SIMULATION OF GAS-SOLID FLOW IN A TRANSFER CHUTE BASED ON CFD-DEM COUPLING METHOD

SHAN ZHANG[1], XIAOLING CHEN[2*], FANHAO DENG[3] ,ZEKUN WANG[4*]
[1]Beijing Key Laboratory of Process Fluid Filtration and Separation, China University of Petroleum - Beijing, Beijing, China , Hky_shan@163.com
[2]Beijing Key Laboratory of Process Fluid Filtration and Separation, China University of Petroleum - Beijing, Beijing, China ,cxling612@126.com
[3]Beijing Key Laboratory of Process Fluid Filtration and Separation, China University of Petroleum - Beijing, Beijing, China ,dengfanhao1119@163.com
[4] BIC-ESAT & State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University, Beijing , China, zekunwang@pku.edu.cn

Keywords: *Transfer Chute, CFD-DEM, OpenFOAM, LIGGGHTS*

[Abstract] This paper focuses on the simulation of gas-solid two-phase flow in the transfer chute based on the coupled CFD-DEM method. The Computational Fluid Dynamics (CFD) is used to solve the gas phase through the open source software OpenFOAM. Discrete Element Method (DEM) is used to solve the particulate flow, through the open source software LIGGGHTS. This method can simulate the complex gas-particle, particle-particle and particle-wall interaction in the transfer chute at mesoscopic scale. Compared with the previous experimental results, the maximum error is 13%. It proves that the method can be used to analyze the gas-solid two-phase flow in the transfer chute .

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. Since belt conveyors typically operate with the lowest overall transport and maintenance cost per ton, they are extensively employed to transport bulk materials in a great many industries, particularly those associated with mining and mineral processing. 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 view of the current stringent standards concerning the environment, occupational health and safety, the study of the dust generation and discharge in a transfer chute is a matter of high importance to control dust emission during bulk material handling.

At present, the main methods to elimination dusts are emission and blockage. Many dust removal devices are designed to control dust emission to a certain extent, but produce higher economic costs. Practice shows that the a reasonable design of the transfer chute can not only reduce the amount of dusts produced, but also greatly reduce the energy consumption generated by the use of dust removal equipments [1,2]. However, it is difficult to further improve the design of transfer chutes for lack of the understanding of the mechanism of dust generation and the dust dissipation law in the transfer chute. Moreover, due to the existing experimental conditions and test level limitations, experimental research is expensive and time-consuming, and is difficult to obtain particle movement information at the meso scale such as particle velocity, concentration and interacting force.

With the rapid development of computer technology, numerical simulation has become another important method in gas-solid two-phase flow research, which serves as a powerful supplement to experimental research. Wang [3] had applied DEM method to simulate and optimize the bulk material transfer system by using the EDEM, but he did not consider the influence of the gas on the bulk material delivery process. Chen [4] used two-fluid model to simulate the gas-solid flow in the transfer chute, and evaluated the dust-removal effect of different transfer chutes. However, due to the limitation of the model per se, we cannot analyze the issue at particle scale. In this paper, the CFD-DEM coupling method is used to study the mechanism of gas-solid two-phase flow at meso-scale. The airflow in the transfer chute entrains particles, and the effect of gas on the entrainment of particles is different under different gas velocities. In this paper, the mechanism of dust generation is analyzed numerically by the obtained gas velocity and volume fraction at the bottom of the transfer chute.

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, and its physical and mechanical properties, such as velocity, pressure, temperature, density, etc., also experience continuous changes with the position. The motion and energy laws are described by mass conservation equation, Navier-stokes equation and energy equation. The particles flow are composed of a large number of discrete particles as discrete phase, and its velocity, position, motion and force are described by Newton's third law and constitutive relation, respectively [5]. Therefore, this paper uses the CFD method to solve the gas phase through the open source codes OpenFOAM. and uses DEM method to solve the particle through open source codes LIGGGHTS. 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

The gas-solid two-phase flow mechanism in the transfer chute is studied based on the CFD-DEM coupling method. According to the previous experimental results, the iron ore particles with diameter of 4mm were selected, and the shape of the particles was assumed to be spherical. The geometry of the transfer chute is shown in Figure 1, and its height is 2.05m. Hertz-Mindlin soft sphere model is adopted as the contact model, and the particle physical parameters required in the simulation are shown in table 1. 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 [6]. In this simulation, the k-ε turbulence model and the Gidaspow drag model were adopted to conduct the simulation study.

Table1: Particle Physical Property Parameters

Parameters	Units	Value
Particle density	Kg/m^3	3948
Sliding friction coefficient		0.48
Rolling friction coefficient		0.21
Recovery coefficient		0.48
Mass flow rate	Kg/s	4.19

First, the CFD-DEM simulation of a single particle is carried out to verify its feasibility and the grid independence of the grid in CFD. Su[5] compared the effects of different grid scales on particle settling velocity in regular rectangular geometry, results show that the particle settling velocity does not change with the increase of ratio, when the ratio of mesh size to particle diameter is greater than 5 for large scale particles. Multiple grids are employed to conduct mesh independence study, the results show that when the number of grids reaches around 18000, both the accuracy and efficiency can be obtained. Thus, this grid with 18426 grids was used in subsequent simulations. The grid partitioning is shown in Figure 2.

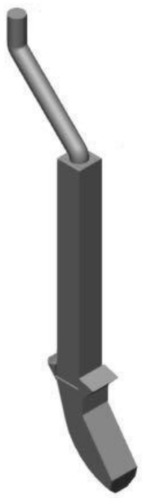

Figure 1: The geometry of the chute

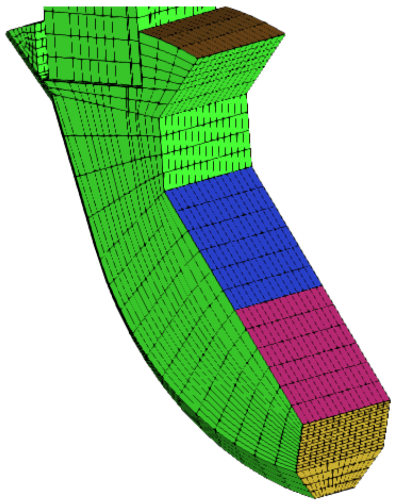

Figure 2: Grid Partitioning

Results and discussion

Through the CFD-DEM coupling simulation we obtained the gas velocity in the transfer chute. In addition, We used PIV measurement technology to measure the gas velocity at the bottom of the transfer chute.Comparing the simulation results with the experimental data, as shown in Figure 3, it shows that the simulation result is comparable with the experimental result and the maximum error is 13%. 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.

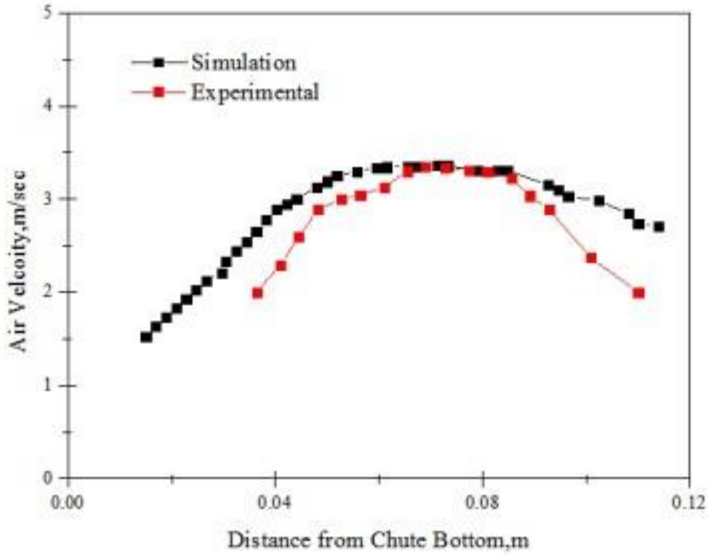

Figure 3: comparison between simulated and experimental

In addition, some other were also obtained through simulation, which are illustrated in the figures4-5. Figure 4 is the gas phase and the particle phase velocity. We can not only obtain the continuous change of gas velocity, but also the velocity and position information of each particle. Figure 5 is the gas volume fraction. These data can help to predict the dust generation in transfer chute dust.

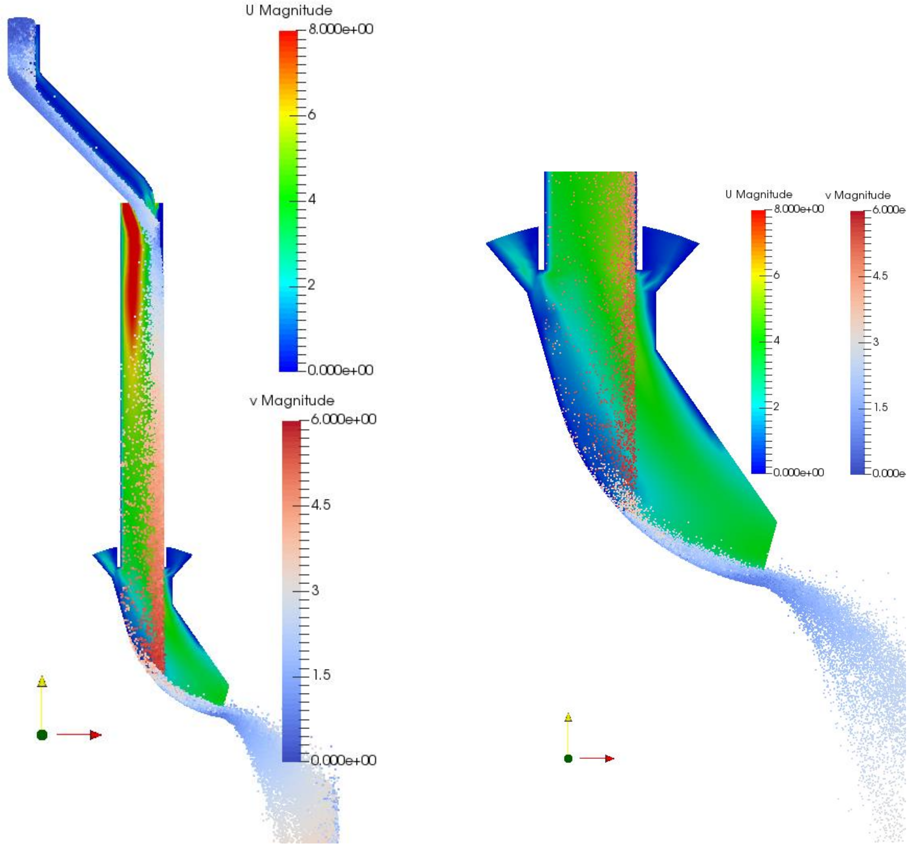

Figure 4: gas phase and particle phase velocity (U-gas velocity; V-phase velocity)

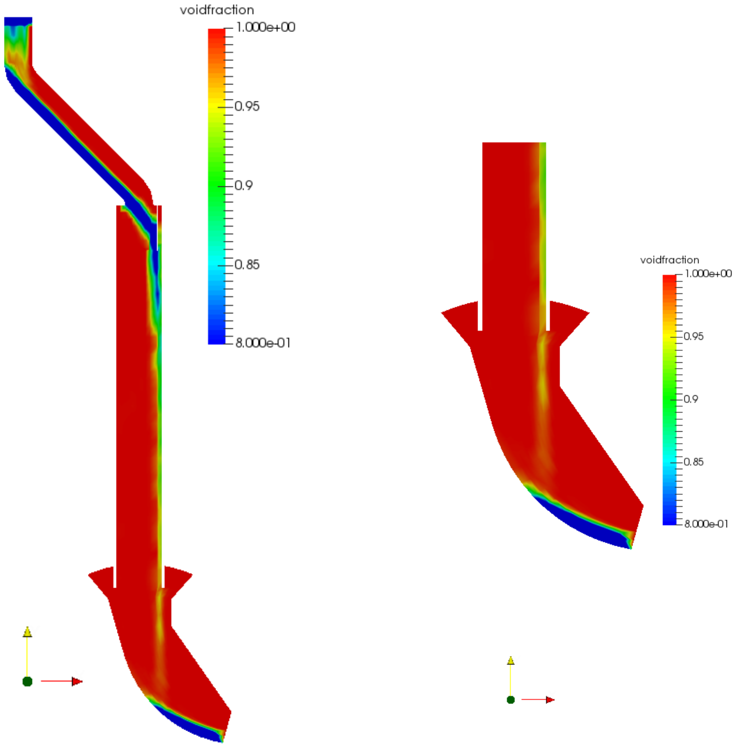

Figure 5: Gas volume fraction

Conclusion

From the above, the gas-solid two-phase flow simulation in the transfer chute can be used to obtain the internal flow parameters on the mesoscopic scale based on the CFD-DEM coupling method. Compared with the two-fluid model, it not only reduces the model hypothesis, but also can obtain more information on the mesoscopic scale, such as the speed, position and force of each particle, which can make the results more accurate. Using this method, different parameters can be changed to simulate the flow mechanism in the transfer chute, which can provide possible optimizations for dust emission controllong.

Acknowledgements

We acknowledge the financial support from National Natural Science Foundation (No.51504273) and Science Foundation of China University of Petroleum, Beijing (No.2462014YJRC014)The simulations have been conducted at Tianhe II Super Computer in the National Supercomputing Center in Guangzhou with the effective support from Beijing Paratera Technology Co., Ltd.

References

[1] Maynard, E.P., Designing Transfer Chutes to Eliminate Spillage and Dusting. Bulk Solids Handling,2003. 23(SPECIAL): p. 28-34.

[2] Petro, G., Baxter, T. and Cabrejos, F., Chutes: The ultimate dust control. Powder and Bulk Engineering, 2004. 18(7): p. 36-46.

[3] Wang, K.L. Study on DEM simulation of unloading trajectory and reprint process of belt conveyor [D]. Northeastern University, 2014..

[4] Chen, X.L., Wheeler, C.A., Donohue, T.J., et al., Evaluation of dust emissions from conveyor transfer chutes using experimental and CFD simulation. International Journal of Mineral Processing, 2012.110-111: p. 101-108..

[5] Su, D.S. Study on flow and sediment movement based on CFD-DEM coupling simulation method [D]. Tianjin University, 2015.

[6] Potapov, A., Chen, X., Donohue, T., et al. Computer simulation of airflow around transfer chutes via linked discrete element method - Computational fluid dynamics approach. in 11th International Conference on Bulk Materials Storage, Handling and Transportation, ICBMH 2013, July 2, 2013 - July 4,2013. 2013. Newcastle, NSW, Australia: Centre for Bulk Solids and Particulate Technologies (CBSPT).