CFD-DEM Onset of Motion Analysis for Application to Bed Scour Risk Assessment

March 2017

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free via DOE's SciTech Connect (<http://www.osti.gov/scitech/>)

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):

U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

CFD-DEM Onset of Motion Analysis for Application to Bed Scour Risk Assessment

by

M.A. Sitek, S.A. Lottes

Transportation Research and Analysis Computing Center (TRACC)

Nuclear Engineering Division, Argonne National Laboratory

March 2017

Table of Contents

1. Introduction and research objectives	1
2. Discrete Element Method overview	1
3. Sediment transport.....	3
3.1. Incipient movement of particles	3
3.2. Forces acting on a DEM Particle on a smooth surface	4
3.3. Forces acting on a DEM Particle on a rough surface.....	5
4. Flume with a Particle Tray	7
4.1. Simulation phases.....	8
4.2. Establishing critical shear stress	9
4.3. Rectangular vs. trapezoidal tray	13
5. Model of a scoured smooth bed around a cylindrical pier	15
6. Decay function.....	16
6.1. Comparison of computational and experimental results	16
6.2. Influence of friction coefficients	26
7. Conclusions	29
8. References	30

List of Figures

Figure 2-1: Example shapes of composite particles.....	2
Figure 2-2: Contact force components between two particles according to Hertz-Mindlin model2	
Figure 3-1: Forces acting on a particle submerged in moving water.....	5
Figure 3-2: A pyramid made of particles laying on a horizontal bed with forces acting on the top particle	6
Figure 3-3: Forces acting on particles in two-dimensional calculations.....	6
Figure 3-4: Two possible directions of motion of the top sphere, a) over one of the bottom spheres, b) over two bottom spheres	6
Figure 3-5: A 3D side view of a 4-sphere pyramid. Two possible directions of motion of the top sphere with marked angles and contact points between spheres.....	7
Figure 4-1: The computational domain, a) naming convention, b) dimensions, c) boundary conditions	8
Figure 4-2: Consecutive time steps at the incipient motion of the first particle	10
Figure 4-3: Shear stress along vertical lines on the symmetry plane.....	11
Figure 4-4: Shear stress component τ_{xz} distribution on the bottom surface (smooth wall) and at the interface between the tray and the flume	11
Figure 4-5: Change of the isosurface for Volume Fraction of Particle = 0.29 during the simulation	12
Figure 4-6: Dimensional Shields' diagram. Comparison of Guo's correlation and CFD-DEM results.....	13
Figure 4-7: A general view of the a) rectangular and b) trapezoidal trays filled with particles	14
Figure 5-1: Computational domain with smooth bed surface.	15
Figure 5-2: Phases of a spherical particle motion on a smooth bed.....	15
Figure 6-1: The side view of the computational domain for 'decay function' studies, a) fully scoured, and partially scoured with maximum flow height: b) 0.145 m, c) 0.125 m, d) 0.105 m, e) 0.0085 m with corresponding flat-bed contour models.	19
Figure 6-2: An example domain with the bottom tray filled with particles: a) partially scoured bed and b) a corresponding flat bed	20

Figure 6-3: The side view of the experimental domain for ‘decay function’ studies, a) fully scoured, and partially scoured with maximum flow height: b) 0.145 m, c) 0.125 m, d) 0.105 m, e) 0.0085 m with corresponding flatbed contour models.	22
Figure 6-4: Comparison of velocities: inlet velocity v_1 , and maximum velocities in the scoured bed and flat bed models. Computational bed geometry.	23
Figure 6-5: Comparison of velocities: inlet velocity v_1 , and maximum velocities in the scoured bed and flat bed models. Experimental bed geometry.	24
Figure 6-6: Onset of motion particle density vs. relative depth of a scour hole around a cylindrical pier for computational bed contour.	25
Figure 6-7: Onset of motion particle density vs. relative depth of a scour hole around a cylindrical pier for experimental bed contour.	25
Figure 6-8: Linear velocity of a particle with varying coefficient of rolling resistance	26
Figure 6-9: Linear velocity of a particle after it touches the bed ($t=0.045$ sec, see Figure 6-8) ..	27
Figure 6-10: Drag force on a particle after it touches the bed ($t=0.045$ sec, see Figure 6-8)	27
Figure 6-11: Linear velocity of a particle with varying friction coefficient	28
Figure 6-12: Angular velocity of a particle with varying friction coefficient	28
Figure 6-13: Drag force on a particle with varying friction coefficient.	29

List of Tables

Table 1: Results of the test cases	13
--	----

1. Introduction and research objectives

This CFD study with DEM was done as a part of the Federal Highway Administration's (FHWA's) effort to improve scour design procedures.

The Computational Fluid Dynamics-Discrete Element Method (CFD-DEM) model, available in CD-Adapco's StarCCM+ software, was used to simulate multiphase systems, mainly those which combine fluids and solids. In this method the motion of discrete solids is accounted for by DEM, which applies Newton's laws of motion to every particle. The flow of the fluid is determined by the local averaged Navier–Stokes equations that can be solved using the traditional CFD approach. The interactions between the fluid phase and solids phase are modeled by use of Newton's third law. The inter-particle contact forces are included in the equations of motion. Soft-particle formulation is used, which allows particles to overlap.

In this study DEM was used to model separate sediment grains and spherical particles laying on the bed with the aim to analyze their movement due to flow conditions. Critical shear stress causing the incipient movement of the sediment was established and compared to the available experimental data. An example of scour around a cylindrical pier is considered. Various depths of the scoured bed and flow conditions were taken into account to gain a better understanding of the erosion forces existing around bridge foundations. The decay of these forces with increasing scour depth was quantified with a 'decay function', which shows that particles become increasingly less likely to be set in motion by flow forces as a scour hole increases in depth. Computational and experimental examples of the scoured bed around a cylindrical pier are presented.

2. Discrete Element Method overview

CFD-DEM is one of the computational methods that can be used to model multiphase flows. A multiphase flow is a flow where various phases (gas and liquid or solid) exist in the same domain, e.g., a mix of fluids of different density, gas bubbles or solid particles in a liquid, granular materials like sandy soils in water or air, etc. DEM is based on a Lagrangian formulation, which solves the equation of motion for representative parcels as they move in space and time. DEM is typically used for simulating granular flows because it provides the capability of efficiently modeling separate particles and their interactions. In DEM, individual particle parcel properties are tracked. These properties include shape, size, material properties, and state properties, such as velocity. The DEM solver solves for inter-particle and particle-boundary contact forces. It also solves for particle-fluid forces (drag).

The DEM module in Star-CCM+ solves coupled problems within the software. It gives a wide range of modeling features. The properties of solid particles and boundaries are defined as for elastic bodies, with specified density, elastic modulus, and Poisson's ratio. Particle shapes may differ from single spherical particles to more complex composite particles, representing any required geometry. Example shapes of these particles are shown in Figure 2-1. The particles are introduced to the domain by point, surface, or volume injection. The software offers several options for random or structured injection.

Figure 2-1: Example shapes of composite particles

Figure 2-2: Contact force components between two particles according to Hertz-Mindlin model

The particle movement in the flow can be of sliding and/or rolling type. The constitutive behavior tangential to the particle in contact is determined by the slip model. It is defined by the friction coefficient μ between the two contacting parts (particle-to-particle, or particle-to-wall).

Slip between particles occurs only if the following condition is satisfied:

$$T > \mu N \quad (1)$$

Components of contact forces between two particles in Hertz-Mindlin model are illustrated in Figure 2-2. The soft contact formulation was used, which allows for a particle overlap.

The conservation law of angular momentum for a single particle describes the rolling motion of a particle

$$Tr + M = I \frac{\partial \omega}{\partial t}, \quad (2)$$

where r – particle’s radius, M – a rotational moment acting on the particle’s centroid, I – polar moment of inertia, ω – angular velocity.

Rolling starts, when

$$|M| > \alpha r N, \quad (3)$$

where α is the rolling friction coefficient, $0 < \alpha < 1$. The physical meaning of α is analogous to the inter-particle friction coefficient μ , and it controls how much rolling resistance can be generated at any contact point.

Modeling of attractive inter-particle forces is possible in StarCCM+ with the use of the parallel bonds model. The bonds between particles are created at collision and, if desired, can be breakable when a failure model is used.

Two-way coupling between the particles and the flow was also enabled. This way the Lagrangian momentum source is added to the continuous phase. When the particle and fluid velocities are not equal, then drag forces act on both fluid and particles, so in addition to the fluid flow acting to accelerate/decelerate the particles, the particles also accelerate/decelerate the fluid. For the case of a bed of particles, this two-way coupling allows a sufficiently fast flow over a packed bed of particles to set the particles on the surface in motion and be carried into the downstream. It also allows the particle bed to act as a rough wall and cause a vertical velocity distribution to develop at and above the bed with the flow velocity very close to zero just below the surface of the bed that increases with the height above the bed. Thus a boundary shear layer is created in the fluid flowing above the bed, and the shear stress at any distance at and above the bed can be calculated.

3. Sediment transport

3.1. Incipient movement of particles

The flow conditions necessary to cause movement of the sediment are most commonly characterized by critical shear stress. The critical shear stress is defined as the minimum value of shear stress acting on the bed, at which particle incipient motion occurs. The relationship between the stress and sand grain size was first established experimentally by Shields [1] in 1936 in form of a graph of non-dimensional boundary shear stress and a particle’s Reynold’s number. Later, in 1990, Guo presented a formula between the Shield’s critical number and dimensionless particle diameter:

$$\tau_* = \frac{0.23}{d_*} + 0.054 \left[1 - \exp \left(-\frac{d_*^{0.85}}{23} \right) \right], \quad (4)$$

where $d_* = d \left[\frac{(s-1)g}{\nu^2} \right]^{1/3}$ is the dimensionless particle diameter, g is the gravitational acceleration, s is the specific gravity of the sediment, d is the diameter of the sediment, and ν is the kinematic viscosity of water.

Formula (4) is used in the present study in its dimensional form as a reference for the computational results. The dimensionless Shields critical number can be transformed to critical shear stress with the relation:

$$\tau = \tau_*(s-1)\rho_w g d, \quad (5)$$

where ρ_w is the density of water.

3.2. Forces acting on a DEM Particle on a smooth surface

On a smooth surface a sphere has only one point of contact with the bed and the predominant type of motion is rolling. In moving water, the only forces acting on the sphere are gravity and drag. This means that, when subjected to the flow drag, the sphere will move for any drag force greater than zero on a flat surface because there is no rolling resistance. On an upward slope, the resistance comes from the component of gravity force that is parallel to the surface. Figure 3-1 shows forces acting on a particle: drag force and gravity. Gravitational force F_G can be represented by two components: parallel to the surface $F_G^{\parallel}$ and perpendicular to the surface $F_G^{\perp}$. If the slope is at the angle α , then the motion will occur if

$$F_G^{\parallel} < F_D, \quad (6)$$

$$F_G^{\parallel} = V\rho g \cos \alpha, F_D = 0.5c_D\rho_w v^2 A, \quad (7)$$

where r is the radius of the spherical particle, $V = \frac{4}{3}\pi r^3$ is the volume of the sphere, $A = \pi r^2$ is cross section area of a sphere, $\rho = \rho_m - \rho_w$ is the density of the material decreased by density of water, g is the gravitational acceleration, $c_D = 0.5$ is the drag coefficient [2], and v is the flow velocity.

Figure 3-1: Forces acting on a particle submerged in moving water

3.3. Forces acting on a DEM Particle on a rough surface

Consider a bed made of several layers of spherical particles. The particles in the top layer are those that will be transported first by the flow. One way to know if movement occurs, is to establish the balance of forces acting on a particle [3]. Figure 3-2 shows four example particles stacked in a pyramid, laying on a horizontal bed with forces acting on the top particle.

Two types of calculations, 2D and 3D were considered, with the assumption that the bed is horizontal and all particles have the same radius. The two-dimensional case is illustrated in Figure 3-3 and the three-dimensional in Figure 3-5.

Forces acting on a particle are the weight of the particle, F_G , buoyancy, F_B and drag, F_D . The first two forces are applied at the sphere's center of gravity. The distribution of the fluid forces on a particle was assumed to be unknown because in the DEM model particle surfaces are not resolved in the mesh. Therefore it is impossible to establish the location of F_D . Friction prevents sliding of one particle past another and the moving particle pivots about an axis normal to the flow direction at point O . For the motion to occur, the moments tending to rotate the particle downstream have to be higher than the moments that tend to hold the grain in place:

$$a_1(F_G - F_B) \sin \alpha < a_2 F_D \quad (8)$$

where: a_1 , a_2 are normal distances to a line through the point of contact from the center of gravity and the point of application of F_D respectively ($a_1 = r$, a_2 is unknown, but in the range from r to $2r$), α is the angle between the direction of the movement and flat bed.

Angle α can be established from the geometry. In the 2D case it is equal 30 deg. In the 3D case we should consider two limit cases, showed in Figure 3-5 and Figure 3-5, where the angles equal to 35.3 deg and 19.5 deg, respectively. In the latter, the movement will start earlier, because the left side of the equation [3] is smaller.

Figure 3-2: A pyramid made of particles laying on a horizontal bed with forces acting on the top particle

Figure 3-3: Forces acting on particles in two-dimensional calculations

Figure 3-4: Two possible directions of motion of the top sphere, a) over one of the bottom spheres, b) over two bottom spheres

Figure 3-5: A 3D side view of a 4-sphere pyramid. Two possible directions of motion of the top sphere with marked angles and contact points between spheres

4. Flume with a Particle Tray

A model of a flume with a small tray filled with sediment particles was created. The model was used to assess the capabilities of the software, computational time needed to run a simulation, and to establish how reliable the results are by comparing them to data found in literature.

The geometry represents a slice down the center of a very wide flume, as seen in Figure 4-1, to simplify the simulation. Appropriate symmetry boundary conditions were assigned to the side walls of the domain, as well as to the top surface. The other boundary conditions consist of a velocity inlet, pressure outlet, and non-slip wall on the bottom surface. A periodic, fully-developed interface between the inlet and outlet with set mass flow rate was created. Initial condition of uniform velocity was assigned. In between the tray and flume there was a porous baffle interface with a high porosity (equal to 0.9). An Unsteady RANS (URANS) with $k-\epsilon$ turbulence model was used. Wall shear stress was computed with an all Y^+ wall treatment. In this treatment standard wall functions are used when Y^+ is greater than 30, shear stress is computed from the definition when Y^+ is less than 5, and a blending function is used for Y^+ between these values. The model was built of about 26600 hexahedral cells.

The DEM phase consists of spherical particles with 1mm, 2mm, and 4mm diameter. The material properties of the particles correspond to those of sand and are as follows: density 2650 kg/m³, Poisson coefficient 0.35, Young's modulus 1 GPa. Particle-particle contact was defined with Hertz-Mindlin contact model. The static friction coefficient equals 0.6, and normal and tangential

restitution coefficients are equal to 0.01. Rolling resistance was set to 0.01. An additional run with higher density equal to 7850 kg/m³ was also conducted.

The tray region was filled with particles with the use of a random injector. The ‘porosity limit’ type of particle packing specification was chosen. This type of particle injection to fill the tray provides for the possibility of setting the particle diameter and required porosity. With porosity equal to 0.6 the whole region was tightly packed with particles. The number of particles differ depending on their diameter. For the particle size 4mm, the number of particles equals 2242.

Figure 4-1: The computational domain, a) naming convention, b) dimensions, c) boundary conditions

4.1. Simulation phases

A DEM simulation was divided into three phases: flow development, particle settlement, and particle movement. In the first phase, only the flow solver is active. The initial velocity and inlet velocity are applied. Cyclic boundary conditions are employed, to reduce the domain dimensions and therefore to achieve a fully-developed flow in a shorter time. In the particle settlement phase,

the DEM solver is activated and a preselected number of spherical particles is injected into the tray. The porous baffle between the tray and flume is active, so that the particles don't leave the region. In the last phase, the injector is deactivated, the porous baffle is removed and the particles are free to move. The simulations were run until the particles started to move due to the flow or until most of them left the tray, depending on the requirements.

Two-way coupling model was used in the simulations. This model indicates that the Lagrangian phase can exchange mass, momentum, and energy with the continuous phase, depending on the active Lagrangian phase models. This model better represents the interaction between phases than the one-way model. In the one-way model, the particles don't influence on flow. That model is appropriate for very sparse particle flows, and it could not model the effect of the packed particle acting as a rough wall.

4.2. Establishing critical shear stress

The goal of this study was to establish the moment of incipient motion of the sediment and the corresponding critical shear stress at the surface of the particle bed. The moment of incipient motion was assumed to happen when the centroid of the first particle reaches the top of the tray and continues to move with the flow. The approach used assumes that the incipient motion of the sediment happens when the centroid of the first particle crosses the interface between the tray and the flume and the particle in question continues to move in the flow. Figure 4-2 combines three images showing the moment before the first particle reaches the interface, and when it crosses the interface.

A component of the shear stress tensor, τ_{xz} , is monitored at interface between the flume and the tray. Its surface average at the time at the time of incipient motion is taken to be the critical stress.

The shear stress is calculated in STAR-CCM+ as a field function according to the formula:

$$\tau_{xz} = (\mu_T + \mu_D) \frac{\partial u_x}{\partial z}, \quad (9)$$

where: μ_T and μ_D are the turbulent eddy and dynamic viscosity respectively, u_x is the velocity in x direction, z is the vertical coordinate.

The velocity gradient is obtained from STAR-CCM+ internal calculations, by checking a 'retain internal derivatives' option. The values of shear stress are stored for the interface between the tray and flume regions and also along five vertical probe lines on the symmetry plane of the domain.

A parametric study was performed, with three particle diameters: 1 mm, 2 mm, 4 mm, and various inlet velocity, see the test matrix in Table 1.

Figure 4-2: Consecutive time steps at the incipient motion of the first particle

Figure 4-3: Shear stress along vertical lines on the symmetry plane

Figure 4-4: Shear stress component τ_{xz} distribution on the bottom surface (smooth wall) and at the interface between the tray and the flume

Figure 4-3 presents the calculated values of shear stress along five vertical probe lines located on the symmetry plane of the domain for the case with 4 mm particles and an inlet velocity of 2 m/s.

The graph shows that the shear stress, calculated according to (9), has zero value at the bottom of the tray, reaches maximum at the interface and then decreases back to zero on the top surface.

Figure 4-4 illustrates the distribution of the shear stress at the interface between the tray and flume regions. The shear stress component τ_{xz} is higher in the upstream region. The roughness changes there from a smooth wall to the test section filled with particles. The values of at the interface vary from about 0.8 Pa to 4.5 Pa, with a surface average of 3.4 Pa. The average is a good approximation of the value calculated from (4), which is equal 3.3 Pa. It is worth noticing at this point that another simulation was run, with one way coupling. The average shear stress in this case was equal to 7.2 Pa, which confirms the need for two-way coupling between flow and DEM particles in the model.

The average shear stress for 4mm particles' test cases was also calculated on an isosurface of 'Volume Fraction of Particles' equal 0.29. The Volume Fraction of Particles is a parameter describing what percentage of cell volume is occupied by particles. The value was chosen such that the isosurface crosses most of the upper layer particles. Particles from this layer will be the first to start moving in the flow, therefore the shear stress should be monitored here as well. This isosurface changes location and shape during the simulation as the particles move and leave the tray, which is presented in Figure 4-5.

Figure 4-5: Change of the isosurface for Volume Fraction of Particle = 0.29 during the simulation

Figure 4-6: Dimensional Shields' diagram. Comparison of Guo's correlation and CFD-DEM results.

Table 1: Results of the test cases

Diameter [mm]	Inlet velocity [m/s]	Density [kg/m ³]	Incipient motion?	Critical shear stress [Pa]		
				Guo's formula	at the interface	at the isosurface
1	1	2650	yes	0.59	0.7	n.a.
2	1		yes	1.3	1.03	n.a.
4	0.5		no	3.3	-	-
	1		yes		2.4	1.4
	2		yes		5.6	2.3
		7850	yes	n.a.	6.5	2.9

The results collected in Table 1 show a change in the average critical shear stress due to particle diameter and density and location of the monitored surface. The critical shear stress increases with increasing diameter. It is also higher for higher densities. The values calculated on the interface are higher in every case than the ones obtained on the isosurface and they are closer to Guo's results. A comparison of Guo's correlation and CFD-DEM results is presented in Figure 4-6.

4.3. Rectangular vs. trapezoidal tray

A different shape of the tray was also considered for the test case of 4 mm particles and inlet velocity 2 m/s. The rectangular tray was replaced with a trapezoidal region. A general view of the rectangular and trapezoidal trays filled with particles is showed in Figure 4-7. In this set up, there

are no straight angles and corners where the particles could get trapped. The slopes of the new tray allow them to follow the flow more easily.

The first particle leaves the tray sooner than in the previous simulation (8th time step, before it was 27th time step). The corresponding critical shear stress has a lower value: on the interface the average equals to 4.78 Pa and on the isosurface 1.7 Pa (comparing to 5.6 Pa and 2.3 Pa respectively).

Figure 4-7: A general view of the a) rectangular and b) trapezoidal trays filled with particles

5. Model of a scoured smooth bed around a cylindrical pier

The next example considered was a scoured smooth bed around a cylindrical pier. The geometry of a partially scoured bed was derived in a computational procedure developed by researchers at TRACC, Northern Illinois University, and FHWA [4]. The bed around the pier was modeled as a smooth no-slip wall boundary without roughness. Two particles are injected into the domain, one on each side of the cylindrical pier. Their interaction with the flow was monitored during the simulation.

The CFD domain is illustrated in Figure 5-1. The inlet velocity in this simulation equals 0.5 m/s. The bed material properties correspond to 0.5 mm quartz sediment. The bed is partially scoured with maximum shear stress equal 0.93 Pa.

Figure 5-1: Computational domain with smooth bed surface.

Two DEM particles were injected in the scour hole close to the pier. The particles were allowed to fall on the bottom due to gravitational loading and then they were subjected to the flow. The particles were 4 mm in diameter and had the density of quartz, 2650 kg/m³. The friction coefficient was set to 0.6 and rolling friction resistance to 0.01. Guo's formula predicts the critical shear stress for this diameter of sediment on a flatbed to be equal to 3.3 Pa, which is much higher than the existing maximum value of 0.93 Pa in the scoured bed profile. If this was the only criterion for onset of particle motion, the particles wouldn't move with the flow. The observed behavior was different, the particle starts to move with the flow and then leaves the scour hole. The reason for this is that the DEM particle is a perfectly smooth sphere and the bed is a smooth surface. A single point interaction between the particle and the surface exists, there is no traction between them, and rolling motion of a particle occurs. The forces acting on the sphere are presented in equation (6). Figure 5-2 shows three phases of motion of two spherical particles on the smooth bed.

Figure 5-2: Phases of a spherical particle motion on a smooth bed

This model doesn't represent well the conditions present on a river bed. There is a need to add roughness to the bed surface. The approach taken in the following simulations comes closer to sandy river bed conditions by adding a rough bed region around the pier. It was filled with DEM particles bonded together, which were heavy enough as a whole, to stay stationary throughout the analysis.

6. Decay function

As a part of the FHWA research on scour design procedure methodology, an attempt was made to quantify the relation between the flow conditions and the particle characteristics in the form of a 'decay function'. The concept of the 'decay function' is based on the assumption that particles become increasingly less likely to be set in motion by flow forces as a scour hole increases in depth.

6.1. Comparison of computational and experimental results

An example of the flow around a cylindrical pier was selected. Two bed surface geometries are considered: (1) a scoured bed obtained from CFD computations [4] and (2) a surface derived from flume experiments conducted at the FHWA Turner-Fairbank Hydraulics Laboratory. In the first case, the computational domain of a partially scoured bed was chosen as a base case. The flow depth was varied by cutting off bottom part of the scour hole, thus creating a partially flat bed at different depths. In the second case, the geometry of the bed was obtained at four stages of scour during the experiment. Side views of the bed around the pier are shown in Figure 6-1 for case (1), and in Figure 6-3 for case (2).

The roughness of the bed was modeled by introducing tightly packed DEM particles to form a stationary 5 mm top layer in the locations of interest around the pier. The particles forming the bed had the density of steel, were 4 mm in diameter, and were joined together with unbreakable bonds. The flow around the pier was analyzed and the point of maximum velocity magnitude just above the newly created bed surface was identified. A separate DEM phase was injected above the surface. Particles that belong to this phase have the same diameter as the particles forming the bed. Their density was increased to establish a limit value at which the particle stops moving after being dropped on to the bed and is not set in motion by the flow. The value of the lowest density of a particle that stays on the bed was recorded for different scour hole depths. The relationship between these two values was plotted and represents the 'decay function'.

An effort was made to find a relationship between maximum velocity in the vicinity of the pier in the scoured bed simulations and flat-bed models with the same approach flow to simplify the scour analysis to studying flat-bed cases with appropriate flow conditions. This approach reduces the computational effort and makes the analysis more time efficient.

The depth of an un-scoured bed is equal to H_0 and the inlet velocity is v_0 . The maximum depth in the partially scoured domain i is equal to H_i . Assuming that the mass flow rate is the same, the inlet velocity in a flat bed case with flow height H_i , must be equal $v_i = \frac{H_0 v_0}{H_i}$. Figure 6-1 illustrates

the side view of the computational domain for 'decay function' studies, a) fully scoured (depth of the scour hole is d_{max}), and partially scoured (depth d_i) with maximum flow height: b) 0.145 m, c) 0.125 m, d) 0.105 m, e) 0.0085 m with corresponding flat-bed contour models. Figure 6-2 shows an example domain with the bottom tray filled with particles: a) a partially scoured bed and b) a corresponding flat bed. Figure 6-3 shows the side view of the experimental domain for 'decay function' studies, a) fully scoured and partially scoured with maximum flow height: b) 0.145 m, c) 0.125 m, d) 0.105 m, e) 0.0085 m with corresponding flat-bed contour models.

A uniform inlet velocity was applied as the inlet velocity boundary condition in case (1). In case (2), a fully developed inlet flow was assigned. The maximum velocity magnitude for the computational bed surface simulations was located in the deepest part close to the pier, independently of the scour hole depth. In the models representing the experimental geometry, this location varies. The bed surface was covered with 'sand dunes', whose tops are the locations of maximum velocity. As the location of these dunes changes constantly, the point of maximum velocity magnitude differs as well. It was not always recorded at the deepest point of the bed. Nevertheless, it was assumed that the height of the corresponding flat-bed simulations were equal to the flow height at the deepest point in the scour hole.

Figure 6-1: The side view of the computational domain for 'decay function' studies, a) fully scoured, and partially scoured with maximum flow height: b) 0.145 m, c) 0.125 m, d) 0.105 m, e) 0.0085 m with corresponding flat-bed contour models.

a)

b)

Figure 6-2: An example domain with the bottom tray filled with particles: a) partially scoured bed and b) a corresponding flat bed

Figure 6-3: The side view of the experimental domain for 'decay function' studies, a) fully scoured, and partially scoured with maximum flow height: b) 0.145 m, c) 0.125 m, d) 0.105 m, e) 0.0085 m with corresponding flatbed contour models.

Figure 6-4 shows a comparison of velocities: inlet velocity v_1 and maximum velocities in the pier vicinity in scour bed and flatbed models, at different relative depths, for computational geometry. The relative depth, d_i/d_{max} , (d_i is the depth at level i , $d_i = H_i - H_0$, d_{max} is the maximum depth) was introduced to enable a comparison between different cases, in which the maximum scour depth may be different. Figure 6-5 shows these velocities for the experimental geometry. The maximum velocity in the flat bed vs. depth relationship can be well approximated with a logarithmic function. A logarithmic trendline was plotted for this in both figures. The relationship for the scour hole models follows the same trend up to a certain depth. The trend changes when the conditions get closer to the equilibrium scour depth. Then the maximum velocity decreases rapidly. It is hard to say at what depth exactly this change starts for the experimental geometry, as only a set of four depths of partially and fully scoured bed were available. It is known that it happens for relative depth equal or greater than 0.7.

Figure 6-4: Comparison of velocities: inlet velocity v_1 , and maximum velocities in the scoured bed and flat bed models. Computational bed geometry.

Figure 6-5: Comparison of velocities: inlet velocity v_1 , and maximum velocities in the scoured bed and flat bed models. Experimental bed geometry.

When the maximum velocity is known for a specific geometry and flow conditions, a limit density of a particle can be estimated from the relation (8). Minimum and maximum values were calculated by taking extreme values of the normal distance a_2 , which ranges from r to $2r$. The limit density obtained from internal Star-CCM+ computations was compared to the calculated values. It was very close to the minimum calculated density, being higher by approximately 11% to 19%. A relationship between the onset of motion particle density and relative depth for the bed contour obtained from scour computation is shown in Figure 6-6.

Figure 6-7 shows a graph of the onset of motion particle density vs. relative depth for the experimental bed contour. The minimum and maximum particle densities were calculated. The location of the maximum velocity near the bed was changing from one scour level to another and usually was not at the deepest point of the scour hole, but on top of one of many sand dunes forming on the bed, as mentioned before. The relationship between this maximum velocity and the maximum relative depth of the scour hole has the same character as the curves obtained for the computational bed. For the scoured bed models, at lower depths the curves follow a logarithmic function. At larger depths, the density value drops. In the flat bed simulations, both curves, meaning lower and upper limit, resemble logarithmic functions. DEM-CFD analysis with moving particles was not performed for this bed contour.

Figure 6-6: Onset of motion particle density vs. relative depth of a scour hole around a cylindrical pier for computational bed contour.

Figure 6-7: Onset of motion particle density vs. relative depth of a scour hole around a cylindrical pier for experimental bed contour

6.2. Influence of friction coefficients

Influence of rolling resistance and linear friction coefficients was studied. A particle was dropped onto a flat horizontal bed formed with spherical particles. Different values of rolling resistance coefficient, $\alpha=0.01, 0.3, 0.7, 0.9$, were assigned to it. In the second set of simulations, the linear friction coefficient was varied from 0.6 to 1.2. The flow conditions were kept the same in all simulations. Particle velocity, after it touches the bed, as well as drag, were recorded.

Figure 6-8 shows the relation between linear velocity of a particle in time with varying coefficient of rolling resistance. The first part of the plot, when the particle is injected above the surface and falls down onto the bed, is the same for all cases. Differences occur, when the particle touches the bed. For smaller values of CRR the particle starts to roll on the bed, moving from crevice to crevice (the behavior is indicated by the jumps in the velocity). For higher values of the coefficient, the particle falls into a crevice and stays there, shielded by other particles from the flow.

Figure 6-9 presents a plot of linear velocity of a particle at the time step when it touches the bed ($t=0.045$ sec, see Figure 6-8) vs. CRR. When the $CRR=0.001$, the velocity equals to 0.146 m/s, whereas for $CRR=0.9$, it drops to 0.046 m/s. Figure 6-10 illustrates the drag force value after the particle touches the bed ($t=0.045$ sec, see Figure 6-8). Due to higher resistance, the drag force acting on the particle increases. Drag increases from $3.8 \cdot 10^{-4}$ N to $5.6 \cdot 10^{-4}$ N due to the change of CRR from 0.001 to 0.9

Figure 6-8: Linear velocity of a particle with varying coefficient of rolling resistance

Figure 6-9: Linear velocity of a particle after it touches the bed ($t=0.045$ sec, see Figure 6-8)

Figure 6-10: Drag force on a particle after it touches the bed ($t=0.045$ sec, see Figure 6-8)

Changes in linear friction coefficient, $\mu=0.6, 1.2$, have a different influence on the particle motion. Linear velocity of a particle with varying friction coefficient is presented in Figure 6-11 and its angular velocity, in Figure 6-12. The linear velocity doesn't vary significantly, in turn the angular velocity drops from 900 rpm ($\mu=0.6$) to 100 rpm ($\mu=1.2$). Drag also decreases, from $6.9 \cdot 10^{-4}$ N to $3.8 \cdot 10^{-4}$ N which is illustrated in Figure 6-13.

Figure 6-11: Linear velocity of a particle with varying friction coefficient

Figure 6-12: Angular velocity of a particle with varying friction coefficient

Figure 6-13: Drag force on a particle with varying friction coefficient

The influence of the rolling resistance coefficient was successfully used to model the rough surface of complicated shapes of sand grains, without changing the spherical shape of a DEM particle [4], [6]. Good results can only be obtained if experimental results of a simple compression test for the soil in question are available. In the mentioned papers an additional triaxial or biaxial compression test was simulated and the CRR was adjusted in a way that the computational results matched the experimental measurements. This feature can be used in future research to model the sediment in great detail.

7. Conclusions

The robustness of the DEM-CFD method was studied for a problem of bed scour around a cylindrical pier and a flume with a tray filled with particles. These examples were chosen to determine if the method predicts the incipient movement of the sediment with sufficient accuracy for engineering applications. Moreover, a relationship between the scour hole depth and particle density in the form of a ‘decay function’ is proposed.

The method can be computationally demanding, limiting the number or size of particles as well as the simulation time. Very small particles can cause the simulation to be too computationally expensive. On the other hand, bigger particles require bigger cell sizes, which may lead to a less accurate flow solution. A small strain formulation was adopted, which may not be sufficient for deformable solids, but is a good approximation when sand grains are analyzed.

In spite of the mentioned limitations, DEM is a useful tool to model granular flows in great detail. The method can represent individual grains of sand and interactions between them. It delivers results which agree well with experimental measurements, e.g. the dimensional Shields’ diagram.

The method made it possible to perform a parametric study to establish the decaying drag forces acting on the sediment with increasing depth of the scour hole.

8. References

- [1] Guo J., Hunter Rouse and Shield's Diagram, Advances in Hydraulics and Water Engineering, Proc. 13th IAHR-APD Congress, vol.2, World Scientific, Singapore, 2002
- [2] www.grc.nasa.gov/www/K-12/airplane/dragosphere.html
- [3] ocw.mit.edu/courses/earth-atmospheric-and-planetary-sciences/12-090-introduction-to-fluid-motions-sediment-transport-and-current-generated-sedimentary-structures-fall-2006/course-textbook/ch9.pdf
- [4] Edwards C., Lottes S., Majumdar P., Three-dimensional mesh morphing methodology for scouring around bridge piers based on computational fluid dynamic solution, Proc. ASME International Mechanical Engineering Congress & Exposition, IMCEE2012, November 20112, Houston, TX
- [5] Nitka M., Tejchman J., Kozicki J., Discrete Modelling of Micro-structural Phenomena in Granular Shear Zones, Proceedings of the 10th International Workshop on Bifurcation and Degradation in Geomaterials, 2015
- [6] Kozicki J., Tejchman J., Mroz Z., Effect of grain roughness on strength, volume changes elastic and dissipated energies during quasi-static homogeneous triaxial compression using DEM, Granular Matter 14, 2012, pp. 457-468

Energy Systems Division

Argonne National Laboratory
9700 South Cass Avenue, Bldg. 362
Argonne, IL 60439-4815

www.anl.gov

Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC