CFD-DEM Modelling on Complex Particle-Fluid Flow with Fluid-Structure Interaction

Yi He*, Andrew Bayly*, Ali Hassanpour*, Michael Fairweather*, Frans Muller*

* School of Chemical and Process Engineering, University of Leeds, Leeds, UK
e-mail: Y.he1@leeds.ac.uk

ABSTRACT

Large free structure are commonly encountered in particle-fluid flow, such as fluidization, chemical reaction and mineral processing [1-3]. It can have different forms, such as internal structure or large sized particles. A distinctive feature of the system is that the motion of the large structure is not predefined, whose motion is affected by fluid flow and collision with bounding walls. In addition, the presence of large object plays an important role in the behaviour of particulate phase, affecting mixing, segregation, heat and mass transfer rate, etc. A sound description of the dynamics of large object is thus necessary for design, operation and optimization in engineering practice. However, it is challenging to model such kind of system since it involves interactions at different scales.

In this study, a new numerical model was presented under the CFD-DEM framework to tackle the fluid-structure interaction involved in particle-fluid flow. The presence of large structure requires explicit consideration of interactions among fluid flow, large immersed structure and small particles. In the present model, the large structure is treated as discrete element. However, the calculation of fluid-solid interaction differs from small particles. For large structure, its motion is accounted for by means of body-fitted dynamic meshing, in which forces and torques due to fluid flow are calculated by integrating the pressure and viscous stress acting on the surface of the structure. Collisions with small particles and bounding walls are modelled by a soft-sphere DEM model. For small solids, conventional CFD-DEM coupling was utilized to calculate the forces caused by fluid flow. The model has been implemented on the platform of commercial CFD software, ANSYS/Fluent, due to its capabilities in handling complex geometries and dynamic re-meshing. The model was fully implemented through its user defined functions (UDFs). The CFD-DEM coupling is based on the Eulerian model in Fluent by treating the secondary phase as a ghost phase without considering phase interaction. The developed CFD-DEM coupling has been validated against literature data in typical gas-solid fluidization processes [4], with good agreement obtained. The capability of the present model in tackling the fluid-structure interaction in particle-fluid flow is demonstrated by gas fluidization with an immersed tube which can be free-moving. The developed model is fully implemented on the Fluent platform such that it can be applied to a wide range of industrial application with complex and/or free-moving boundary conditions which beyond the capability of conventional CFD-DEM coupling.

REFERENCES

[1] Zhao, Y.Z., et al., Particle-Scale Simulation of the Flow and Heat Transfer Behaviors in Fluidized Bed with Immersed Tube. Aiche Journal, 2009. 55(12): p. 3109-3124.

[2] Hou, Q.F., Z.Y. Zhou, and A.B. Yu, Computational study of heat transfer in a bubbling fluidized bed with a horizontal tube. Aiche Journal, 2012. 58(5): p. 1422-1434.

[3] Di Maio, F.P., A. Di Renzo, and D. Trevisan, Comparison of heat transfer models in DEM-CFD simulations of fluidized beds with an immersed probe. Powder Technology, 2009. 193(3): p. 257-265.

[4] Muller, C.R., et al., Validation of a discrete element model using magnetic resonance measurements. Particuology, 2009. 7(4): p. 297-306.