中国石油大学
CHINA UNIVERSITY OF PETROLEUM

Simulation of Gas-Solid Flow in a Transfer Chute Based on CFD-DEM Coupling Method

Shan Zhang[a], **Xiaoling Chen[a]**, Fanhao Deng[a], Zekun Wang[b]

[a] *Beijing Key Laboratory of Process Fluid Filtration and Separation, China University of Petroleum - Beijing, Beijing, China*

[b] *BIC-ESAT & State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University, Beijing , China*

Introduction

Bulk solids handling is a crucial stage during coal, ore processing and chemical engineering in various industrial fields. Typically, dust is generated when the bulk materials loaded, dumped and transferred. In the case of belt conveyors, an area of particular concern for dust control occurs during transfer of bulk material from one conveyor to another, namely, transfer point. Usually, a chute is employed at a transfer point to make sure that the loads be discharged in a centralized stream and in the same direction as the receiving conveyor. Therefore, the performance of transfer chutes has a significant impact on not only the efficiency of conveyor belt systems, but also on the level of fugitive dust emissions. In this paper, the CFD-DEM coupling method is used to study dust generation and discharge mechanism in a transfer chute.

Method

There are two different substances in the flow of the transfer chute, air and particles. From a mesoscopic point of view, the gas can be treated as a continuous medium, the particles flow can be treated as discrete phase since it is composed of a large number of discrete particles. This paper uses the CFD-DEM coupling method to solve the gas-solid flow by coupling open source codes OpenFOAM and Ligghts. This method not only considers the complex flow of gas, but also simulates the complex interaction between gas-particle, particle-particle and particle-wall.

The step-by-step CFD-DEM implementation route is shown as follows:

1. Predict momentum exchange in CFD
2. Obtain particle velocity position information by solving Newton's Law in DEM
3. Obtain the particle information in the DEM, identify the grid ID of the particle and set the porosity in CFD
4. Select the reasonable force model in CFD to carry out momentum exchange and transfer to DEM to continue solving
5. Solve the whole flow field based on the Finite Volume Method (FVM) in CFD

Simulation

According to the previous experimental conditions, the iron ore particles with diameter of 4mm were selected, and the shape of the particles was assumed to be spherical. Hertz-Mindlin soft sphere model is adopted as the contact model. Potapov verified the influence of different turbulence models on gas-solid two-phase flow, the results show that the turbulence model has no great effects on the simulation result [1]. In this simulation, the k-ε turbulence model and the KochHill drag model were adopted to conduct the simulation study. The geometric model and mesh as shown in Figure 1, Figure 2.

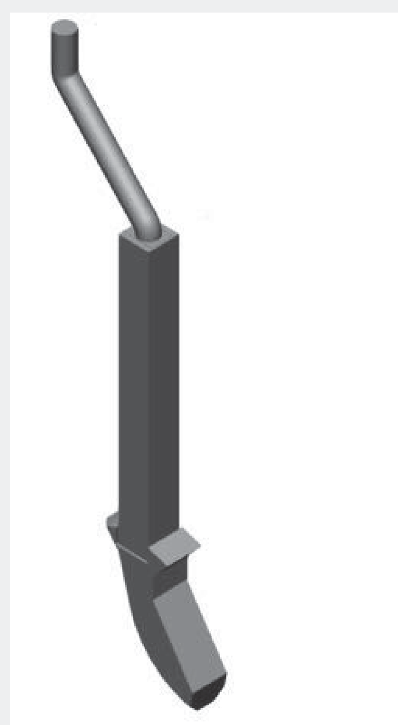

Figure 1: Geometric Model

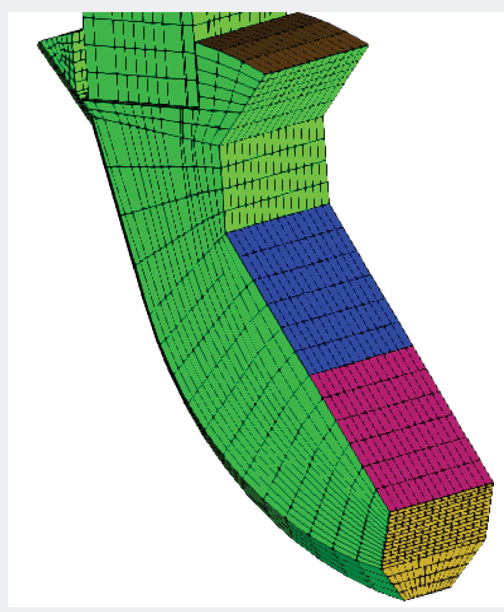

Figure 2: Mesh

Results and discussion

The gas velocity distribution in the transfer chute were obtained from the CFD-DEM coupling. To verify the simulation method, the air velocity distribution at the outlet of the transfer chute obtained from CFD-DEM coupling were compared with the previous experimental results[2], as shown in Fig. 3. It shows that the simulation results have the same trend as the experimental results. It is proved that the gas-solid two-phase flow in the transfer chute can be well predicted by using the CFD-DEM coupling method. It can be used as a powerful tool to evaluate the effect of transfer chute on the particulate flow.

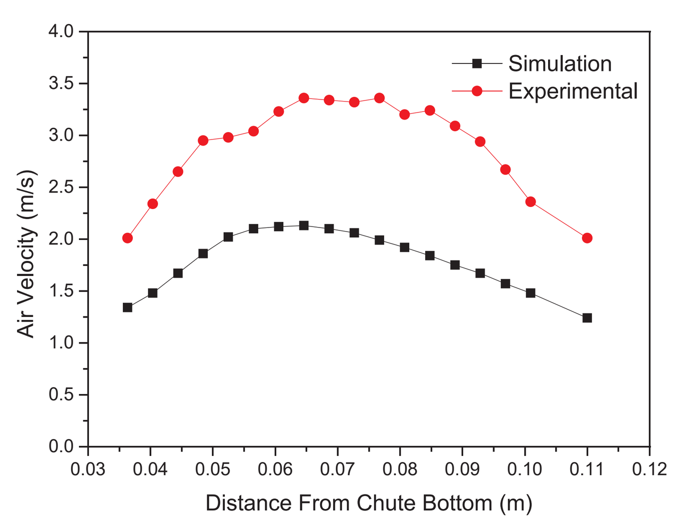

Figure 3: Comparison Between Simulated and Experimental

The turbulent motion of the gas will entrainment the dust, reducing the gas velocity will reduce the dust entrainment. Figure 4 and Figure 5 show the flow conditions of the gas in the chute, and the dust generation can be predicted based on the gas velocity and flow direction. In addition, the particle velocity and gas volume fraction are shown in Figure 6, Figure 7.

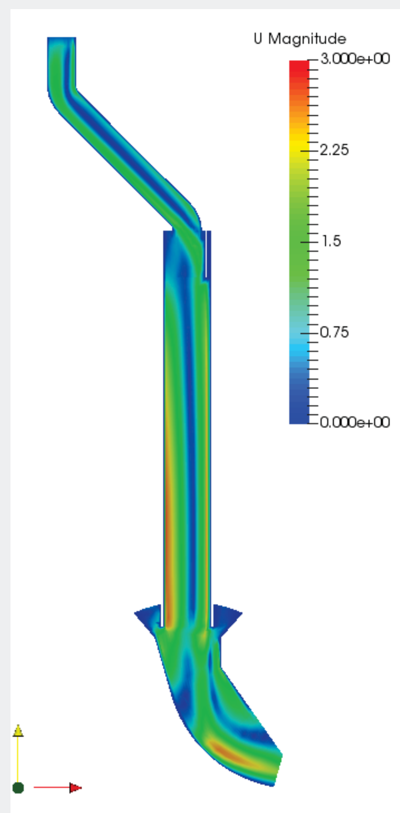

Figure 4: Gas Velocity

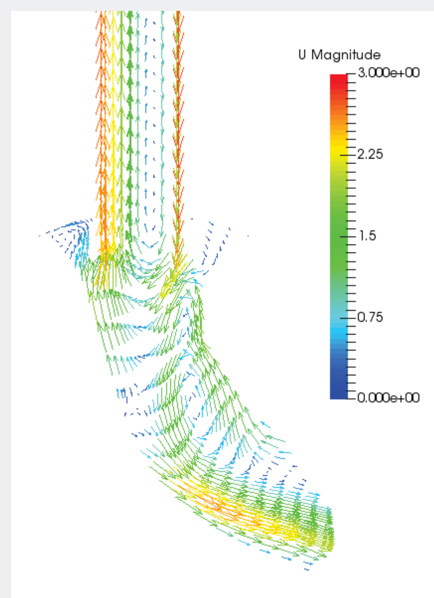

Figure 5: Gas Velocity Vector

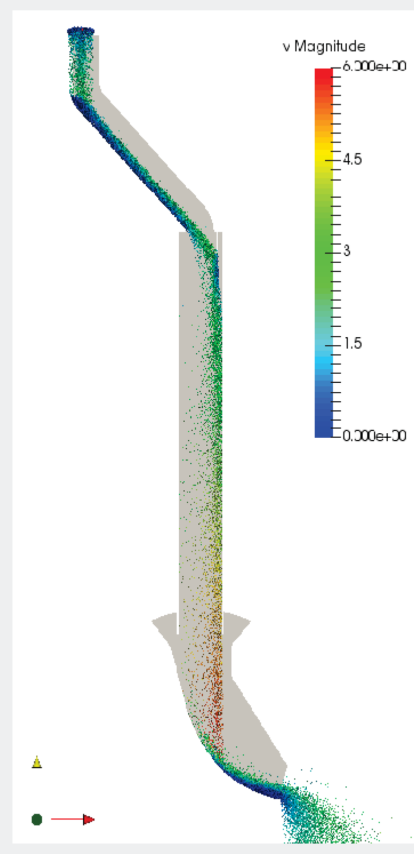

Figure 5: Particle Velocity

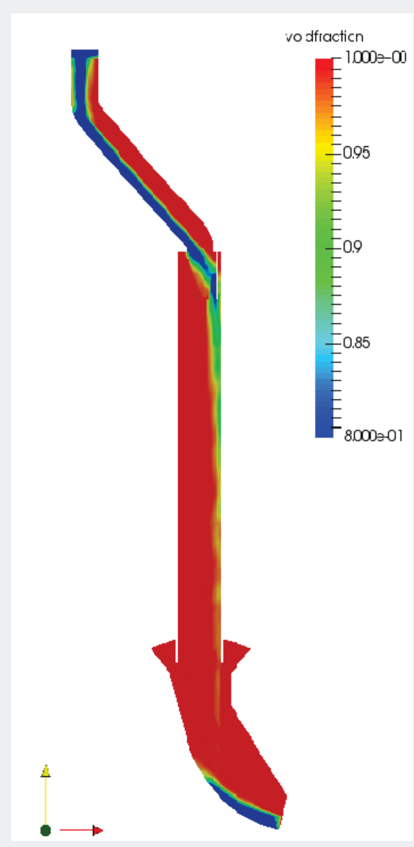

Figure 6: Gas Volume Fraction

References

[1] A. Potapov, X.L. Chen, T.J. Donohue and C.A. Wheeler. Computer Simulation of Airflow around Transfer Chutes via Linked Discrete Element Method - Computational Fluid Dynamics Approach. 11th International Congress on Bulk Materials Storage, Handling and Transportation 2013.

[2] Chen X, Wheeler C. Computational Fluid Dynamics (CFD) modelling of transfer chutes: A study of the influence of model parameters[J]. Chemical Engineering Science, 2013, 95(3):194-202.

Acknowledgements

The authors would like to acknowledge the financial support from National Natural Science Foundation (No.51504273) and Science Foundation of China University of Petroleum, Beijing (No.2462014YJRC 014).