Title
Numerical Simulation of Gas-Solid Flows in Fluidized Bed

Members of project and affiliation
Name: Ronith Stanly
Purpose: Internship *160697*
Address: Novosibirsk State University, 2 Pirogova Str., Novosibirsk, 630090, Russia

Name: Georgy Shoev
Purpose: Scientific adviser
Address: Novosibirsk State University, 2 Pirogova Str., Novosibirsk, 630090, Russia
Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, 4/1 Institutskaya str., Novosibirsk, 630090, Russia

Scientific Content

1. Problem Formulation

This work evaluates the effectiveness of Two-Fluid Model (TFM) and CFD-Discrete Element Method (CFD-DEM) to simulate gas flows with dense particles by using a simplified Fluidized Bed as a test case.This document includes: (i)TFM simulations performed using two drag models, namely Gidaspow and Syamlal-O'Brien, using Ansys Fluent 18.0, corresponding to the experiments performed by Kuipers (1992) (ii)TFM and CFD-DEM simulations performed using Gidaspow's drag model in OpenFOAM corresponding to the experiments performed by Goldschmidt (2004).

2. The current state of the problem

The TFM model considers all the included phases as inter-penetrating continuum media [2, 3]. The governing equations are the continuity equation for gas and solid phases, momentum conservation equation for gas and solid phases and the equation for granular temperature. Whereas, for the CFD-DEM approach, the particles are individually tracked and the collisions are resolved by considering the collisions as a combination of spring-dashpot-friction slider systems using the Hertzian contact law.

3. Detailed Description

Two-Fluid Model

Simulations have been performed based on the experiments conducted by Kuipers in [4] with the dimensions of the domain as shown in Fig. 1. About 1500 seconds of computational time was required to simulate 1 second of real flow field using 24 cores and 32GB of memory. Computations were performed using Ansys Fluent 18.0 available in the supercomputing cluster at the Information and Computing Center of Novosibirsk State University which offered good performance for simulating these cases.

At the initial time moment, glass beads having a diameter of 500 μm and density of 2660 kg/m 3 are filled in the shallow bed up to a height of 0.5 m, as shown in Fig. 1. The bed is fluidized with a minimum fluidization velocity of 0.25 m/sec across the entire bottom surface except the center portion where a jet of 10 m/sec is issued through a rectangular hole of dimension 0.015 m × 0.015 m for producing the bubbling effect. Initial solids volume fraction is set at 0.6, which results in an initial inventory of 6.82 kg.
Simulations were performed in Ansys Fluent 18.0 using the pressure-based, transient solver with phase-coupled SIMPLE algorithm. The flow was considered to be laminar. Interfacial momentum exchange was assumed to consist of drag and buoyancy forces, while neglecting others like Saffman and Magnus forces. First-order discretization was used for all variables. Gradients were

calculated using the least squares cell-based scheme. A time step of 2.510 −4 with 60 iterations per time-step was used to ensure convergence. Schaeffer's closure [5] for frictional viscosity along with Syamlal's correlation [6] for frictional pressure has been used. Gidaspow and Syamlal-O'Brien drag models were used for studying bubble evolution; while Gidaspow model alone was used for the time-averaged studies.

For the gas-phase, no-slip boundary condition was imparted at the walls; whereas for the solid-phase, a specular reflection coefficient of 0.5 was given, as used in [7]. The interaction/collisions between phases was also computed with a coefficient of restitution of 0.5. For the computations of bubble evolution, a frictional packing limit of 0.6 and packing limit of 0.9 was used; whereas for the computations of time-averaged volume fraction distribution, 0.63 was used for both the parameters.

CFD-DEM

The CFD-DEM simulations were performed for another case (Goldschmidt [8]) using OpenFOAM. About 1000 seconds of computational time was required to simulate 1 second of real flow field using 24 cores and 32GB of memory. The TFM simulations corresponding to the same problem took about 1600 seconds of computational time to simulate 1 second of real flow field. Computations were performed using OpenFOAM installed at Computing Center of Novosibirsk State University. A fluid timestep of 210 −4 and a particle timestep of 210 −5 was used. And an artificially low spring constant of 2100 N/m was used for quick computation. This is the reason for the CFD-DEM simulations to take much lesser time than the TFM simulations.

4. Results

Bubble Evolution

Using the two drag models and the two mesh refinements, simulations were performed for time moments from 0 s to 0.2 s, while obtaining the results every 0.02 s. This gave instantaneous bubble evolution, as shown in Fig. 2, by observing the flow-field of volume fraction of solid phase. The contours of the bubbles were identified by specifying the boundary with # s ≤ 0.15; as done by Kuipers in [4].

Time-Averaged Results

Simulations were performed using Gidaspow drag model with the mesh as used in [7] for time-moments from 0 s to 30 s; while sampling time-averaged data during the last ten seconds. The contours of the time-averaged volume fraction of the present TFM simulation have been compared with the TFM simulation performed using MFIX (MFIX-TFM) and the Euler-Lagrange or CFD-Discrete Element Method (CFD-DEM) simulations performed using MFIX (MFIX-PIC) and Barracuda (CPFD), as seen in [7] and is shown in Fig. 4.

For more specific comparison, the time-averaged volume fraction distributions were obtained. To make comparisons more convenient, we treat x=0 at the middle of the central jet and volume fraction distributions were obtained along three vertical lines at x=0.4 cm,3.4 cm and 6.4 cm from the central jet at the mid-plane along the bed thickness as shown in Fig. 5 and compared with the results found in literature [7] as shown in Fig. 6.

It is clear from Fig. 5 and Fig. 6 that the present Euler-Euler computations differ from the experimental data at those locations where more solid particles are present. The same behavior can be observed for the TFM results obtained using MFIX. The results of the present computations as well as those of MFIX-TFM, despite using the same mathematical model, are not in good agreement with each other at all points. This hints at the need for extensive validations of the model across varied test cases and solvers, which can be carried out in the future.

The results of the TFM and CFD-DEM simulations done for the Goldschmidt's case are shown in Fig 7. It can be clearly seen that even though neither CFD-DEM nor TFM was able to reasonably predict the behaviour of the bed expansion, the CFD-DEM simulation was comparitively better as it was atleast able to predict the fluctuations in the ensemble-averaged particle height. Using a different drag model, particularly the ones derived from Lattice Boltzmann simulations, would have given better results; this will be the basis for further studies.

5.Illustrations

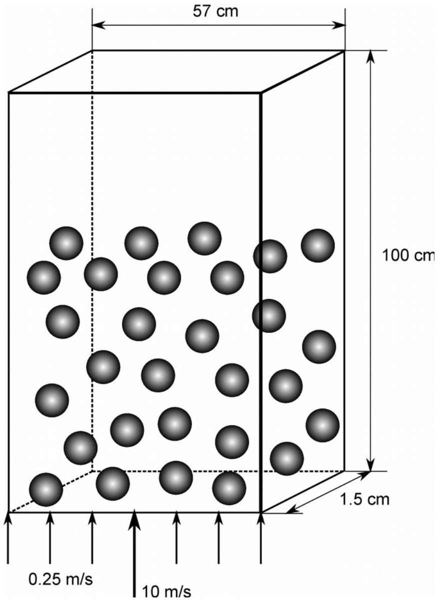

FIGURE 1. The fluidized bed test setup by Kuipers [4]

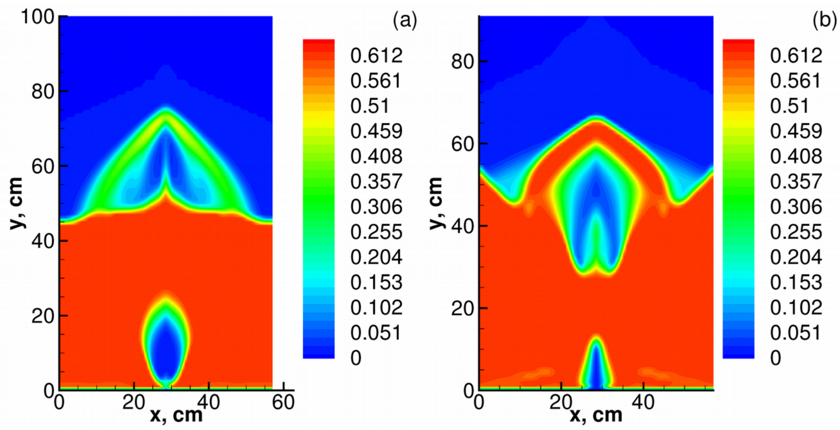

FIGURE 2. Bubble evolution at 1 sec (a) and 9 sec (b).

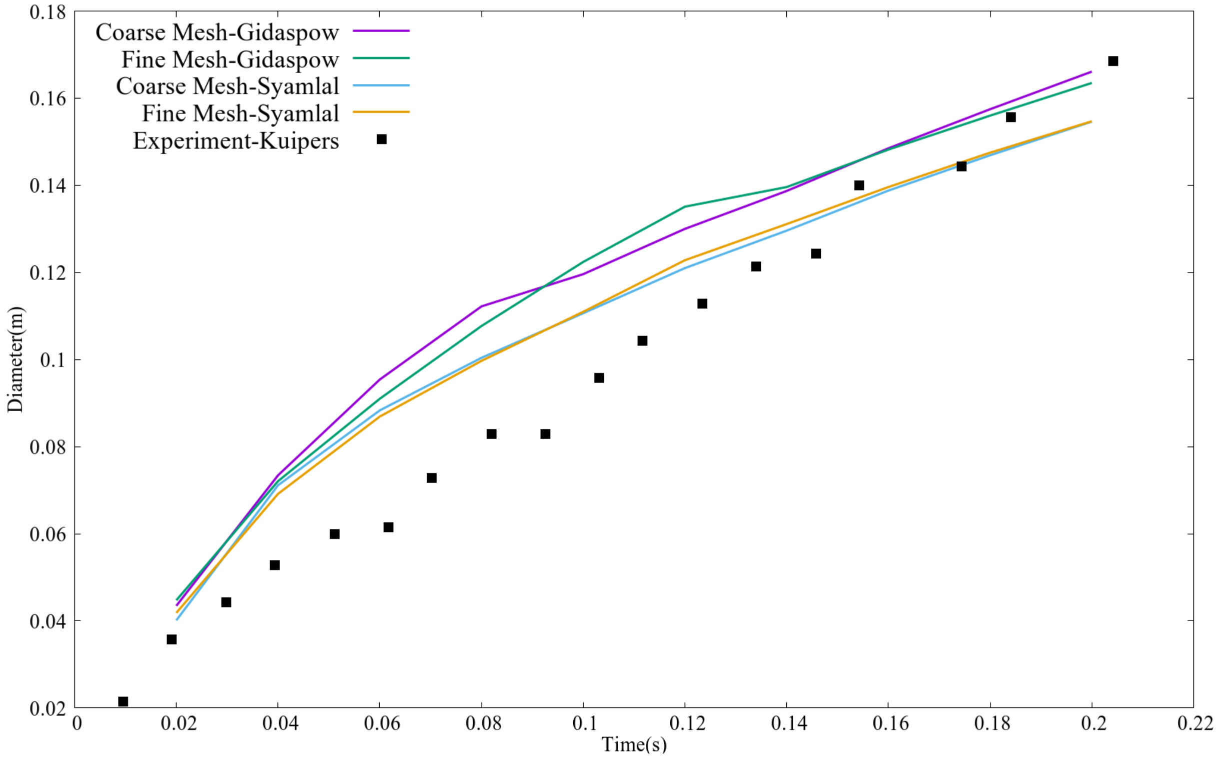

FIGURE 3. Comparison of Experimental and Computational bubble diameters

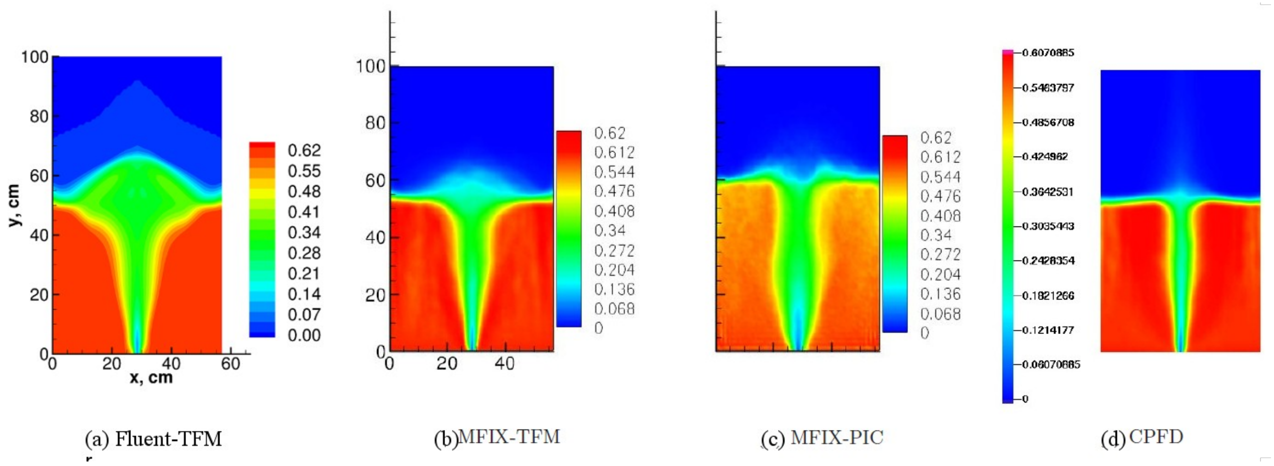

FIGURE 4. Time-averaged solid volume fraction of the present simulation (a) compared with literature data [7].

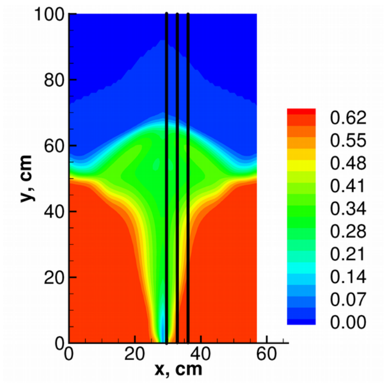

FIGURE 5. Vertical lines at x=0.4cm,3.4cm and 6.4cm from the central jet along which time-averaged data were obtained.

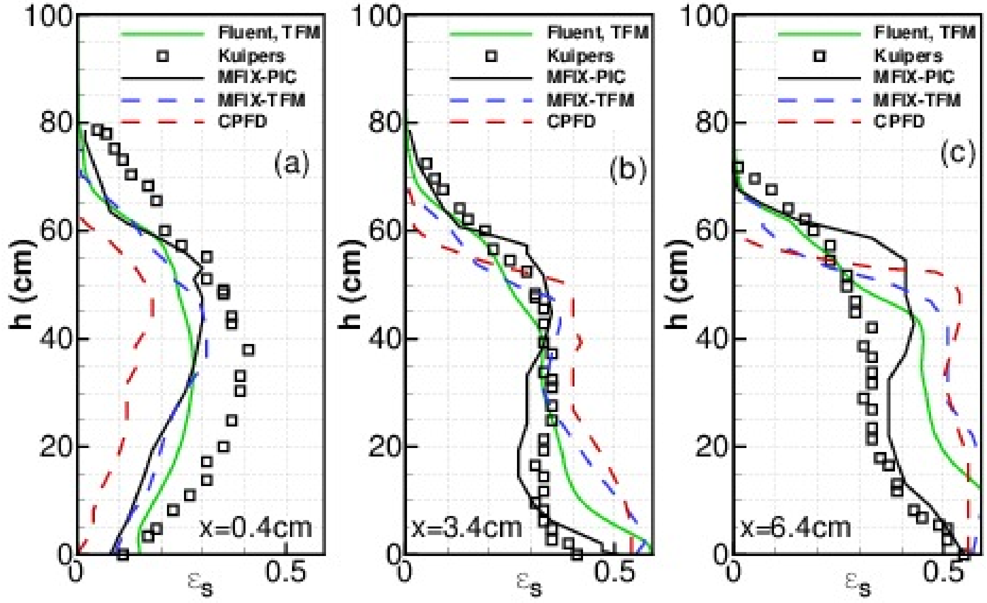

FIGURE 6. Average volume fraction (20–30sec) distribution of solid particles along 3 vertical lines at x=0.4cm(a), x=3.4cm(b), x=6.4cm(c) from the central jet.

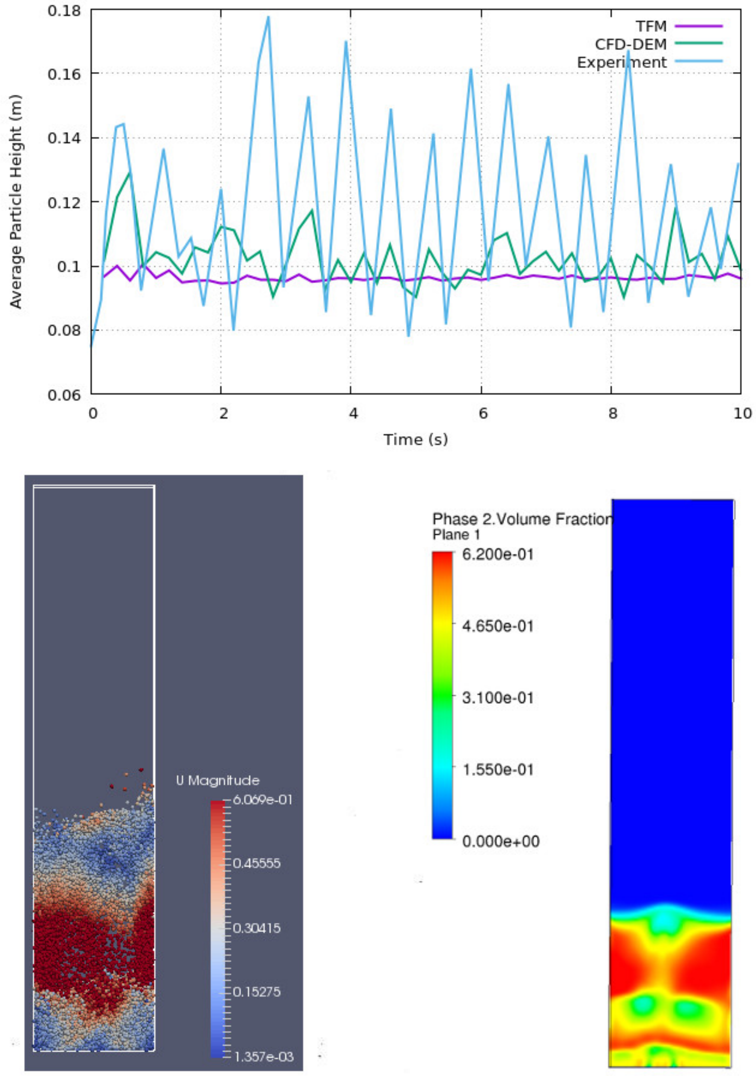

FIGURE 7. Temporal variation of ensemble averaged particle height

Performance of Cluster

For the TFM simulations, about 1500 seconds of computational time was required to simulate 1 second of real flow field using 24 cores and 32GB of memory. Computations were performed using Ansys Fluent 18.0 available in the supercomputing cluster at the Information and Computing Center of Novosibirsk State University which offered good performance for simulating these cases.
The CFD-DEM simulations were using OpenFOAM. About 1000 seconds of computational time was required to simulate 1 second of real flow field using 24 cores and 32GB of memory. The TFM simulations corresponding to the same problem took about 1600 seconds of computational time to simulate 1 second of real flow field.

List of Publications related to this work (Till now)

1) "Numerical Simulation of Gas-Solid Flows in Fluidized Bed with TFM Model", Ronith Stanly, Georgy Shoev and Kokhanchik A.A., AIP Conference Proceedings, 1893, p. 030040 (2017)
DOI: https://doi.org/10.1063/1.5007498

References

[1]R. Stanly, G. Shoev, and A. A. Kokhanchik, AIP 1893, p. 030040 (2017), DOI: https://doi.org/10.1063/1.5007498
[2]T. O. S. Pannala, M. Syamlal, Computational Gas solids Flows and Reacting Systems: Theory, Methods and Practice (IGI Global, USA, 2011), pp. 01–65.
[3]H. Enwald, E. Peirano, and A.-E. Almstedt, International Journal of Multiphase Flow 22, 21–66 (1996).
[4]J. Kuipers, "A two-fluid micro balance model of fluidized beds," Ph.D. thesis, University of Twente 1990.
[5]D. Schaeffer, J. Differ. Equ. 66, 19–50 (1987).
[6]T. O. M. Syamlal, W. Rogers, Mfix Documentation: Theory Guide, National Energy Technology Laboratory,
Department of Energy, Technical Note DOE/METC-95/1013 and NTIS/DE95000031 (8 July 1993).
[7]R.Garg and J.F.Dietiker, Documentation of open-source mfix-pic software for gas-solids flows, https://mfix.netl.doe.gov/documentation/mfix-pic-doc.pdf.
[8]M. Goldschmidt, R. Beetstra, and J. Kuipers, Powder Technology 142, 23 – 47 (2004).