Reply to the Comment on "Numerical study on pore clogging mechanism in pervious pavements"

G. Ma, J. Zhang, Z. Dai, R. She, S. Xia, N. Hu

Reply to the Comment on "Numerical study on pore clogging mechanism in pervious pavements"

Ma Guodong[1], Zhang Jiong[2,*]

Abstract: In this paper, to the best of our knowledge, we replied to the comment on our published article "Numerical study on pore clogging mechanism in pervious pavements" (Zhang et al., 2018). First, we solved the problem about inappropriate descriptions to the cited papers and some errors in equations. Second, we explained the relationship between fluid cell size in CFD (computational fluid dynamics) and particle size in DEM (discrete element method) in our CFD-DEM model, and we verified the reliability and accuracy of our CFD-DEM model using single-particle sedimentation case. It demonstrated that the CFD-DEM method used in our study was acceptable.

Key Words: computational fluid dynamics; discrete element method; pore clogging

1. Reply to the "Inappropriate expressions."

The comments said that the references (Luo et al., 2015; Blais et al., 2016; Li and Li, 2018) at the end of the sentence *"The commercial CFD software of Fluent coupled with commercial DEM software of EDEM to realize the connection of CFD with DEM*" would cause misunderstanding to readers. In fact, what we initially mean is that all of these references can realize the CFD-DEM coupled model, and using Fluent software coupled with EDEM software to realize the connection of CFD with DEM is only one of the choices. Also, Wu et al. (2018), Shao et al. (2013) and Yu et al. (2018) realized the CFD-DEM modeling based on Fluent and EDEM software.

In addition, the "volume of friction (VOF)" method should be corrected as the "volume of fluid (VOF)" method (Hirt and Nichols, 1981).

2. Reply to the errors of the "Mathematical equations."

The Eq. (6) in our paper should be corrected as

$$\vec{f}_{fluid,i} = \vec{f}_{drag,i} + \frac{4}{3}\pi r^3 \nabla p$$

In fact, the VOF method used in our study is only for tracking the interface between water and air. Wu et al. (2018) developed a DEM-VOF method, and the continuity equation used for CFD-DEM coupled model in his study is the same as ours. Therefore, the selection of continuity equation in the water-air two-phase flow in CFD-DEM modeling should be determined by the conditions.

Last but not least, the grid size in some region of our CFD-DEM model is definitely a little bit smaller than the aggregate size. As illustrated in the comment on "Numerical study on pore clogging mechanism in pervious pavements," it will cause inaccuracy to some degree when calculating the porosity in fluid cells using unresolved CFD-DEM approach. However, the results in our CFD-DEM modeling are acceptable, and the reasons are as follows:

First, as introduced in the paper of Sun and Sakai (2015), a typical gas–solid–liquid flow (shown in Fig.1) involves fluid–fluid interaction (evolving fluid interface), fluid–solid interaction (fluid–particle momentum exchange) and solid–solid interaction (particle–particle collision).

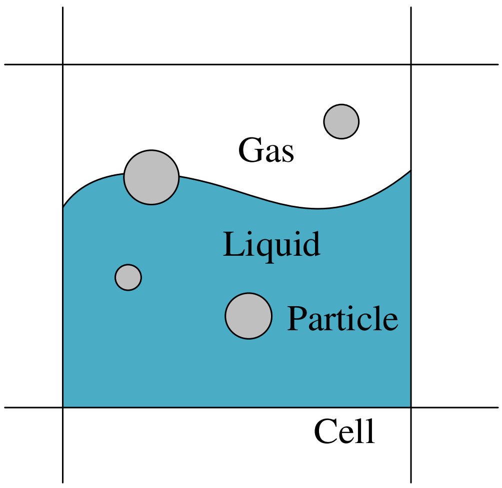

Fig.1 The gas–solid–liquid flow in a fluid cell

To the best of our knowledge, the core of our study is to observe the sediments clogging in the pervious concrete. Thus we focus only on the fluid-solid interaction, especially the water-sediment interaction. In our CFD-DEM model (shown in Fig. 2), the particles (aggregates) with a diameter of 4 mm, 8 mm and 12 mm are boned to form the pervious concrete pavement, and these particles are fixed and cannot have any motions. The interface between water-gas is stable and above the pervious concrete pavement model (as depicted in Fig.3). Therefore, aggregate-gas interaction and sediment-gas interaction are neglected in our CFD-DEM model. Even though there are water flow into the pores of the pervious concrete pavement, this study does not analyze these interactions including the aggregate-water interaction, because these factors exert little influence to sediment-water interaction.

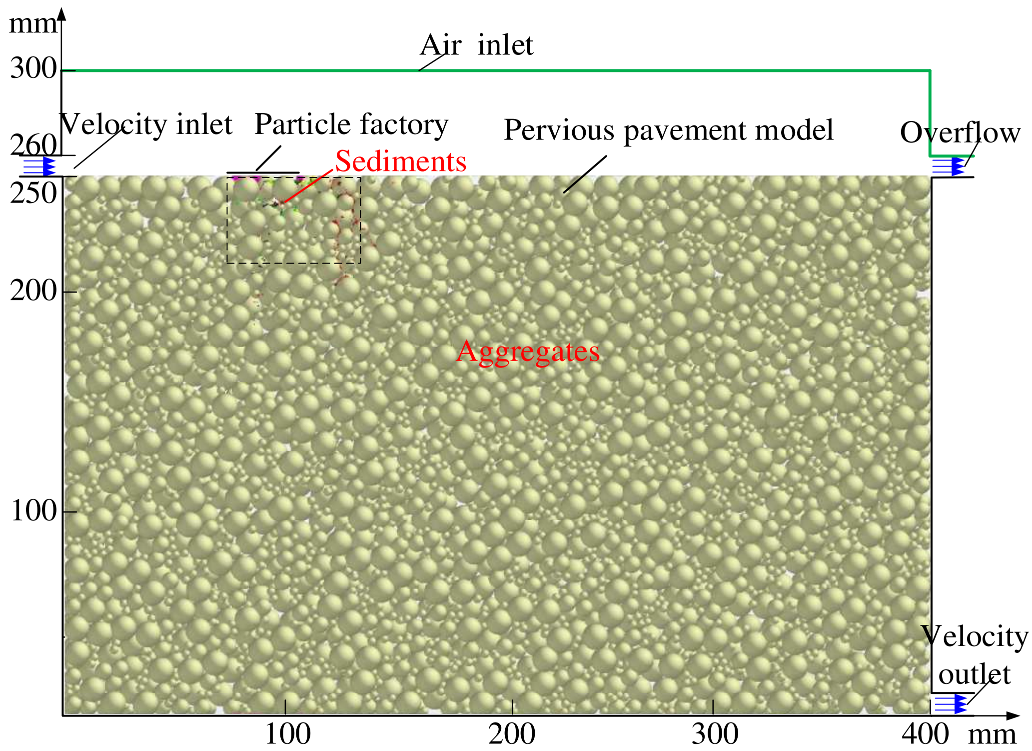

Fig. 2 The CFD-DEM model

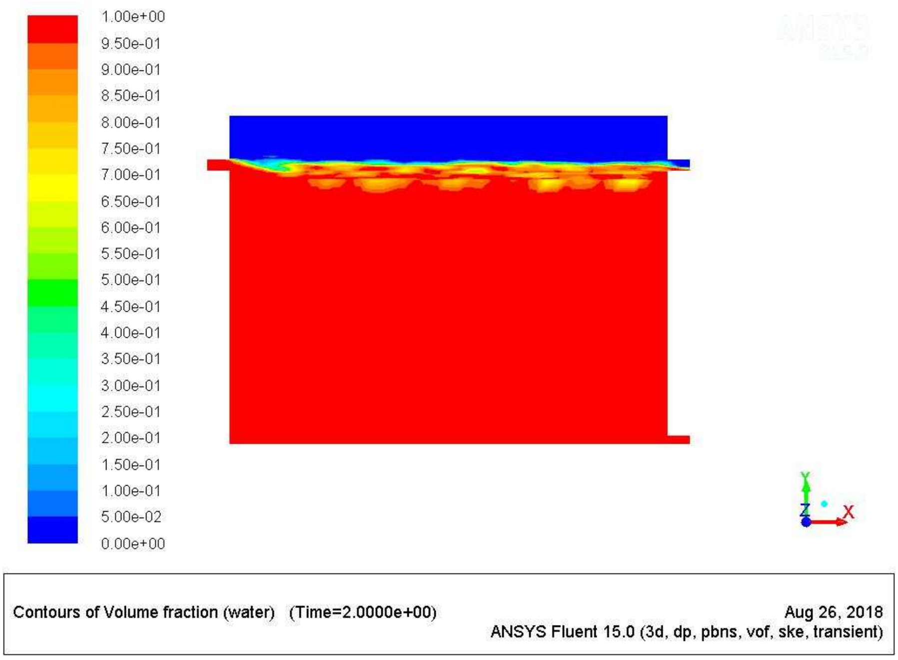

Fig.3 The interface between water and air

Second, as shown in Fig. 4, the grids in region A and B are refined to observe the interface between water and air. It is evident that the grid size is small, but the girds contain no particles. Therefore, it does not violate the rule of unresolved CFD-DEM approach and would not influence the CFD-DEM simulation. The grids in region C are also refined to observe the interface between water and air, and they contain some aggregates and sediments. The grid size is smaller than the aggregates (12 mm diameter), which may cause inaccuracy to the CFD-DEM simulation. However, as mentioned above, the aggregates are fixed and have no motions. In this study, we do not consider their influence on the CFD-DEM modeling. Most importantly, the grid size in region C is still larger than the particles of sediments. Last, the grid size (appropriately10 mm) in region D is just a little bit smaller than the aggregate size. It will not change the motions of sediments significantly, and the reasons are the same as above mentioned. Therefore, it is credible to model the sediments transport using our CFD-DEM model.

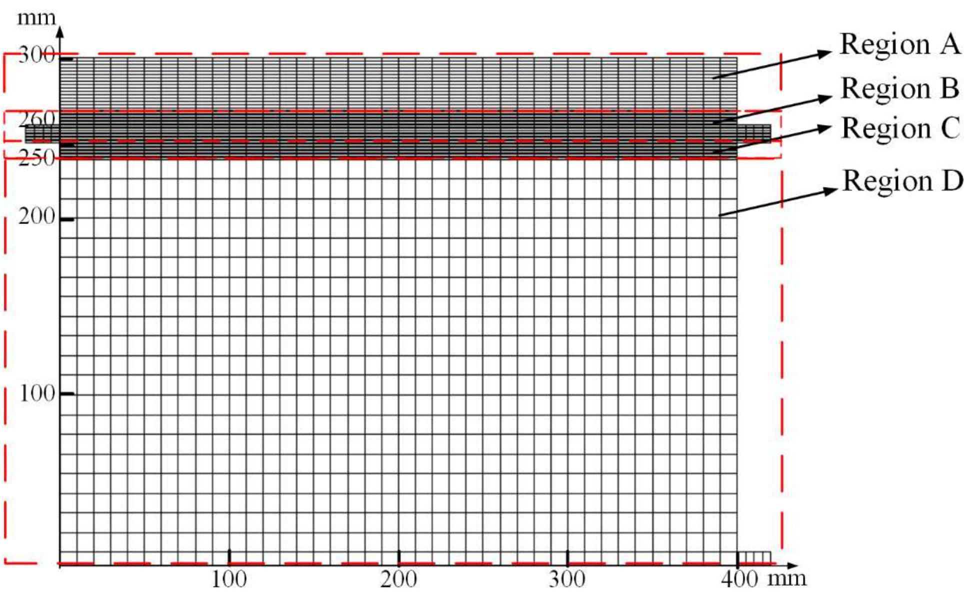

Fig. 4 The grid size of CFD-DEM model

Furthermore, we used single-particle sedimentation simulation to validate the CFD-DEM method used in our study. As shown in Fig.5, the tank is full of water, whose dimensions are 200 mm × 200 mm × 200 mm. The walls of the tank are defined as nonslip boundary conditions. A small particle (diameter 3 mm) is placed 180 mm high from the bottom surface, and it will fall freely until colliding with a fixed big particle (diameter 12 mm, placed 80 mm high from the bottom surface) in the water. The density of particles is 2650 kg/m^3. The viscosity and density of the water are 1×10^{-3} Pa·s and 1000 kg/m^3, respectively. The grid size is set as 40 mm, 20 mm, and 10 mm separately to observe the effect on the motion of single-particle. In CFD simulation, the standard k-ε turbulence model and PISO (Pressure Implicit with Splitting of Operator) method are used. The time step of DEM simulation is 2 × 10^{-5} s, and the time step of CFD simulation is 2 × 10^{-4} s.

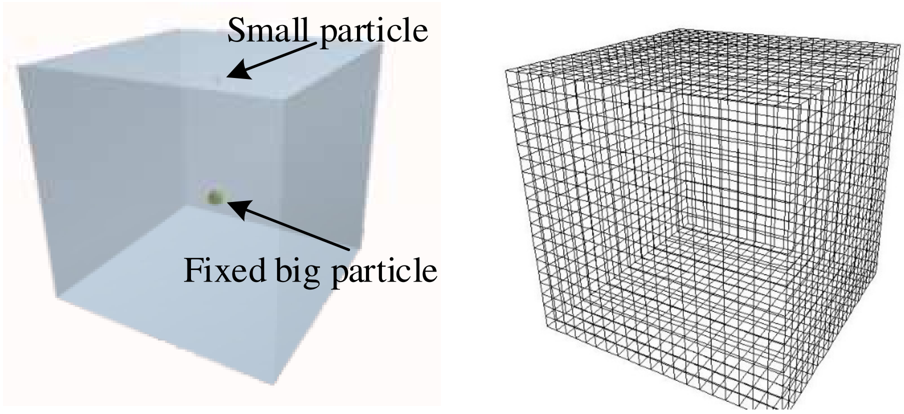

Fig.5 The single-particle sedimentation in water and grid

Fig.6 shows the results of the single-particle sedimentation simulation. As shown in Fig.6, the grid sizes of 40 mm and 20 mm are larger than the particle size while the grid size of 10 mm is smaller than the particle size. However, results show the velocities of single-particle in different grid size are almost the same. Moreover, all of the parameters and configuration of single-particle sedimentation simulation are the same as the CFD-DEM model in our study. Consequently, it demonstrated the results using the current grid size (Fig.4) in our CFD-DEM model are acceptable.

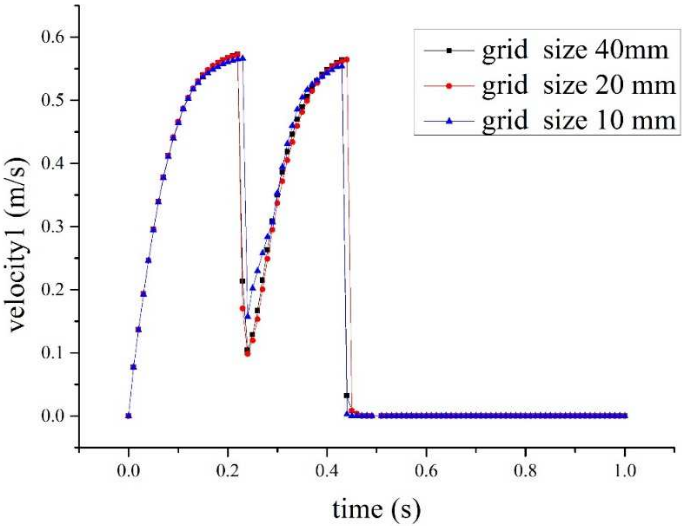

Fig.6 The velocity of single-particle in different grid size

In our further study, we will investigate the CFD-DEM coupling modeling methodology and contrast the numerical results using different CFD-DEM method such as the resolved, unresolved, and semi-resolved method.

Acknowledgements:

The authors would like to thank for Dr. Li Linmin from Hohai University, the reviewers and Prof. Corrado Corradini (Editor in chief, Journal of Hydrology). Thank you for the constructive comments on our article.

References:

Blais, B., Lassaigne, M., Goniva, C., Fradette, L., Bertrand, F., 2016. Development of an unresolved CFD–DEM model for the flow of viscous suspensions and its application to solid–liquid mixing. J. Comput. Phys. 318, 201–221.

Hirt, C.W., Nichols, B.D., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. J. Comput. Phys. 39 (1), 201–225

Li, L., Li, B., 2018. Implementation and validation of a volume-of-fluid and discrete element-method combined solver in OpenFOAM. Particuology 39, 109–115.

Luo, K., Wu, F., Yang, S., Fan, J., 2015. CFD–DEM study of mixing and dispersion behaviors of solid phase in a bubbling fluidized bed. Powder Technol. 274, 482–493.

Wu, L., Gong, M. and Wang, J., 2018. Development of a DEM–VOF Model for the Turbulent Free-Surface Flows with Particles and Its Application to Stirred Mixing System. Ind. Eng. Chem. Res., 57(5): 1714-1725.

Shao, T., Hu, Y., Wang, W., Jin, Y. and Cheng, Y., 2013. Simulation of Solid Suspension in a Stirred Tank Using CFD-DEM Coupled Approach. Chinese J. Chem. Eng., 21(10): 1069-1081.

Sun, X. and Sakai, M., 2015. Three-dimensional simulation of gas–solid–liquid flows using the DEM–VOF method. Chem. Eng. Sci., 134: 531-548.

Yu, H., Cheng, W., Xie, Y. and Peng, H., 2018. Micro-scale pollution mechanism of dust diffusion in a blasting driving face based on CFD-DEM coupled model. Environ. Sci. Pollut. R., 25(22): 21768-21788.

Zhang, J., Ma, G., Dai, Z., Ming, R., Cui, X., She, R., 2018. Numerical study on pore clogging mechanism in pervious pavements. J. Hydrol. 565, 589-598.