© 2017 IJSRSET | Volume 3 | Issue 3 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

A Review on Multiphase Models and Multiphase Flow Problems at Different Flow Sections

Atul Singh, S. K. Bharti, N. V. Saxena
Mechanical Engineering Department, Millenium Institute of Technology, RGPV University Bhopal, Madhya Pradesh, India

ABSTRACT

Multiphase flow is a common phenomenon in many industrial processes, amongst them the oil and gas industry. In multiphase flow different phases are present like solid, liquid and gases. Different models are available for the study of different flow field according to the applications. Multiphase flow introduces different flow models like euler-euler model, lagrangian model , volume of fluid model etc. these models are used in different fluid flows according to present and distribution of phases. Euler –Euler model is best suitable model for pipe flow problems in implicit initial conditions. For sedimentation related problems euler- euler model is best suitable model. In this present work a brief survey is done on multiphase flow and different multiphase model for implicit conditions and somewhere found that euler – euler model is most adoptable model in most of the multiphase flow problems.
Keywords : Multiphase Flow, Multiphase Models, Euler-Euler Model, Dispersed Phase.

I. INTRODUCTION

Multiphase mixtures appear everywhere in nature, from blood flow, to the formation and motion of rain droplets, sand storms, and volcanic clouds. Also the flow of compressible multiphase mixtures is of great importance in numerous industrial and technological applications. For example, in power plants, heat exchangers, as well as in chemical and nuclear reactors. Due to the wide range of applications of the compressible multiphase flows considerable attention has been devoted to the modeling and simulation of these flows. Both the mathematical modeling and numerical computations have certain inherent difficulties. These difficulties originate from the existence of deformable and moving interfaces separating the phases or fluids. The modeling difficulties are concerned with the interaction between the fluids, which includes the transfer of mass, momentum and energy across the interfaces. While the discontinuities of the fluid properties at the interfaces are mainly responsible for difficulties in numerical methods. Therefore, the manner of treatment of the interfaces is the key point of each model.

Multiphase flow is common in many industrial processes, amongst them the oil and gas industry. Enormous quantities of oil and gas are consumed on a daily basis (CIA 2013) and even a slight enhancement in extraction efficiency will have a significant influence on revenues for companies in the oil and gas industry. Hence, finding reliable analysis tools for understanding and optimization of multiphase flows is a priority for these companies. One of these companies is Aker Solutions, where computational fluid dynamics (CFD) is used in the development of subsea equipment.

CFD was developed during the second half of the 20th century and became an established analysis tool for single-phase flow calculations during the 90ies with the appearance of commercial CFD software such as ANSYS Fluent and ANSYS CFX. The use of CFD in the area of multiphase flow is not as established. However, with the development of computer resources, making more complex analyses possible, along with the incorporation of multiphase flow models in commercial codes such as those previously mentioned, CFD is now gaining more importance also in this field. Several error sources exist for numerical simulations. Numerical approximation errors will always occur but another error source, which often is difficult to detect, is usage error. Unintended application of models, badly chosen parameters or wrongfully applied boundary conditions can lead to unphysical and inaccurate results. With the extended use of CFD simulations in engineering work it is of high importance to investigate the accuracy of commercial codes as well as understanding the choice of models. This is particularly important for multiphase flow where the complexity of both physical laws and numerical treatment makes the development of general models difficult. Not much published work has been done on comparing commercial CFD codes and as models and codes may be intended and developed for a certain multiphase area, what is accurate and applicable for one business area might be unsuitable to use for another area. Therefore, there is a need to examine and compare the models available to create a knowledge base for multiphase flow simulations using commercial software in the oil and gas industry.

Since the early 1960s, researchers have endeavoured to model multiphase flows. To this end, they have used different approaches. Eulerian-Lagrangian models track the motion of each particle and solve the dynamics of the fluid at a length scale much smaller than the particle diameter (microscopic length scale). Eulerian-Eulerian models treat the fluid and solid phases as interpenetrating continua and study their dynamics by means of averaged equations of motion. Between these two approaches, the second is often preferred because it is computationally less demanding. Owing to the enormous number of particles present in industrial plants, Eulerian Eulerian (continuum) models are not likely to be replaced by their Lagrangian-Eulerian (discrete) counterparts in the foreseeable future. Discrete modelling is nevertheless paramount. The method, to be regarded more as an effective research tool than as a practical design instrument, by providing information about the dynamics at the microscopic length scale, can significantly help develop and improve continuous macroscopic models.

II. LITERATURE SURVEY

Athulya A.Sa,Miji Cherian Rb *(*2016*)* CFD Modelling of multiphase flow through T junction is done.Have found that Phase separation is more at the outlet than at the junction also deformation of pipe is more at the upper part of junction of pipe. This is due to the higher value of stress concentration at that region. There Study is based on fluid structure interaction.

X.J. Shi, P. Zhang (2016) Solid–liquid two-phase flow and heat transfer characteristics of Tetra-n-butyl ammonium bromidein horizontal 90 elbow pipe and U-pipe were numerically investigated using Eulerian–Eulerian multiphase model. The numerical models, the numerical pressure drops and wall temperatures as well as heat transfer coefficients were compared with the experimental evidences. The numerical pressure drops and wall temperatures as well as the local heat coefficients agreed well with the experimental evidences.

Stephen Ambrose, IanS. Lowndes (2015) The rise of Taylor bubbles through expansions in vertical pipes is modelled using Computational Fluid Dynamics.The predictions from the models are compared against existing experimental work and show good agreement, both quantitatively and qualitatively. Some bubbles break into smaller parts, others remain largely intact as they pass through the expansion.The bubbles exhibit oscillatory behaviour as they traverse the expansion, resulting in pressure variations in the liquid phase.

Crowe et. al (2014) has found in his research that CFD was developed during the second half of the 20th century and became an established analysis tool for single-phase flow calculations during the 90ies with the appearance of commercial CFD software such as ANSYS Fluent and ANSYS CFX. The use of CFD in the area of multiphase flow is not as established. However, with the development of computer resources, making more complex analyses possible, along with the incorporation of multiphase flow models in commercial codes such as those previously mentioned, CFD is now gaining more importance also in this field.

Cai et al. (2014) studied the flow characteristics and stability of dense phase pneumatic conveying of pulverized coal under high pressure in an experimental test facility. The influences of operating parameters (fluidizing gas flow rate and supplementary gas flow rate) and material properties (coal category, particle size, and moisture content) on conveying characteristics were investigated with the conveying pressure up to 4 MPa. Wavelet transform and Shannon entropy analysis of the pressure drop were used to reveal the flow stability.

Jing et al. (2012) studied the resistance properties of gas-solid flows in a horizontal branch pipe. Two types of particles as glass bead and millet, with the average particle diameter 2 mm, were used. The results indicated that the pressure drop value of particles with a smaller density was reported to be smaller.

Guangbin et al. (2010) studied the characteristics of gas-solid two-phase flows in a Y-shaped pipeline. It was found that the solids flow distribution and pressure drop of the micro glass bead and millet particles had similar trend, and were significantly affected by the branch angle and gas velocity.

Eskin et al. (2007) presented a model for the poly-dispersed gas solid flows in a pneumatic pipeline. The model was validated against the experimental data found in the literature for the pressure losses. It was reported that the impact of solid's poly dispersity on the flow parameters is significant, and should be taken into account in engineering calculations.

Gu and Guo (2007) studied the simulation of a 3D wave-like slug flow pneumatic conveying in a horizontal pipe with the kinetic theory. The characteristics of flow, such as pressure drop, air velocity distribution, slug length, settled layer thickness, and the detailed changing characteristics of slug length and settled layer thickness with the air velocity were obtained. The results indicated that the kinetic theory can represent the physical characteristics of the non suspension dense phase wave-like slug-flow in pneumatic conveying.

Heinl and Bohnet (2005) carried out a CFD study of pneumatic conveying in a horizontal pipe including the particle wall adhesion. The dispersed phase was modeled with the Lagrangian approach, and the continuous phase was resolved with the Realizable model. The influence of different wall treatments on the pressure drop and particle-wall adhesion was investigated.

Murrone and Guillard [2005] derive the five-equation model. In his research, he will apply this procedure to derive the six-equation model accompanied by heat and mass transfer from the full seven-equation model with heat and mass transfer. He found that the reduced models can be achieved by applying a reduction procedure in the presence of stiff relaxation terms.

Zhu et al. (2004) studied the 3D CFD simulations of pneumatic conveying of granular solids in horizontal and inclined pipes. The particle-wall collisions were found to have a very significant effect on the solid distribution over the cross-section of the conveying tube for large particles.

Doneal et al. [2004] used Arbitrary Lagrangian-Eulerian (ALE) methods in his investigation This method allows for both types of strategies that are used by Lagrangian or Eulerian methods. The mesh may be moved in a Lagrangian fashion, or be held fixed in an Eulerian manner, or be moved in other ways to give a continuous rezoning capability. This makes the method flexible and one can collect the benefits of both Lagrangian and Eulerian methods. But the difficulty of this method lies in the decision of which type of grid or strategy is used through the domain of computation and during the flow process.

Wachem and Almstedt (2003) have found several error sources exist for numerical simulations. Numerical approximation errors will always occur but another error source, which often is difficult to detect, is usage error. Unintended application of models, badly chosen parameters or wrongfully applied boundary conditions can lead to unphysical and inaccurate results. With the extended use of CFD simulations in engineering work it is of high importance to investigate the accuracy of commercial codes as well as understanding the choice of models. This is particularly important for multiphase flow where the complexity of both physical laws and numerical treatment makes the development of general models difficult.

III. CONCLUSION

It is evident that the Euler-Euler approach is best suited for the T-junction application. Using an Euler-Euler model the flow redistribution phenomenon was captured in all cases, with the exception of an erroneous prediction of water separation in the pipe.Velocity of water gives immense effects on mid section of pipe in by using Eulerian approach in multiphase model. In VOF (volume of fluid) model water flows like a constant velocity in defined inlet velocity conditions in solver setup.

IV. FUTURE SCOPE

As concluded, multiphase flow simulations involve a large number of parameters and models and due to the limited timeframe many of these parameters have not been investigated in this study. To propose models/settings resulting in better agreement with the experimental data, especially regarding local profiles, a further study could be made on the polydispersed modeling as well as on the effect of adding other forces of interest.

V. REFERENCES

[1] Athulya A.Sa, Miji Cherian Rb , CFD Modelling of Multiphase Flow through T International Conference on Emerging Trends in Engineering,

Science and Technology, Procedia Technology 24 (2016) 325 – 331
[2] X.J. Shi, P. Zhang, Two-phase flow and heat transfer characteristics of tetra-n-butyl ammonium bromide clathrate hydrate slurry in horizontal 90 elbow pipe and U-pipe, International Journal of Heat and Mass Transfer 97 (2016) 364–378
[3] Stephen Ambrose, Ian S. Lowndes, David M. Hargreaves,Barry Azzopardi,Numerical modelling of the rise of Taylor bubbles through a change in pipe diameter, Computers and Fluids 148 (2015) 10–25 Crowe, C.T., Schwarzkopf, J.D., Sommerfeld and M., Tsuji, Y. (2012): CRC Press, Taylor & Francis Group, Boca Raton, Florida.
[4] Jing R., Ren F., and Wang X., 2014. The resistance properties of gas-solid flow for horizontal branch pipe. Advanced Materials Research 361-363, 887–890.
[5] Cai L., Pan Z., Xiaoping C., and Changsui Z., 2014. Flow characteristics and stability of dense-phase pneumatic conveying of pulverized coal under high pressure. Experimental Thermal and Fluid Science 41, 149–157.
[6] Wang Y., Williams K.C., Jones M.G., and Chen B., 2010. CFD simulations of gas-solid flow in dense phase bypass pneumatic conveying using the Euler-Euler model. Applied Mechanics and Materials 26-28, 1190–1194.
[7] Gu Z. and Guo L., 2007. Simulation of horizontal slug-flow pneumatic conveying with kinetic theory. Frontiers of Energy and Power Engineering in China 1, 336–340.
[8] Eskin D., Leonenko Y., and Vinogradov O., 2007. An engineering model of dilute polydisperse pneumatic conveying. Chemical Engineering and Processing 46, 247–256.
[9] A. Murrone and H. Guillard. A five-equation reduced model for compressible two phase flow problems. *J. Comput. Phys.*, 202(2):664–698, 2005.
[10] Heinl E. and Bohnet M., 2005. Calculation of particle-wall adhesion in horizontal gas-solids flow using CFD. Powder Technology 159, 95–104.
[11] J. Donea1, A. Huerta, J.-P. Ponthot, and A. Rodr´ıguez-Ferran. Arbitrary Lagrangian-Eulerian Methods. In E. Stein, R. de Borst, and T. J. Hughes, editors, *Encyclopedia of Computational Mechanics*, chapter 14. John Wiley & Sons, 2004.
[12] Zhu K., Wong C.K., Rao S.M., and Wang C.H., 2004. Pneumatic conveying of granular solids in horizontal and inclined pipes. AIChE Journal 50, 1729–1745.
[13] Van Wachem, B.G.M. and Almstedt, A.E. (2003): Methods for Multiphase Computational Fluid Dynamics, *Chemical Engineering Journal*, Vol. 96, No. 1, December 2003, pp. 81-98.
[14] A. Kapila, R. Menikoff, J. Bdzil, S. Son, and D. Stewart. Two-phase modelling of DDT in granular materials: Reduced equations. *Phys. Fluid*, 13:3002–3024, 2001.
[15] Ferreira M.C., Freire J.T., and Massarani G., 2000. Homogeneous hydraulic and pneumatic conveying of solid particles. Powder Technology 108, 46–54.
[16] Levy A. and Mason D.J., 2000. Two-layer model for non-suspension gas-solids flow in pipes. Powder Technology 112, 256–262.
[17] R. Saurel and R. Abgrall. A multiphase Godunov method for compressible multifluid and multiphase flows. *J. Comput. Phys.*, 150(2):425–467, 1999.
[18] Huber N. and Sommerfeld M., 1998. Modelling and numerical calculation of dilute-phase pneumatic conveying in pipe systems. Powder Technology 99, 90–101.
[19] J. P. Cocchi and R. Saurel. A Riemann problem based method for the resolution of compressible multimaterial flows. *J. Comput. Phys.*, 137:265–298, 1997. Hong J. and Tomita Y., 1995. Analysis of high density gas-solids stratified pipe flow. International Journal of Multiphase Flow 21, 649–665.
[20] Oesterle B. and Petitjean A., 1993. Simulation of particle to particle interactions in gas-solid flows. International Journal of Multiphase Flow 19, 199–211.
[21] S. Osher and J. Sethian. Fronts propagating with curvature dependent speed: Algorithms based on the Hamilton-Jacobi formulation. *J. Comput. Phys.*, 79:12–49, 1988.
[22] H. Stewart and B. Wendroff. Two-phase flow: Models and methods. *J. Comput. Phys.*, 56:363–409, 1984.
[23] C. Hirt and B. Nichols. Volume of Fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.*, 39:201–225, 1981.