089 Investigation of the flow patterns of gas-solid granular flow in a horizontal pipe by FLUENT-EDEM

Yong LU, Donald H. Glass, William J. Easson, Martin Crapper
(Y.Lu@ed.ac.uk, Don.Glass@ed.ac.uk, Bill.Easson@ed.ac.uk, Martin.Crapper@ed.ac.uk)
School of Engineering and Electronics, the University of Edinburgh
King's Buildings, Mayfield Road, Edinburgh, EH9 3JL, UK

Abstract:

Pneumatic conveying systems are applied widely in modern industries processing solid materials. In recognition of this fact, features of gas-solid pipe flow affecting transportation have been extensively studied. These include the fluid turbulence, the lift force due to the shear flow, the Magnus effect on rotating particles, particle interactions, particle-wall collisions, particle size and shape, pipe roughness and gravitational settling. Two numerical methods, the Eulerian model and Discrete Phase Model, have been developed to describe gas-solid effects in the commercial FLUENT Computational Fluid Dynamics (CFD) package. The key difference between these models is as following: The Eulerian model treats the solid phase as a second phase interpenetrating, and interacting with, the gas phase; The Discrete Phase Model treats the solid phase by solving the equations of motion of each particle (including gas/particle momentum transfer) in a Lagrangian frame of reference for each time step, re-solving the equations of motion of the gas after each interval. This gives the trajectories of all the particles. Since neither model in FLUENT accounts for inter-particle collisions and particles under the influence of gravity migrate towards the base of the pipe, giving a region of relatively high solids concentration where both particle-particle and particle-wall interactions are significant, their numerical results are inaccurate, particularly in the region close to the bottom of a long horizontal straight pipe. Sommerfeld (2003) published the 2D numerical results of DPM simulation of solid transport in a 35mm height, 6m long horizontal smooth channel in which 110µm particles at a solid mass loading ratio of 10% were transported with an air velocity of 18m/s. This simulation showed particle trajectories indicating that the most particles deposited on the channel bottom and afterwards rolled or bounced with small amplitude along it at distances greater than 3m downstream of the injection point.

The present article introduces the numerical results of CFD simulation carried out by FLUENT combined with EDEM. The fluid inlet conditions are based on the experimental data obtained with Laser Doppler Anemometry (LDA) in apparatus where 38-100µm solid particles were transported at a solid mass loading ratio of 1.6% in an air stream with velocity 12m/s. Fig. 1 shows the typical contours of particle velocity and particle number on a pipe cross-section. Here the EDEM module was developed by DEM Solutions Limited of Edinburgh, UK, and it is the first commercial Discrete Element Method software to be coupled with FLUENT for granular flow researches. In the computational work, the gas phase flow equations were solved by Fluent and every particle in the solid phase was tracked using EDEM, which accounted for inter-particle interactions and particle-wall collisions. A coupling module described the interchange of energy and momentum between the phases. The boundary conditions of the computational model were matched with the actual conditions in the experimental rig. The fluid patterns of the gas-solid pipe described by particle velocity and particle number were compared between experimental and computational results.

Keywords: gas-solid flow, computational fluid dynamics, particle velocity, particle number, flow pattern.

References

M. Sommerfeld. Analysis of collision effects for turbulent gas-particle flow in a horizontal channel: Part I. Particle transport. International Journal of Multiphase Flow, 29(2003) 675-699.

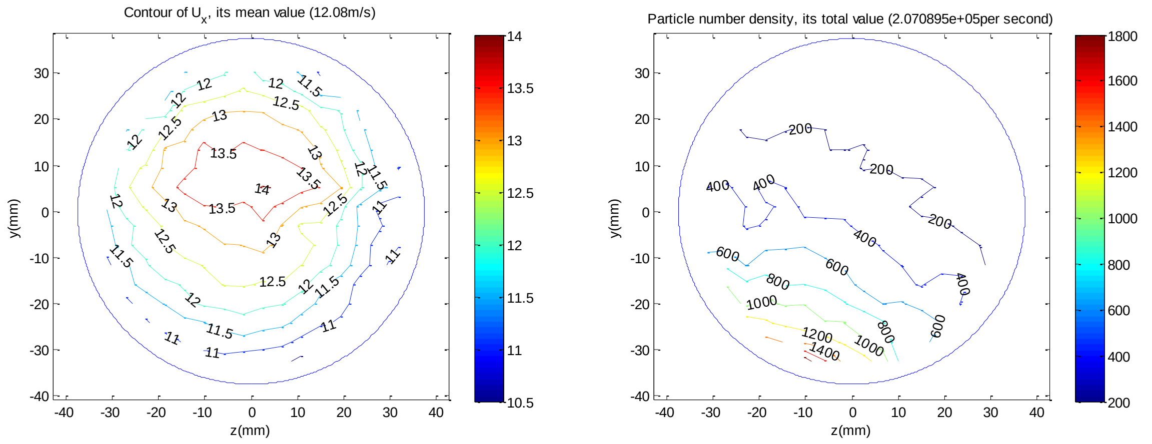

Fig. 1 the contours of particle velocity and particle number (x=300mm, away from the elbow inlet).