disk System. *Processes*, vol. 7: (2019) pp. 632.
- [9] CFX-Solver, A. N. S. Y. S.: "Theory guide." Release 11 (2006).
- [10] Bhatti, M. M., Zeeshan, A., Ellahi, R., Bég, O. A., & Kadir, A. : Effects of coagulation on the two-phase peristaltic pumping of magnetized Prandtl biofluid through an endoscopic annular geometry containing a porous medium. *Chinese Journal of Physics*, vol. 58: (2019) pp. 222-234.
- [11] Stamatiou, E., & Kawaji, M. : Thermal and flow behavior of ice slurries in a vertical rectangular channel—Part II. Forced convective melting heat transfer. *International Journal of Heat and Mass Transfer*, vol. 48: (2005) pp. 3544-3559.
- [12] Jasak, H., Jemcov, A., & Tukovic, Z.: OpenFOAM: A C++ library for complex physics simulations. *International workshop on coupled methods in numerical dynamics*, Vol. 1000: (2007) IUC Dubrovnik Croatia.