CFD-DEM modelling of sediment transport in sewer systems under steady and unsteady flow conditions

Maryam Alihosseini, Sveinung Sægrov and Paul Uwe Thamsen

ABSTRACT

Numerical and experimental investigations were undertaken to study sediment transport under steady flow conditions and under flush waves in sewer pipes. Experiments were carried out with sand and gravel of different size distributions under smooth and rough bed conditions. Moreover, different hydraulic boundary conditions were investigated for flush waves. The numerical part of this study was carried out in the computational fluid dynamics (CFD) software ANSYS Fluent, which is two-way coupled to the Discrete Element Method (DEM) software EDEM. The main focus of this study is to determine if the CFD-DEM coupled method could reasonably predict the behaviour of sediments in sewers and thus be used for studying various features of sediment transport that are not easy to determine in laboratory experiments or *in-situ* measurements. Furthermore, it is important to replace the traditional empirical approaches developed for fluvial conditions with new methodologies, which are able to consider the high number of variables involved in sediment transport in sewers. The numerical model was validated with laboratory experiments and used to study details of sediment transport processes in sewers.

Key words | CFD-DEM, sediment transport, self-cleansing, sewer flushing, sewer pipes

Maryam Alihosseini (corresponding author)
Paul Uwe Thamsen
Department of Fluid System Dynamics,
Institute of Technology of Berlin,
Straße des 17. Juni 135, 10623 Berlin,
Germany
E-mail: *m.alihosseini@tu-berlin.de*

Sveinung Sægrov
Department of Civil and Environmental Engineering,
Norwegian University of Science and Technology,
Valgrinda 2000, 424, Valgrinda, S. P. Andersens veg 5, Trondheim,
Norway

INTRODUCTION

Solids in sewers build deposits on the bottom of channels at low flow rates, leading to various challenges for the sewer system itself as well as for the environment. Therefore, it is important to avoid sewer sediment deposits by obeying the low flow criteria in self-cleansing design concepts of sewer systems (Vongvisessomjai *et al.* 2010). Nevertheless, it is not always possible to avoid deposition despite taking into account the self-cleansing criteria. Then, flushing actions must be undertaken to remove sediments to guarantee a smooth sewer operation. Among the different existing cleansing devices, flushing gates are frequently used. These devices are designed to produce flushing waves with high flow velocities and shear stresses, which re-suspend and transport the solids along sewers (Campisano *et al.* 2004).

Calculation of sediment transport in steady flow is performed using traditional formulas derived for fluvial flows. The threshold for the sediment transport is normally based on theoretical equations such as the Shields theory or visual observations of particle motion on the bed (Ab Ghani 1993). These simplified approaches mostly fail in predicting sediment transport in complex sewer flow conditions. Moreover, the flow in sewer systems is mostly unsteady and non-uniform due to the presence of hydraulic structures or storm events. Thus, it is important to find a promising solution for modelling the sediment transport in sewers in both steady and unsteady flow conditions. As sediment transport is a very complex phenomenon that includes various influencing parameters, three-dimensional numerical modelling may be a suitable tool to investigate this. Despite numerous attempts to develop numerical models to simulate sediment transport in sewer systems, there is still a need to find a reliable 3D model to study different features of sediment transport, which are not easily observable in the laboratory or field. Especially, measuring wall shear stress in real sewers is difficult due to the clogging of measurement devices (Bonakdari *et al.* 2008). Also measuring the sediment velocity is a big challenge. In addition, understanding and modelling sediment transport of solids mixtures is still in a preliminary state. Traditional theories for flows in laboratory channels using mono-sediments in steady uniform flows, may have only limited applicability to real sewer conditions. It is necessary to understand the behaviour of particles in detail

doi: 10.2166/wst.2020.030

and model this phenomenon using new methodologies and computer-based techniques (Ashley *et al.* 2004).

With increasing computational resources and advancements in numerical methods, computational fluid dynamics (CFD) provides a good alternative to investigate complicated phenomena in a less expensive and more flexible way. Some researchers (e.g. Stovin *et al.* 1999; He *et al.* 2004; Bonakdari *et al.* 2015) have used the particle tracking facility of CFD solvers to simulate sedimentation in sewer systems. However, these models do not include interactions between particles and all particles are assumed to be spherical. Consequently, real behaviour of particles is not modelled. In addition to grid-based methods, mesh-free (particle-based) methods such as smoothed particle hydrodynamics (SPH) have been also applied to modelling solid-fluid two-phase flows. Alihosseini & Thamsen (2016) provide a comprehensive overview of SPH application for sediment transport in free-surface flows. SPH treats both the fluid and solids as particles in a Lagrangian frame. On the one hand, this permits the overcoming of some problems related to the computational mesh, such as the treatment of moveable boundaries (Vetsch 2012), but on the other hand it makes this method highly computationally expensive.

Another numerical approach for modelling of granular particles in a more realistic way is the Discrete Element Method (DEM), introduced for the first time by Cundall & Strack (1979). DEM is a powerful tool for the numerical modelling of systems with many particles. Recent investigations (e.g. Schmeeckle 2014; Sun & Xiao 2016; Bravo-Blanco *et al.* 2017) demonstrated that a coupling of CFD and DEM could enhance the particle tracking facility of CFD codes by resolving particle contacts, modelling bonded particles and non-spherical particles.

To the best knowledge of the authors, the application of the coupled CFD-DEM method in sewer systems is rarely available. Therefore, a study has been conducted to validate the CFD-DEM model with laboratory data. The overall aim of this research is to evaluate the applicability of the CFD-DEM approach in modelling sediment transport in sewer systems. The laboratory experiments had to be simplified, for instance by limiting the depositions to a short part in the channel, or using only mineral materials. Therefore, the current laboratory research cannot provide specific values of velocity or shear stress to be used in design of real sewer systems. This research could be extended for conditions in real systems. This would be a topic for further research.

Different setting parameters of the DEM model, the CFD model and the coupling server were varied to find the best fitted numerical set-up, such as the mesh size, the collision contact model, DEM and CFD time step sizes, fluid-particle interaction forces and so on. The numerically obtained critical velocity for incipient motion and the scouring efficiency of flush waves were compared to the experimental data. Furthermore, qualitative comparisons have been performed between these two data sets regarding the transport of sediments. Good agreement between experimental and numerical results confirmed that the model can reproduce the behaviour of the sediments quite well under steady and unsteady flow conditions. The validation is presented in different papers of the authors (Alihosseini & Thamsen 2018, 2019).

In the current paper, some results of the validated numerical model are presented concerning the velocity of particles and the bottom shear stress.

MATERIALS AND METHODS

Numerical model

In the CFD-DEM coupling method, the Euler-Lagrange approach is used, in which the fluid phase is treated as a continuum and the dispersed phase is solved by tracking individual particles. DEM tracks the motion of particles based on Newton's second law, calculates the contact model and particle body forces. CFD computes the motion of the fluid flow using the incompressible Navier-Stokes equations. The CFD part of the simulation was carried out in the commercial CFD software ANSYS Fluent 17.2, which is two-way coupled to the commercial DEM software EDEM. The renormalization-group (RNG) k-ε turbulence model and the multiphase flow model Volume of Fluid (VOF) were used to simulate the free surface turbulent flow in the CFD code. After connecting the coupling server, the Lagrangian Discrete Phase Model (DPM) in Fluent was enabled and the EDEM-particles were modelled as DPM injections. The collision forces between particles and particle-wall were determined using the Hertz-Mindlin collision law that is the default model used in EDEM due to its accuracy and efficiency. The DEM particles were modelled using a three-spherical shape. The shape of particles plays an important role in sediment transport; however, it is neglected in the most theoretical approaches and most CFD codes. The models used are well described in the documentation of the software (EDEM 2.6 Theory Reference Guide 2014; Ansys Fluent Theory Guide 2016). The geometry consisting of an inlet tank and a pipe was subdivided into a mesh of hexahedral elements with a size of 10 mm using the Cutcell method. The PISO algorithm was selected for the pressure-velocity coupling and the

PRESTO! discretization for the pressure. The Second Order Upwind discretization was used for momentum, turbulent kinetic energy and dissipation rate. The air-water interface was tracked using the Geometric Reconstruction scheme.

Experimental work

To validate the CFD-DEM model, a series of experiments have been conducted. A set-up of an acryl-glass pipe in a closed system was constructed under laboratory conditions (Figure 1). Three different pipe surface roughness sizes were investigated by gluing sand paper on the bottom of the pipe. Sand and gravels of four different size distributions with a specific density of 2.65 were used as sediments.

In the first series of experiments, a steady uniform flow condition was adjusted. Sediments were put at the bottom of the pipe in a 500 mm measurement section positioned at 3,200 mm from the pipe inlet. The flow discharge and thus the flow velocity were increased slightly until the incipient motion of sediment transport occurred. The incipient motion was defined as the *general* movement introduced by Kramer (1935) via visual observation. *General* movement occurs when larger grains from a sediment bed start to move, slowly changing the bed configuration. The values of discharge were obtained using a MID flowmeter, and a compact echo sounder was used to determine the water level in the pipe. The bed slope was set to 0.17%. In total, 60 experiments were conducted; 12 different scenarios (combination of four different sediment sizes and three different pipe roughness sizes) with five repeating. All experiments were video recorded using a digital camera. The shape of the front of the sediment bed was observed regarding the deforming of the bed configuration.

In the second series of experiments, sediment transport under flush waves was considered. A sluice gate was installed to realize flush waves, which are similar to a dam-break event. Experiments were conducted by varying the storage height behind the sluice gate, pipe surface roughness and the amount of sediments initially deposited on the bottom behind the sluice gate. To assist the experiment analysis, two high-speed cameras were positioned at different angles to record the flow in the test stretch. In addition, scoured sediments after each flush were dried and weighed. The velocity of the wave, the water level and the scouring efficiency were obtained and used to validate the numerical model. The experimental variables are summarized in Table 1.

RESULTS

After the CFD model was validated with experimental data without sediments, it was coupled to the DEM solver. The CFD-DEM model was then validated with experimental data with the presence of sediments. Qualitatively as well as quantitatively, good agreement between experimental and numerical results confirmed that the CFD-DEM is a good tool to study detailed features of sediment transport in steady flow conditions and under flush waves. In the following, some results obtained from the numerical model are presented.

Steady flow condition

Figure 2 shows the wall shear distribution of a section with sediment ($h_w/D = 0.14$ & $h_s/D = 0.02$, h_w and h_s being the

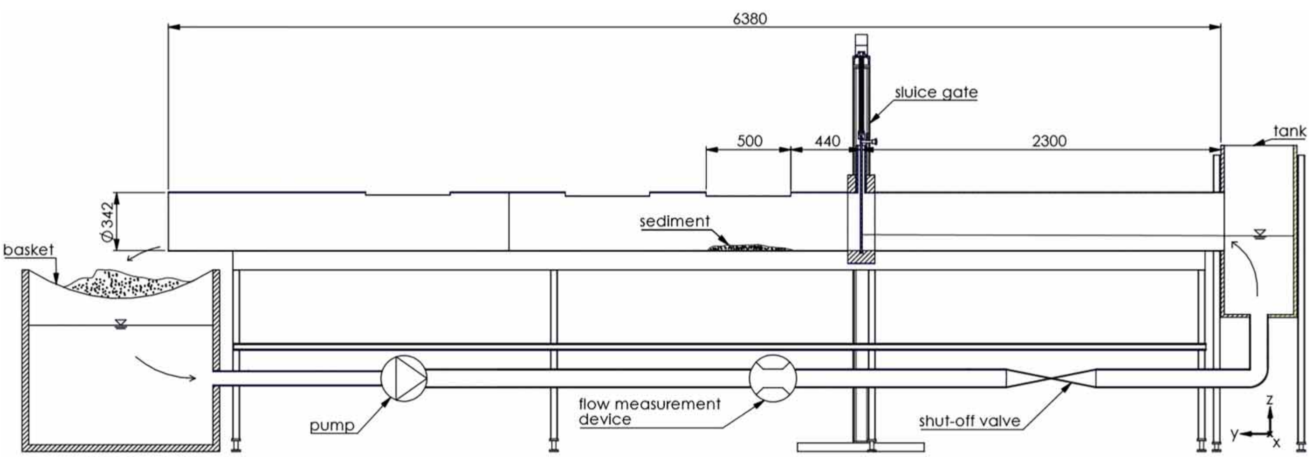

Figure 1 | Sketch of the laboratory test stand and the investigated sediment bed for the unsteady flow experiment (in mm).

Table 1 | Experimental parameters

	Steady flow	Flush waves
Pipe roughness k (mm)	0, 0.2, 0.5	0, 0.5
Sediment median size d_{50} (mm)	0.74, 1.58, 2.33, 4.33	mixture
Sediment amount m (g)	500	1,500, 3,000
Storage height h (mm)	–	170, 350

water height and sediment bed height, respectively) compared to a section without sediment. It can be seen that in the presence of a sediment bed in a circular flume, the maximum wall shear occurred at two symmetrical points of the walls away from the centre. This is in accordance with type b presented by Kleijwegt (1992), where different types in shear stress distributions have been observed. The wall shear in the centre of the flume reached its minimum value in the area of sediments. Most particles in this area (in the bottom layers of the sediment bed) have zero velocity and do not move. In laboratory experiments, it was also observed that sediments were mostly transported from the two side walls of the bed and/or from the surface of the sediment bed. Furthermore, from Figure 2 can be concluded that the presence of a sediment bed leads to a decrease of shear stress in the central bottom up to 70%. The wall shear in a circular section without sediments reached its maximum value in the centre. This is in accordance with numerical findings of Berlamont *et al.* (2003), who also concluded that the presence of a sediment bed has a significant effect on the shear stress distribution. In addition, Figure 3 shows that the maximum sediment velocity is only 0.05 while the mean flow velocity is 0.43 m/s. In an experimental study on particle velocity at limit deposition in sewers (Ota & Perrusquia 2011), it was concluded, that the particle velocity even for the fastest moving particle (glass spheres), on a clean fixed bed is as low as half of the mean flow velocity. Figure 4 shows the comparison between the longitudinal velocity in a section with and without sediment. In earlier experimental works (Ambrose 1953; May *et al.* 1989), it was observed that the presence of small depths of deposits in a pipe increases the sediment transport rate compared to that in clean pipes. Ab Ghani (1993) also concluded that for pipe with sediment deposits, a lower transporting velocity is required than those for clean pipes.

This can be explained by the results obtained from the current numerical research. The presence of sediments in the pipe induces an increase of flow velocity in the free surface region and also an increased roughness on the surface

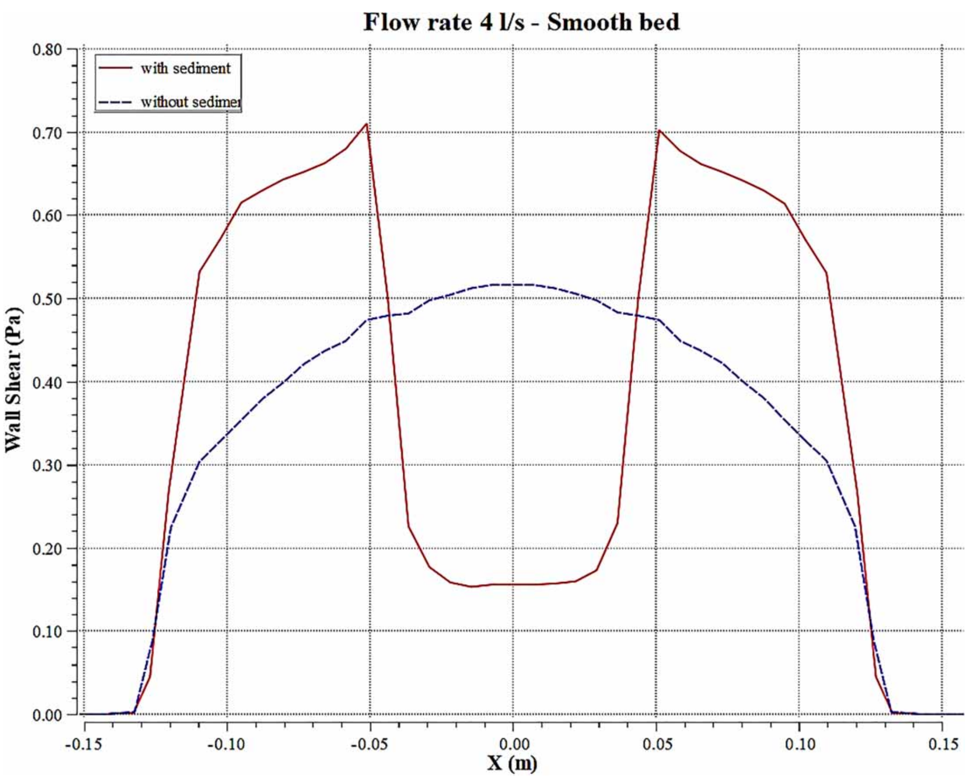

Figure 2 | Shear stress distribution in a section with and without sediment for the case $k = 0$, $d_{50} = 4.33$ mm and a flow rate of 4 l/s.

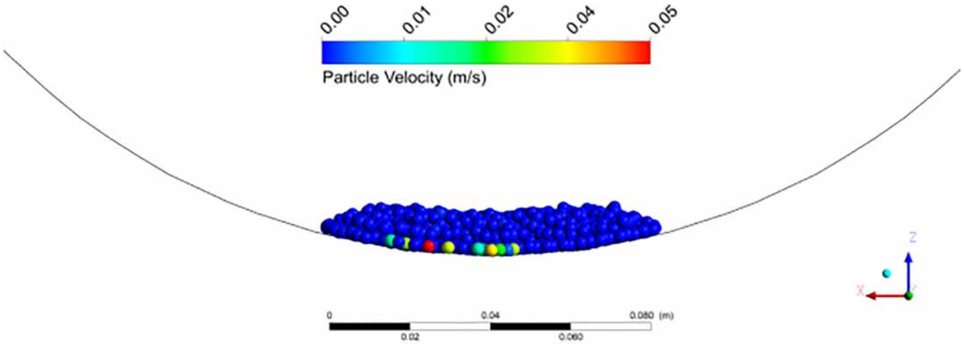

Figure 3 | Particle velocities for the case $k = 0$, $d_{50} = 4.33$ mm and a mean flow velocity of 0.43 m/s.

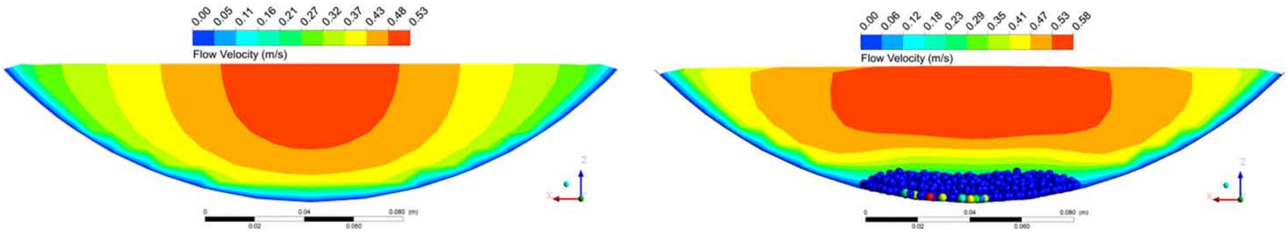

Figure 4 | Longitudinal flow velocities in a cross section without sediment (left) and in a cross section with sediment (right) for the case $k = 0$, $d_{50} = 4.33$ mm and a flow rate of 4 l/s.

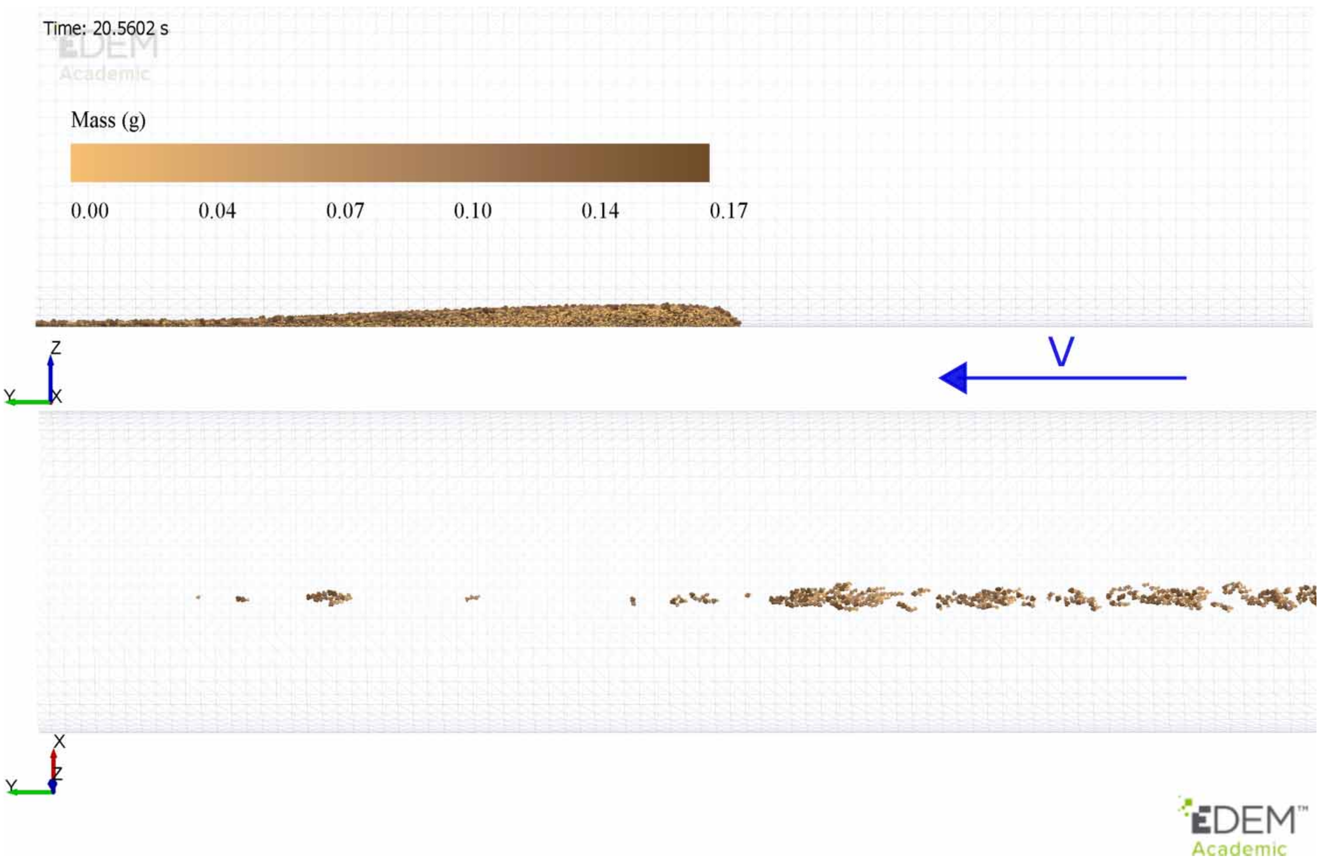

Figure 5 | Side view of the sediment bed (above) and top view of the scattered sediment at the end of the pipe (down) after the flush in DEM model for the case $k = 0$, $m = 1{,}500$ g, $h = 170$ mm.

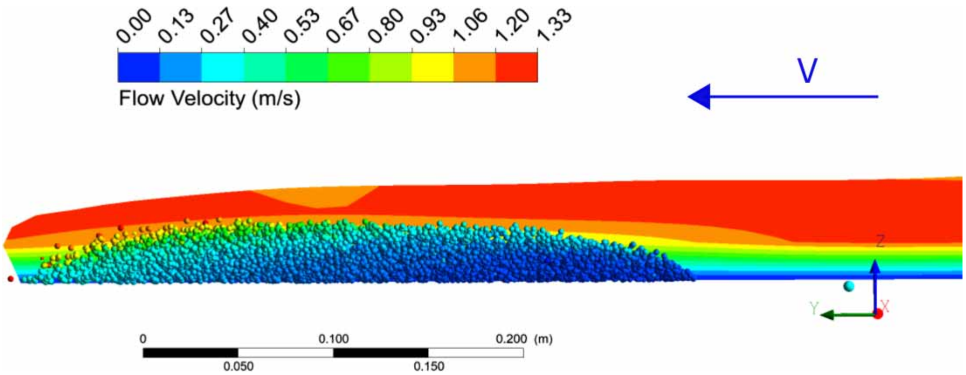

Figure 6 | Velocity of the wave on the middle section plane for the case $k = 0$, $m = 1{,}500$ g, $h = 170$ mm.

of the sediment bed. Consequently, the shear stress exerted from the flow on the surface of the sediment bed increases. Therefore, sediments will be transported from the top layer of the sediment bed. However, in the near wall region, more turbulence or lateral movements are available due to the presence of sediments, which reduces the flow velocity and the shear stresses in this area. As a result, particles in the bottom layers of a sediment bed could start to move when sediments from the top layer are already detached and removed.

Unsteady flow condition

Figure 5 shows the evolution of the sediment bed in the CFD-DEM model. In all cases, after the first flush, the flat sediment bed decreased in height and increased in length. Some sediment was found as single clumps of solids at the end of the channel. The same shape of sediment bed after a flush was also observed in the experimental work. Moreover, the results of the field experiments of Ristenpart (1998) with real wastewater and the results of the laboratory experiments of Campisano *et al.* (2004) with clear water and sand showed the same fashion. From Figure 5, it can be furthermore observed that after the first flush most of the solids that remained on the bottom are fine/light particles. This was also verified in experimental results. The reason is, among others, the smaller contact surfaces of fine particles in interacting with waves and the so called 'hiding effect' of larger particles, which prevent the smaller particles being transported by the flow.

Another observation can be made regarding the velocity of the solids under the wave. Figures 6 and 7 show the flow velocity on the middle plane and the particle velocities. Unlike the steady flow condition, here the fastest particle has almost the same maximum velocity as the wave. It can also be seen that the solids, which started to move with the wave, were mostly from the top layer as well as the front of the bed. Solids moved partly as bed-load and partly in suspension.

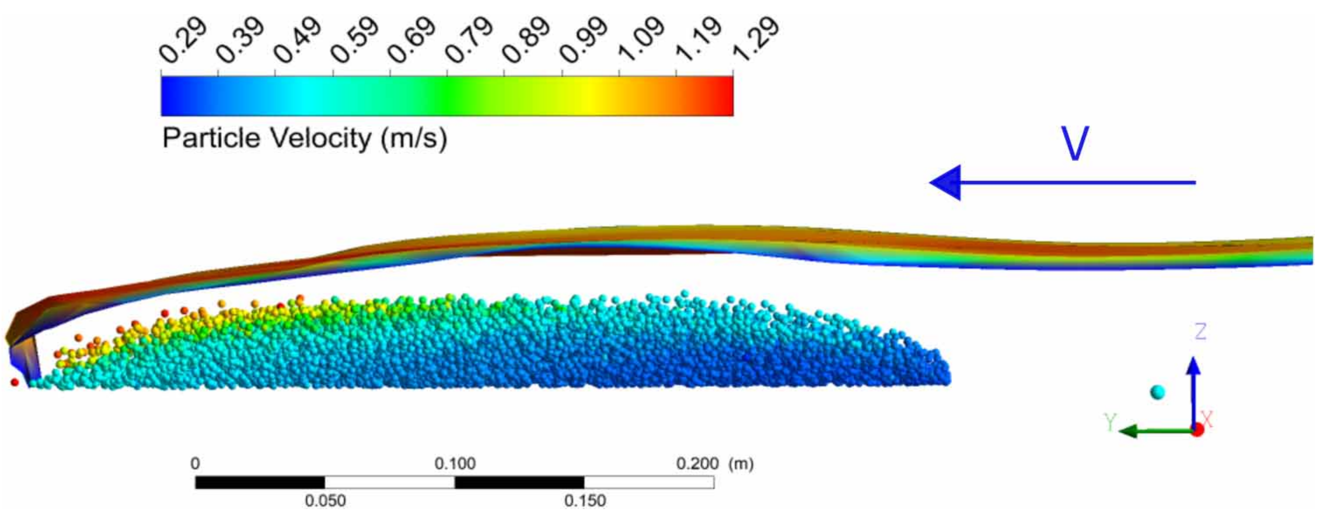

Figure 7 | Average particle velocity under flush wave on a smooth bed for the case $k = 0$, $m = 1{,}500$ g, $h = 170$ mm.

CONCLUSIONS

The proposed CFD-DEM coupling method is a powerful tool to study various features of sediment transport in detail. It can partly capture the basic behaviour of sediments in steady as well as unsteady flow. However, it should be noted that the three-dimensional modelling is computationally expensive and time-consuming when calculating a system with thousands or several millions of particles. For example, the coupling simulation with 1,500 and 3,000 g sediments resulting in 28,879 and 57,513 particles took about 3:30–4 hours and 5–6 hours for 1 s simulation time, respectively on a workstation PC with 8 CPU cores. Using new computational technologies such as GPUs and CUDA (Compute Unified Device Architecture) will speed up simulations significantly (Jajcevic *et al.* 2013). This research can be seen as a preliminary study toward 3D modelling of sediment transport in sewer systems. Further investigations should be carried out in order to evaluate the capability of the CFD-DEM model in reproducing the sediment transport in real sewer systems. The model should be calibrated and validated based on further experimental work as well as *in-situ* measurements.

REFERENCES

Ab Ghani, A. 1993 *Sediment Transport in Sewers*. Doctoral thesis, University of Newcastle, Newcastle Upon Tyne.

Alihosseini, M. & Thamsen, P. U. 2016 An overview of SPH simulation and experimental investigation of sediment flows in sewer flushing. *Technical Transactions, Environment Engineering* **113** (1), 17–28.

Alihosseini, M. & Thamsen, P. U. 2018 Experimental and numerical investigation of sediment transport in sewers. In: *Proceedings of the ASME 2018, 5th Joint US-European Fluids Engineering Division Summer Meeting (FEDSM 2018), Quebec, Canada*.

Alihosseini, M. & Thamsen, P. U. 2019 On scouring efficiency of flush waves in sewers – a numerical and experimental study. In: *Submitted to the ASME-JSME-KSME 2019, Joint Fluids Engineering Conference, AJKFLUIDs2019, San Francisco, CA, USA*.

Ambrose, H. 1953 The transportation of sand in pipes – free surface flow. In: *5th Hydraulic Conference*. State University of Iowa Studies in Engineering, Ames, Iowa, USA, pp. 77–88.

Ansys Fluent Theory Guide 2016 *Release 17.2. Southpointe*. ANSYS, Inc., Canonsburg, PA, USA.

Ashley, R., Bertrand-Krajewski, J.-L., Hvitved-Jacobsen, T. & Verbranck, M. 2004 *Solids in Sewers*. IWA Publishing, London, UK.

Berlamont, J., Trouw, K. & Luyckx, G. 2003 Shear stress distribution in partially filled pipes. *Journal of Hydraulic Engineering* **129**, 697–705.

Bonakdari, H., Larrarte, F. & Joannis, C. 2008 Study of the shear stress in narrow channels: application to sewers. *Urban Water Journal* **5** (1), 15–20.

Bonakdari, H., Ebtehaj, I. & Azimi, H. 2015 Numerical analysis of sediment transport in sewer pipe. *International Journal of Engineering (IJE)* **28**, 1564–1570.

Bravo-Blanco, A., Sanchez-Medina, A. & Ayuga, F. 2017 Analysis of the incipient motion of spherical particles in an open channel bed, using a coupled computational fluid dynamics-discrete element method model. *Biosystems Engineering* **155**, 68–76.

Campisano, A., Creaco, E. & Modica, C. 2004 Experimental and numerical analysis of the scouring effects of flushing waves on sediment deposits. *Journal of Hydrology* **299**, 324–334.

Cundall, P. & Strack, O. 1979 A discrete numerical model for granular assemblies. *Geotechnique* **29** (1), 47–65.

EDEM 2.6 Theory Reference Guide 2014 DEM Solutions, Edinburgh, UK.

He, C., Marsalek, J. & Rochfort, Q. 2004 Numerical modelling of enhancing suspended solids removal in a CSO facility. *Water Quality Research Journal of Canada* **39**, 457–465.

Jajcevic, D., Siegmann, E., Radeke, C. & Khinast, J. 2013 Large-scale CFD-DEM simulations of fluidized granular system. *Chemical Engineering Science* **98**, 298–310.

Kleijwegt, R. 1992 *On Sediment Transport in Circular Sewers with non-Cohesive Deposits*. Doctoral thesis, Technische Universiteit Delft, Delft, The Netherlands.

Kramer, H. 1935 Sand mixtures and sand movement in fluvial models. *American Society of Civil Engineers* **100**, 798–838.

May, R., Brown, P., Hare, G. & Jones, K. 1989 *Self-cleansing Conditions for Sewers Carrying Sediment*. Hydraulic Research Ltd, Wallingford, UK.

Ota, J. & Perrusquia, G. 2011 Particle velocity and sediment transport at limit deposition in sewers. In: *12th International Conference on Urban Drainage, Porto Alegre, Brazil*.

Ristenpart, E. 1998 Solids transport by flushing of combined sewers. *Water Science and Technology* **37** (1), 171–178.

Schmeeckle, M. 2014 Numerical simulation of turbulence and sediment transport of medium sand. *Journal of Geophysical Research: Earth Surface* **119**, 1240–1262.

Stovin, V., Saul, A., Drinkwater, A. & Clifforde, I. 1999 Field testing CFD-based predictions of storage chamber gross solids separation efficiency. *Water Science and Technology* **39** (9), 161–168.

Sun, R. & Xiao, H. 2016 Sedifoam: a general-purpose, open-source CFD-DEM solver for particle-laden flow with emphasis on sediment transport. *Computers & Geosciences* **89**, 207–219.

Vetsch, D. 2012 *Numerical Simulation of Sediment Transport with Meshfree Methods*. Doctoral thesis, ETH Zurich.

Vongvisessomjai, N., Tingsanchali, T. & Babel, M. 2010 Non-deposition design criteria for sewers with part-full flow. *Urban Water Journal* **7** (1), 61–77.

First received 25 October 2019; accepted in revised form 14 January 2020. Available online 29 January 2020