

1 Reply to the Comment on “Numerical study on pore clogging mechanism  
2 in pervious pavements”

3 Ma Guodong<sup>1</sup>, Zhang Jiong<sup>2,\*</sup>

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

14

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

17

18 **1. Reply to the “Inappropriate expressions.”**

19 The comments said that the references (Luo et al., 2015; Blais et al.,  
20 2016; Li and Li, 2018) at the end of the sentence “*The commercial CFD*  
21 *software of Fluent coupled with commercial DEM software of EDEM to*  
22 *realize the connection of CFD with DEM*” would cause misunderstanding

23 to readers. In fact, what we initially mean is that all of these references  
24 can realize the CFD-DEM coupled model, and using Fluent software  
25 coupled with EDEM software to realize the connection of CFD with  
26 DEM is only one of the choices. Also, Wu et al. (2018), Shao et al. (2013)  
27 and Yu et al. (2018) realized the CFD-DEM modeling based on Fluent  
28 and EDEM software.

29 In addition, the “volume of friction (VOF)” method should be  
30 corrected as the “volume of fluid (VOF)” method (Hirt and Nichols,  
31 1981).

## 32 **2. Reply to the errors of the “Mathematical equations.”**

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

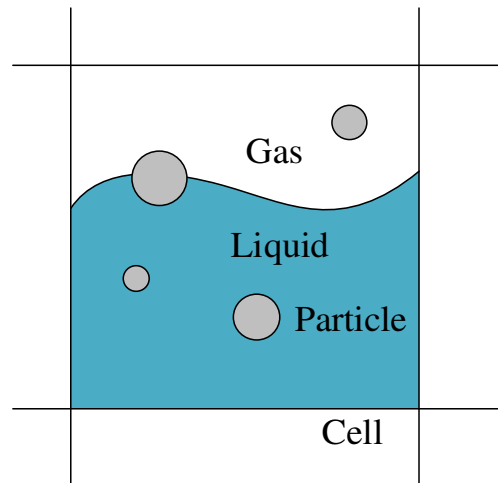
$$34 \quad \vec{f}_{fluid,i} = \vec{f}_{drag,i} + \frac{4}{3}\pi r^3 \nabla p$$

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

41 Last but not least, the grid size in some region of our CFD-DEM  
42 model is definitely a little bit smaller than the aggregate size. As  
43 illustrated in the comment on “Numerical study on pore clogging  
44 mechanism in pervious pavements,” it will cause inaccuracy to some

45 degree when calculating the porosity in fluid cells using unresolved  
46 CFD-DEM approach. However, the results in our CFD-DEM modeling  
47 are acceptable, and the reasons are as follows:

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

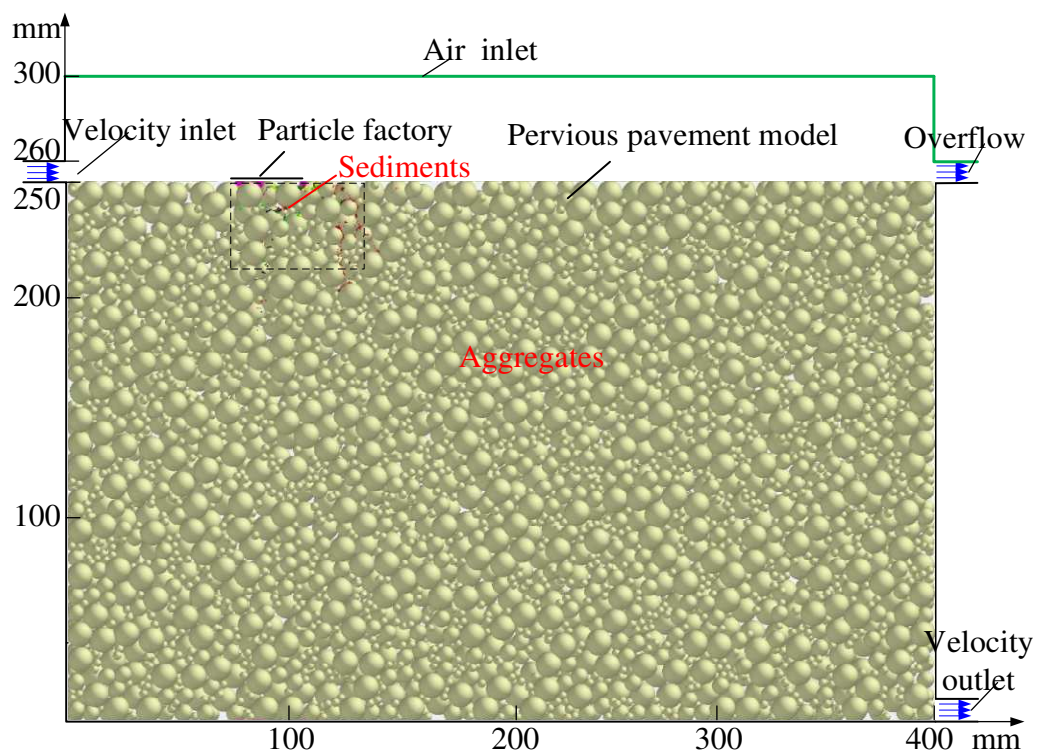


53

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

55 To the best of our knowledge, the core of our study is to observe the  
56 sediments clogging in the pervious concrete. Thus we focus only on the  
57 fluid-solid interaction, especially the water-sediment interaction. In our  
58 CFD-DEM model (shown in Fig. 2), the particles (aggregates) with a  
59 diameter of 4 mm, 8 mm and 12 mm are bonded to form the pervious  
60 concrete pavement, and these particles are fixed and cannot have any  
61 motions. The interface between water-gas is stable and above the

62 pervious concrete pavement model (as depicted in Fig.3). Therefore,  
63 aggregate-gas interaction and sediment-gas interaction are neglected in  
64 our CFD-DEM model. Even though there are water flow into the pores of  
65 the pervious concrete pavement, this study does not analyze these  
66 interactions including the aggregate-water interaction, because these  
67 factors exert little influence to sediment-water interaction.



68

69

Fig. 2 The CFD-DEM model

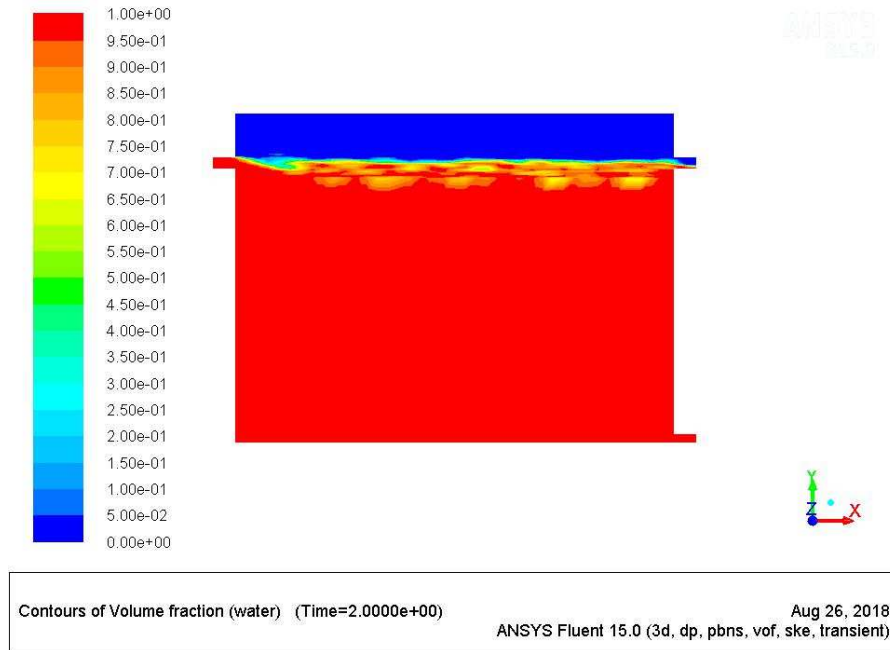


Fig.3 The interface between water and air

70

71

72 Second, as shown in Fig. 4, the grids in region A and B are refined to

73 observe the interface between water and air. It is evident that the grid size

74 is small, but the grids contain no particles. Therefore, it does not violate

75 the rule of unresolved CFD-DEM approach and would not influence the

76 CFD-DEM simulation. The grids in region C are also refined to observe

77 the interface between water and air, and they contain some aggregates

78 and sediments. The grid size is smaller than the aggregates (12 mm

79 diameter), which may cause inaccuracy to the CFD-DEM simulation.

80 However, as mentioned above, the aggregates are fixed and have no

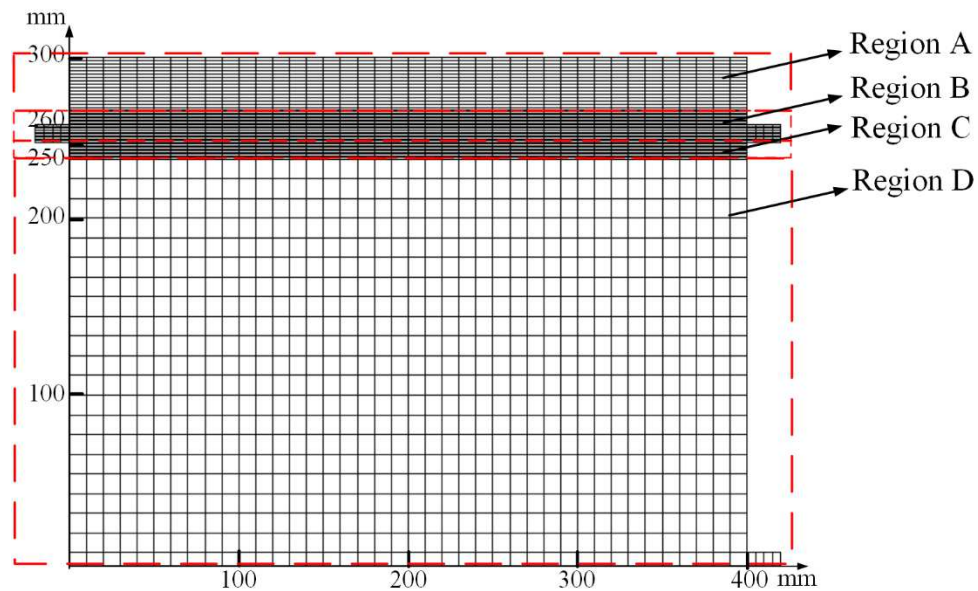
81 motions. In this study, we do not consider their influence on the

82 CFD-DEM modeling. Most importantly, the grid size in region C is still

83 larger than the particles of sediments. Last, the grid size (appropriately 10

84 mm) in region D is just a little bit smaller than the aggregate size. It will

85 not change the motions of sediments significantly, and the reasons are the  
86 same as above mentioned. Therefore, it is credible to model the sediments  
87 transport using our CFD-DEM model.

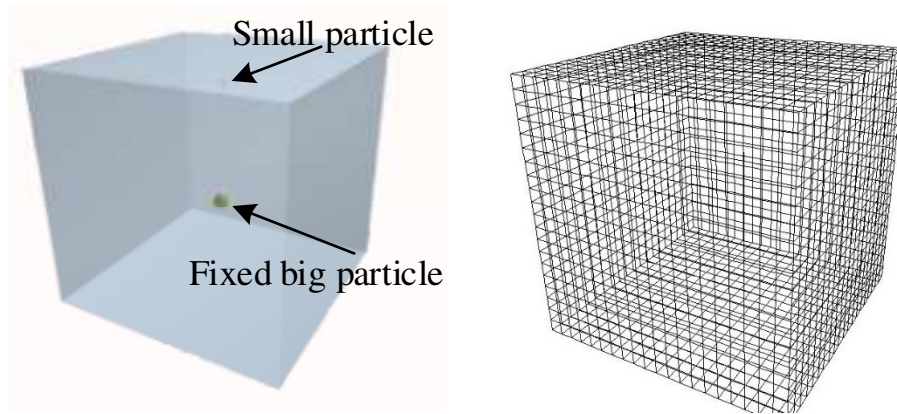


88

89 Fig. 4 The grid size of CFD-DEM model

90 Furthermore, we used single-particle sedimentation simulation to  
91 validate the CFD-DEM method used in our study. As shown in Fig.5, the  
92 tank is full of water, whose dimensions are 200 mm × 200 mm × 200  
93 mm. The walls of the tank are defined as nonslip boundary conditions. A  
94 small particle (diameter 3 mm) is placed 180 mm high from the bottom  
95 surface, and it will fall freely until colliding with a fixed big particle  
96 (diameter 12 mm, placed 80 mm high from the bottom surface) in the  
97 water. The density of particles is 2650 kg/m<sup>3</sup>. The viscosity and density of  
98 the water are 1×10<sup>-3</sup> Pa·s and 1000 kg/m<sup>3</sup>, respectively. The grid size is  
99 set as 40 mm, 20 mm, and 10 mm separately to observe the effect on the  
100 motion of single-particle. In CFD simulation, the standard k-ε turbulence

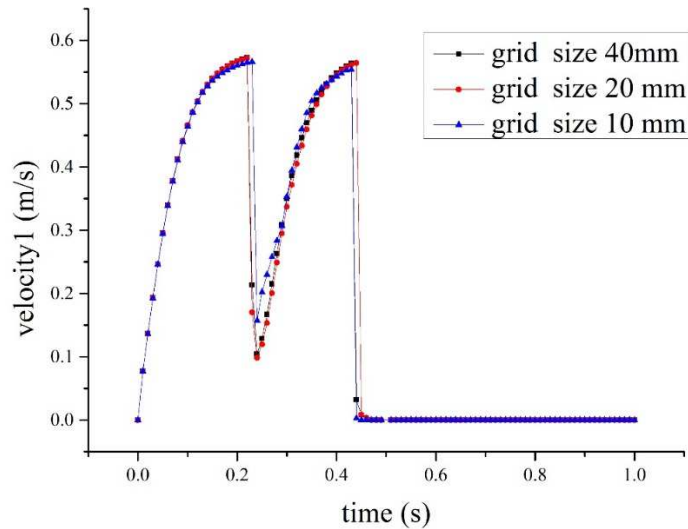
101 model and PISO (Pressure Implicit with Splitting of Operator) method are  
102 used. The time step of DEM simulation is  $2 \times 10^{-5}$  s, and the time step  
103 of CFD simulation is  $2 \times 10^{-4}$  s.



104

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

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



115  
116 Fig.6 The velocity of single-particle in different grid size

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

121 **Acknowledgements:**

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

126 **References:**

127 Blais, B., Lassaigne, M., Goniva, C., Fradette, L., Bertrand, F., 2016.  
128 Development of an unresolved CFD-DEM model for the flow of viscous  
129 suspensions and its application to solid-liquid mixing. J. Comput. Phys.  
130 318, 201-221.

131 Hirt, C.W., Nichols, B.D., 1981. Volume of fluid (VOF) method for the



132 dynamics of free boundaries. *J. Comput. Phys.* 39 (1), 201–225

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

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

139 Wu, L., Gong, M. and Wang, J., 2018. Development of a DEM–VOF  
140 Model for the Turbulent Free-Surface Flows with Particles and Its  
141 Application to Stirred Mixing System. *Ind. Eng. Chem. Res.*, 57(5):  
142 1714-1725.

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

146 Sun, X. and Sakai, M., 2015. Three-dimensional simulation of gas–solid–  
147 liquid flows using the DEM–VOF method. *Chem. Eng. Sci.*, 134:  
148 531-548.

149 Yu, H., Cheng, W., Xie, Y. and Peng, H., 2018. Micro-scale pollution  
150 mechanism of dust diffusion in a blasting driving face based on  
151 CFD-DEM coupled model. *Environ. Sci. Pollut. R.*, 25(22):  
152 21768-21788.

153 Zhang, J., Ma, G., Dai, Z., Ming, R., Cui, X., She, R., 2018. Numerical

154 study on pore clogging mechanism in pervious pavements. J. Hydrol. 565,  
155 589-598.