**Proceedings 10th NUMGE 2023**
10th European Conference on Numerical Methods in Geotechnical Engineering
Zdravkovic L, Kontoe S, Taborda DMG, Tsiampousi A (eds)


https://doi.org/10.53243/NUMGE2023-107

# Numerical investigation of backward erosion piping

H. Kaiser[1], J. Grabe[1]

*[1] Institute of Geotechnical Engineering and Construction Management, Hamburg University of Technology, Hamburg, Germany*

**ABSTRACT:** Backward erosion is a failure mechanism for water retaining structures such as dikes, dams or measuring weirs. In this process, all grain fractions of a soil are transported by a water flow in a path (pipe) which develops against the direction of flow. This process cannot be determined and therefore is not considered in the safety calculation of constructions. For a better understanding of the erosion process, the numerical investigation by CFD-DEM coupling offers continuously advancing possibilities due to increasing computing power. The soil particles are modelled by the Discrete Element Method (DEM) and the fluid flow by the Computational Fluid Dynamics (CFD) which in a next step are coupled. As the first sub-process of backward erosion, suffusion, i.e., the transport and rearrangement of smaller particles through the void space of a grain skeleton of larger particles, is investigated in the present work. For this purpose, the unresolved CFD-DEM coupling was used. To compare the numerical simulation to real life behaviour, the experiment was replicated and examined using computational tomography (CT).

**Keywords:** CFD-DEM simulation; erosion; backward erosion, piping, suffusion

## 1 INTRODUCTION

Water-retaining structures such as dikes, dams, sheet piles and measuring weirs are severely stressed by the increasing frequency of flood events. In addition to the main risk of water overtopping, other mechanisms exist, most notably erosion (the transport and rearrangement of soil particles by water flow), which can lead to the failure of such structures. Since all failure mechanisms have to be considered in the design of these structures, a basic understanding of material transport in soil by water flow is important.

Backward erosion is a special form of internal erosion which cannot be computed. Therefore, there is no verification procedure and the possible damage is commonly prevented constructively by means of filters or by interruption of the flow (Bundesanstalt für Wasserbau, 2013). Consequently, the process of backward erosion is usually only assumed to be the cause of damage retrospectively.

The material transport takes place in tubular "cavities" (channels or pipes). The process is called backward erosion because the pipes form against the flow direction of the water. In this process, erosion begins at the downstream end of the structure and spreads against the direction of flow in a pipe in the soil, between the soil and the structure or at the layer boundary. During backward erosion, different processes take place. First, small particles move in very small pipes (suffusion). When the channel reaches the upstream end, the widening process starts and when this in turn reaches the upstream end, the material transport rates and thus the flow velocity increase. This leads to the failure of the dike (van Beek, 2015).

Numerical modelling of such processes bears great potential due to increasing computing power. Structural safety verifications of structures (e. g. dikes) are based on continuum mechanics, with so-called material laws that reflect the soil behaviour. However, soil behaviour is very complex and, among other things, the particle-fluid interaction cannot be determined with this approach. Multiscale (length scale) approaches such as CFD-DEM couplings enable the numerical modelling of particle-fluid systems at the particle level. Here, the particles are modelled as discrete particles using the Discrete Element Method (DEM) and the fluid is modelled using Computational Fluid Dynamics (CFD) which are then coupled together. The simulations in this paper were carried out with the software ASPHERIX® provided by the company DCS Computing GmbH from Linz in Austria. ASHPERIX® is a software for the simulation of discrete elements using DEM. For the CFD-DEM coupling, ASPHERIX® is coupled via the extension CFDEM®coupling with the open source software for CFD OpenFOAM®, which was published in 2004 by OpenCFD Ltd.

## 2 CFD-DEM COUPLING

In the numerical simulation of particle-fluid systems, one speaks of dispersed two-phase flow, in which one phase in the form of particles, droplets or bubbles is finely distributed in another continuous phase. For such a model, two numerical approaches are combined, each describing one material (Sommerfeld, 2017). The fluid flow is modelled with the cell-based continuum approach and the movement of the particles is modelled by discrete particles, additionally fluid-particle interactions are considered. The coupled CFD-DEM method was first introduced by Tsuji et al. (1993), but is a relatively new investigation method in geotechnical engineering and is therefore briefly explained in the following section.

## *2.1 DEM*

Modelling soil particles with DEM is based on the approach of Cundall and Strack (1979), whose main elements are the particles, the walls and the contact models. During the simulation, individual particles can move freely in the simulation domain and are bounded by walls (structures/geometries). For each particle, different information such as mass, velocity and the forces acting on it are calculated.

The motion of the individual particles is tracked using the Lagrangian approach. An iterative solution algorithm is used for the motion behaviour and the force action of the particles, in which each time segment consists of two sub-steps. First, all forces acting on the individual particles are calculated, which include the particle-particle interaction, particle-wall interaction, acceleration due to gravity, and external forces. This results in a force for each contact of a particle, which arises from the relative velocity of the elements and the overlap of the particles. Using Newton's 2nd law, the resulting translational and rotational velocity of the particles are calculated for the next time step.

In geotechnics, the soft-sphere approach is used for the contact forces between the particles, as this is suitable for densely bedded particle systems, like soil. The different contact models vary in computational efficiency as well as in physical accuracy and are chosen depending on the particle shape, material properties and state of motion. The most common contact models are the linear model of Cundall and Hart (1992) and the non-linear model based on Hertz and Mindlin (Langston et al.,1994).

## *2.2 CFD-DEM Coupling*

Coupled CFD-DEM simulations can be achieved using either resolved or unresolved coupling. The differences between the two models depend on the CFD grid cell. If it is significantly larger than the particle size, it is unresolved CFD-DEM, while if it is resolved CFD-DEM, the CFD cell is smaller than the particles. Unresolved CFD-DEM permits a simulation of large particle numbers, whereby the flow around the individual particles is not solved here because the flow velocity in the CFD cells is averaged and the resulting fluid (drag) force is distributed evenly among the particles within this CFD cell. Therefore, force models are necessary in the calculation of the interaction between particles and fluid to account for the averaging of the flow velocity in the application (Crowe et al., 1998). For research in geotechnics the interaction of particles and water needs to be studied on a larger scale therefore unresolved CFD-DEM simulations are a viable option if the interaction laws of the system are sufficiently accurate. The effect of the soil particles on the water and vice versa is considered, which means that no additional models need to be implemented for the reaction of the soil structure to the fluid.

Computational Fluid Dynamics (CFD) allows the analysis of the flow of a continuous fluid using different discretization techniques. In the finite volume method (FVM), the solution domain is discretized and the governing equations for each cell of the mesh are solved. These are the continuity equation and the Reynolds-Averaged Navier-Stokes equation (RANS). The RANS equation is a simplification of the Navier-Stokes equation for the numerical approximation of turbulent flows therefore it is the time-averaged Navier-Stokes equation by turbulence models. The RANS equation in the CFD-DEM coupling is contribution by Anderson and Jackson (1967). To account for the particles in the fluid, two variables are added to the continuity equation of the fluid. The void fraction $\alpha_f$ describes the volume fraction of the fluid in a cell while the momentum exchange term $R_{pf}$ represents the interaction between particles and fluid. The exchange term between CFD and DEM $R_{pf}$ is based on the relative velocity between fluid and particle and an interaction force coefficient from the superposition of all particle fluid interaction forces. The conservation of mass and momentum of a fluid with particles must still be given.

The exchange of data, between the two methods, takes place after several time steps, so that the CFD and DEM are calculated parallel and the data is exchanged depending on the coupling interval. In each coupling interval, the particle data (velocity, position, radius) of the DEM are passed to the CFD model, and at the same time the forces acting on the particles are transferred from the CFD to the DEM. The system is then uncoupeld so that the DEM can calculate new positions and velocities with the forces acting on the particles, while the CFD uses the particle data from the data exchange to determine new approximations for the velocity and pressure of the fluid, from which the force of the fluid is calculated, Three types of forces act on each particle due to the fluid-particle interaction: pressure-induced, friction-induced and contact-induced forces. The pressure-induced force results from the pressure gradient around the particle. The pressure difference at the top and bottom of the particle results in a buoyancy force in the direction of the pressure gradient. The friction-induced force can be calculated using the shear line tensor. The forces induced by the fluid can be divided into drag and lift. Drag is the dominant force in particle-fluid interaction and the main cause of fluidisation effects in flow-through particle-fluid systems (Zhu et al., 2007). It arises from the resistance of a particle to fluid flow and depends mainly on the particle shape and Reynolds number.

## 3 NUMERICAL SET-UP

Backward erosion is a very complex process and consists of several phases. Van Beek (2015) described backward erosion through four phases: 1. Single particles are transported and form a preferential flow path (pipe), 2. Boiling phase, in which the sand liquefies, 3. Sand particles are transported to the opening a pipe begins to dorm and 4. Backward erosion continues and the pipe develops against the direction of flow. Suffusion as an easy case of an erosion problem is suitable for an initial numerical investigation, as here only smaller particles move through the pore space of the rigid particle skeleton of larger particles (Bundesanstalt für Wasserbau, 2013). Accordingly, only one particle size moves through the sample container, like in phase 1.

The setup is based on an experimental setup that was carried out in a micro CT. The simulation consists of two steps of sample preparation first in the dry state and then the flow through the sample.

The geometric suffusion criterion of Kenney and Lau (1985) assumes that a grain of diameter d moves through a pore of particles of diameter 4d. In the following experiments, the diameter of the small particles is $d = 0.25$ mm. According to Kenney and Lau (1985), a particle diameter of the particle skeleton with a diameter of at least $4d = 1$ mm is required for the small particles to move through the water flow. The particle diameters of 0.25 mm and 2 mm were chosen and are within the size range of sand.
The material properties of the spheres correspond to those of monodisperse SiLibeads glass spheres made of soda-lime glass from SIGMUND LINDNER GmbH with the Hertz-Mindlin model used as the contact model. The material parameters and contact model parameters for the particles and the sample container (wall) can be found in Tables 1 and 2.

### 3.1 Procedure

The cylindrical sample container, which is made of plastic in the experimental tests, has an inner diameter and a height of 12 mm. In a first pure DEM simulation (dry state) the sample is prepared. First, 0.3 g of the large particles are generated in a lower layer of the sample. Then 0.25 g of the small particles are generated (Figure 1 top left) and sunk into the void spaces of the large particles (Figure 1 top right). Finally, the sample container is filled with 1.5 g of the large particles (Figure 1 bottom left). The lid presses the particles downwards so that at the end of the sample production there are approximately 12400 particles in a cylinder (Figure 1 bottom right). The time step for the DEM is chosen to be 5 microseconds, which is due to the high young's modulus. After each step of particles generation, the next step will only continuewhen the particles have settled.

In the second step, the suffusion is triggered by a water flow with a velocity of 0.002 m/s from the bottom to the top through the sample container for 1 s by using a coupled CFD-DEM simulation, during which 226 mm³ of water flow through the sample.

For the interaction with the DEM the drag force is transferred to the particles by the fluid using the Koch-Hill approach. Based on the PisoFoam, a finite volume based solver for the turbulent Navier-Stokes equation (RANS) using the PISO algorithm, the cfdem-SolverPiso has additional functionalities for the coupling with the DEM code of ASPHERIX®. The RANS equations are solved considering the momentum exchange and volume displacement of discrete particles whose trajectories are calculated in the DEM code (Goniva, 2010). The time step of the DEM remains at 5 microseconds as in the previous simulation. The CFD simulation, however, runs faster with 1 microsecond time steps. Every 200 DEM steps, the forces are exchanged between CFD and DEM.

*Table 1: Interaction parameters between particles and wall for the contact Model*

| **Particle-Wall-Interaction Parameter** | |
|---|---|
| **Resitution coefficient** | 0.7 |
| **Friction coefficient** | 0.4 |

*Table 2: Material and contact model properties of the spheres particles and the sample Container*

| **Parameter** | **Particles** | **Wall** |
|---|---|---|
| **Density [kg/m³]** | 2500 | 1390 |
| **Youngs modulus [GPa]** | 65 | 0.065 |
| **Poisson ratio** | 0.3 | 0.3 |
| **Resitution coefficient** | 0.6 | 1 |
| **Friction coefficient** | 0.3 | 1 |

## 4 RESULTS

Figure 2 shows the initial (left) and final (right) state of the numerical simulation from all sides. It can be clearly seen that the small particles (blue) have migrated upwards as a result of the flow through the void space of the large particles (red). Especially in the 3rd and 4th row of the figure, the small particles have migrated to the upper half of the sample. Even in the initial state, the movement are formed. Based on openings in the particle across the cross-section. After flowing through, it is clearly visible that preferred paths for particle distribution of the small particles is not homogeneous geometry and along flow paths, the small particles continue to migrate upwards, while the way upwards is blocked by densely packed large particles. In contrast, a change in the positions of the large particles after a water flow is not observable.

In figure 3 in the cross-section 0.003 m above the sample bottom, a clear increase in small particles can be observed. The green line in the three-dimensional view (left) marks the height of the cross-section. The pore space in the upper right corner, which was previously only filled with water, is now completely filled with small particles. In the cross-section 0.0045 m above the sample bottom, the last upper small particles in the pore space can be seen in the initial state. When looking at the cross-section after the flow, it can be clearly seen that more small particles have migrated upwards across the cross-section. These are not evenly distributed over the entire cross-section, but mainly in the upper left-hand corner (see Figure 4). Especially in the large gap between the particles in the upper area, small ones have been added, which may gathered favoured by the pore geometry.

## 5 CONCLUSIONS

The numerical investigation of the suffusion was realised using the unresolved CFD-DEM coupling. Only the sample preparation and a flow through the sample from bottom to top were modelled. An inhomogeneous distribution of the small particles by the flow over the cross-section can be seen. Preferred paths are formed along which several small particles migrate upwards through the void space. Ideal spheres have been used to simplify the numerical simulation. For CFD density, pore fraction and drag are the main factors determining the behaviour of the pressure drop. For non-ideal spheres, the drag force has a different effect than on real particles.

For the investigation of backward erosion, the phases of initiation and formation of the pipes are of interest. The initiation always starts with a suffusion, which is why further investigation of this process is important. For future numerical simulations, it is possible to take the initial positions of the particles from the CT images of real samples. This will allow a better investigation of the correspondence between the experiments and the numerical modelling. For the observation of suffusion, a solid particle skeleton can be produced in a 3D printer and only small particles could be incorporated in order to also obtain exact particle positions of the particle pellet for the numerical simulation. In addition, the coupling of resolved and unresolved CFD-DEM coupling offers further possibilities in the numerical modelling of erosion processes. In areas where the material transport can be localised, the particle fluid interaction can be investigated more precisely by means of resolved CFD-DEM coupling.

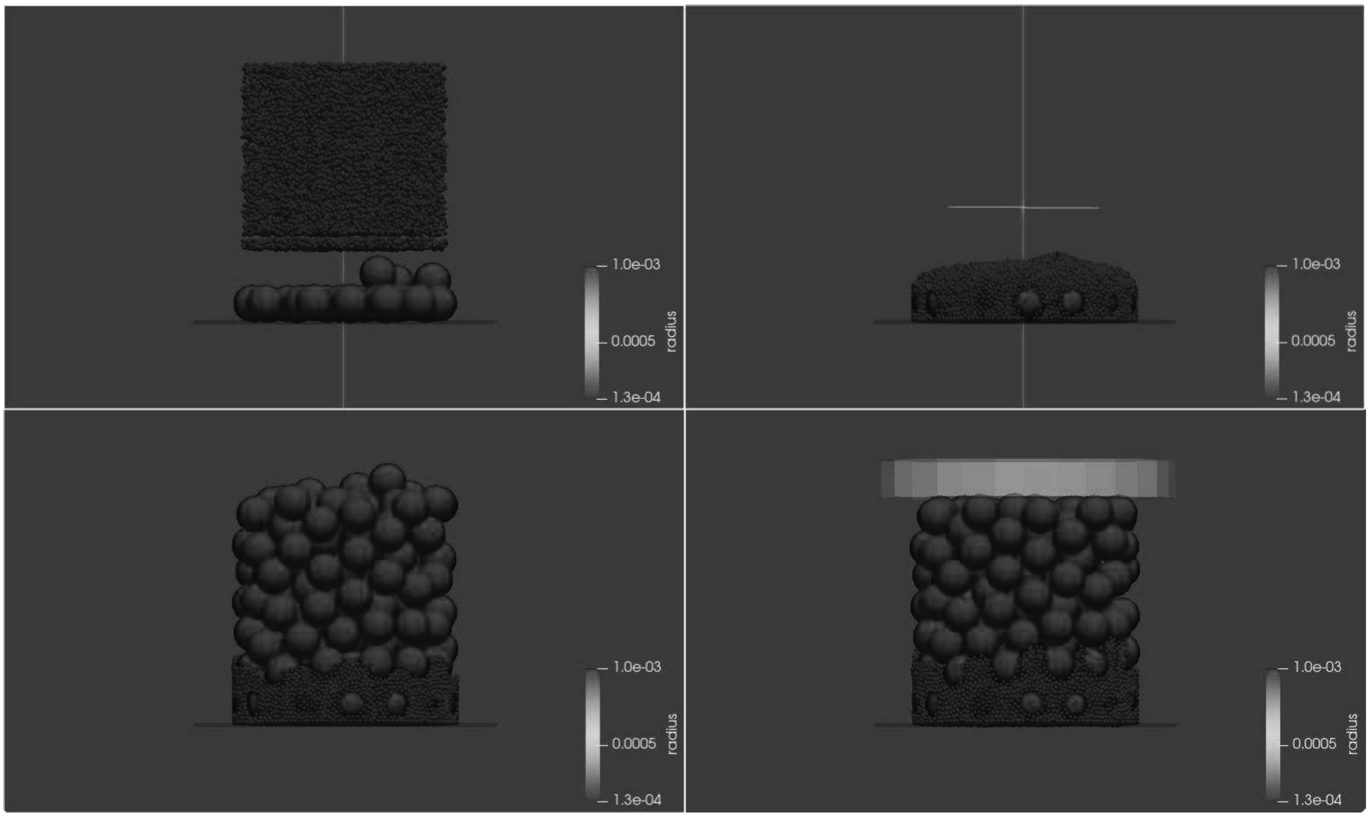

*Figure 1. DEM Sample preparation in 4 steps.*

In case of numerical investigations of erosion processes on particle level with the multiscale (length scale) approaches CFD-DEM and Lattice Boltzmann Method (LBM)-DEM offer enormous potential. Increased computational power allows for more detailed and elaborate models that can be used to analyse larger areas with larger numbers of particles. In addition to a better understanding of erosion processes, the knowledge gained in this way can also potentially be used to construct structures with an erosion risk more efficiently in real life.

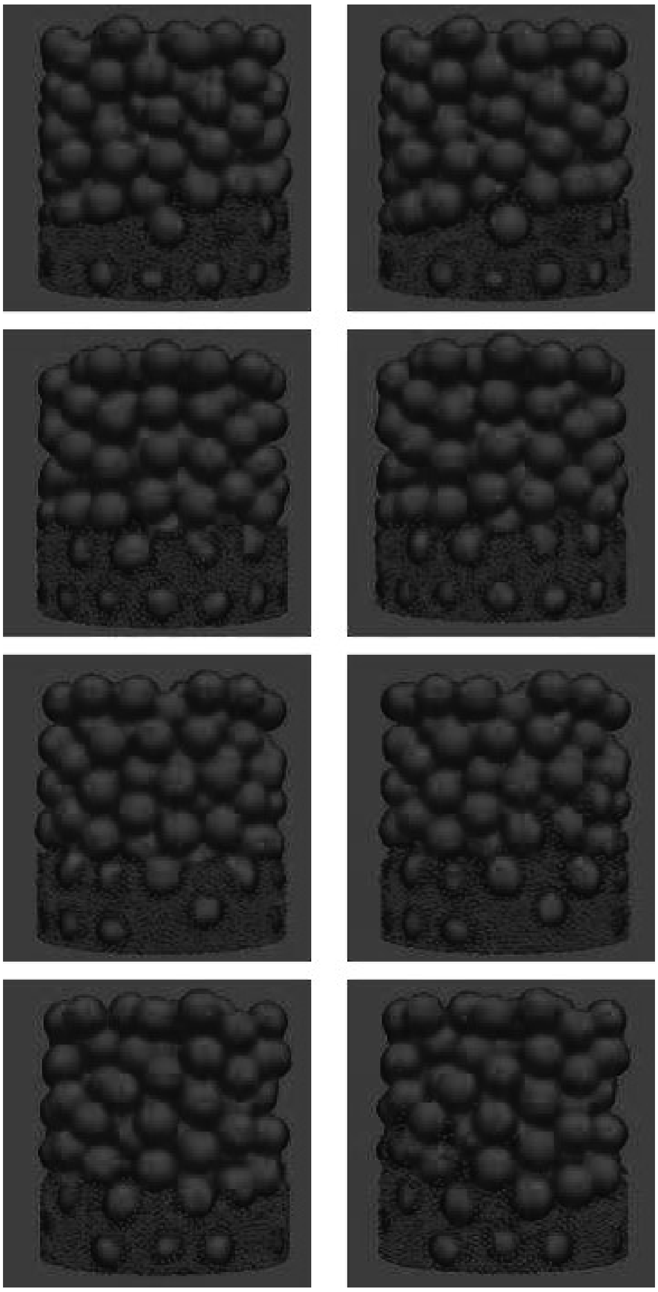

*Figure 2. Initial (left) and (right) state of the simulation from all sides.*

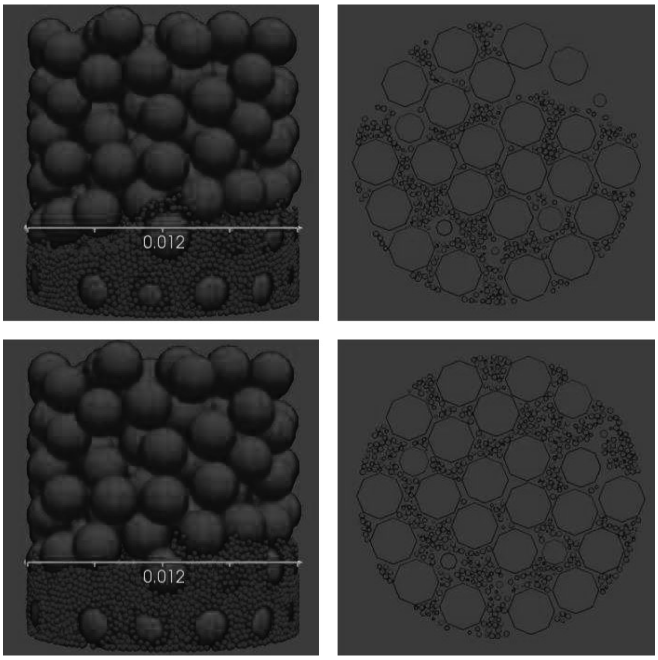


*Figure 3. Comparison of initial (top) and final (bottom) state with Cross-section view at a height of 0.003 m above the sample floor.*

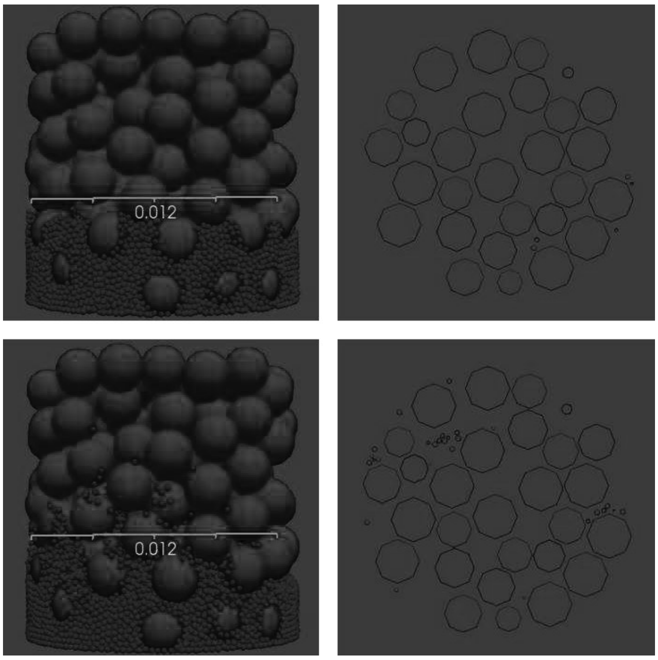


*Figure 3. Comparison of initial (top) and final (bottom) state with Cross-section view at a height of 0.0045 m above the sample floor.*

## 6 ACKNOWLEDGEMENTS

The authors acknowledge the funding of this research by the German Research Foundation (Deutsche Forschungsgemeinschaft, DFG) in the framework of Research Training Group GRK 2462: Processes in natural and technical Particle-Fluid-Systems at Hamburg University of Technology (TUHH).

## 7 REFERENCES


Anderson, T. B. and Jackson, R. (1967). Fluid mechanical description of fluidized beds. Equations of motion. Industrial and Engineering Chemistry Fundamentals, 6(4):527– 539.

Bundesanstalt für Wasserbau (2013). BAWMerkblatt Materialtransport im Boden (MMB).

Crowe, C., Sommerfeld, M., Tsuji, Y., und Crowe, C. (1998). Multiphase Flows with Droplets and Particles CRC. Boca Raton, FL.

Cundall, P. A. and Hart, R. D. (1992). Numerical modelling of discontinua. Engineering computations.

Cundall, P. A. und Strack, O. D. L. (1979). A discrete numerical model for granular assemblies. geotechnique, 29(1):47–65.

Goniva, C. Kloss, C. Hager, A. Pirker, S.(2010) An Open Source CFD-DEM Perspective, In: 5th OpenFOAM Workshop, Chalmers, Gothenburg, Sweden, June 21-24, 2010

Kenney, T. C. and Lau, D. (1985). Internal stability of granular filters. Canadian geotechnical journal, 22(2):215–225.

Langston, P. A., U. Tüzün, & D. M. Heyes (1994). Continuous potential discrete particle simulations of stress and velocity fields in hoppers: Transition from fluid to granular flow. Chemical Engineering Science 49(8), 1259–1275.

Sommerfeld, M. (2017). Numerical Methods for Dispersed Multiphase Flows. In Particles in Flows, Seiten 327–396. Springer International Publishing, ChamTsuji et al. 1993

Van Beek, V. M. (2015). Backward erosion piping: initiation and progression. Dissertation, Technische Universiteit Delft