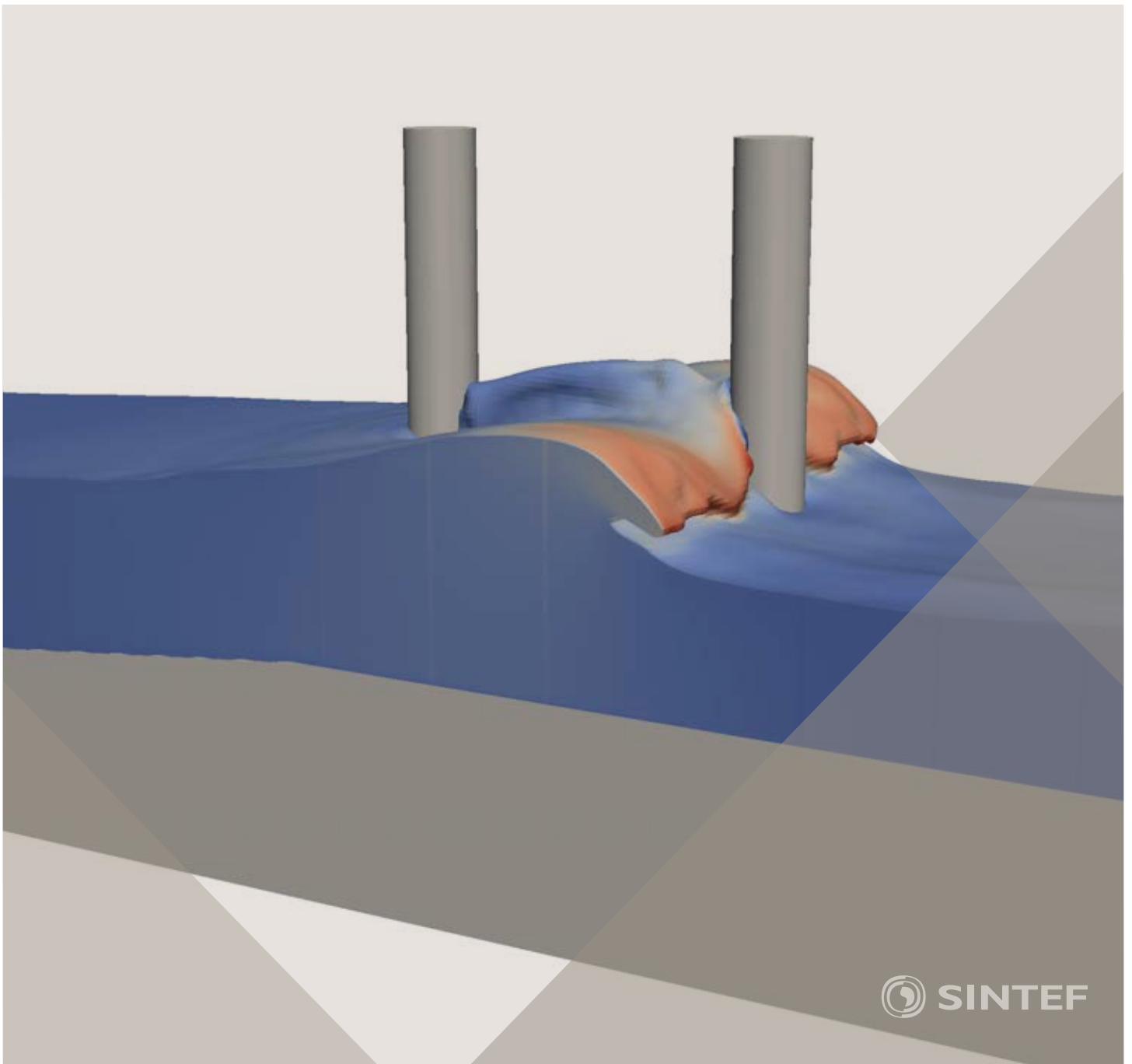


Proceedings of the 12th International Conference on
Computational Fluid Dynamics in the Oil & Gas,
Metallurgical and Process Industries

Progress in Applied CFD – CFD2017



SINTEF Proceedings

Editors:

Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Proceedings of the 12th International Conference on Computational Fluid Dynamics
in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings no 2

Editors: Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Selected papers from 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

Key words:

CFD, Flow, Modelling

Cover, illustration: Arun Kamath

ISSN 2387-4295 (online)

ISBN 978-82-536-1544-8 (pdf)

© Copyright SINTEF Academic Press 2017

The material in this publication is covered by the provisions of the Norwegian Copyright Act. Without any special agreement with SINTEF Academic Press, any copying and making available of the material is only allowed to the extent that this is permitted by law or allowed through an agreement with Kopinor, the Reproduction Rights Organisation for Norway. Any use contrary to legislation or an agreement may lead to a liability for damages and confiscation, and may be punished by fines or imprisonment

SINTEF Academic Press

Address: Forskningsveien 3 B
 PO Box 124 Blindern
 N-0314 OSLO

Tel: +47 73 59 30 00

Fax: +47 22 96 55 08

www.sintef.no/byggforsk

www.sintefbok.no

SINTEF Proceedings

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

PREFACE

This book contains all manuscripts approved by the reviewers and the organizing committee of the 12th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in May/June 2017 and is also known as CFD2017 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focuses on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. In addition pragmatic modelling concepts and bio-mechanical applications have become an important part of the conference. The papers in this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are included in the proceedings. 108 contributions were presented at the conference together with six keynote presentations. A majority of these contributions are presented by their manuscript in this collection (a few were granted to present without an accompanying manuscript).

The organizing committee would like to thank everyone who has helped with review of manuscripts, all those who helped to promote the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: ANSYS, SFI Metal Production and NanoSim.

Stein Tore Johansen & Jan Erik Olsen



Organizing committee:

Conference chairman: Prof. Stein Tore Johansen

Conference coordinator: Dr. Jan Erik Olsen

Dr. Bernhard Müller

Dr. Sigrid Karstad Dahl

Dr. Shahriar Amini

Dr. Ernst Meese

Dr. Josip Zoric

Dr. Jannike Solsvik

Dr. Peter Witt

Scientific committee:

Stein Tore Johansen, SINTEF/NTNU

Bernhard Müller, NTNU

Phil Schwarz, CSIRO

Akio Tomiyama, Kobe University

Hans Kuipers, Eindhoven University of Technology

Jinghai Li, Chinese Academy of Science

Markus Braun, Ansys

Simon Lo, CD-adapco

Patrick Segers, Universiteit Gent

Jiyuan Tu, RMIT

Jos Derksen, University of Aberdeen

Dmitry Eskin, Schlumberger-Doll Research

Pär Jönsson, KTH

Stefan Pirker, Johannes Kepler University

Josip Zoric, SINTEF

CONTENTS

PRAGMATIC MODELLING	9
On pragmatism in industrial modeling. Part III: Application to operational drilling	11
CFD modeling of dynamic emulsion stability	23
Modelling of interaction between turbines and terrain wakes using pragmatic approach	29
FLUIDIZED BED	37
Simulation of chemical looping combustion process in a double looping fluidized bed reactor with cu-based oxygen carriers.....	39
Extremely fast simulations of heat transfer in fluidized beds.....	47
Mass transfer phenomena in fluidized beds with horizontally immersed membranes	53
A Two-Fluid model study of hydrogen production via water gas shift in fluidized bed membrane reactors	63
Effect of lift force on dense gas-fluidized beds of non-spherical particles	71
Experimental and numerical investigation of a bubbling dense gas-solid fluidized bed	81
Direct numerical simulation of the effective drag in gas-liquid-solid systems	89
A Lagrangian-Eulerian hybrid model for the simulation of direct reduction of iron ore in fluidized beds.....	97
High temperature fluidization - influence of inter-particle forces on fluidization behavior	107
Verification of filtered two fluid models for reactive gas-solid flows	115
BIOMECHANICS.....	123
A computational framework involving CFD and data mining tools for analyzing disease in carotid artery	125
Investigating the numerical parameter space for a stenosed patient-specific internal carotid artery model.....	133
Velocity profiles in a 2D model of the left ventricular outflow tract, pathological case study using PIV and CFD modeling.....	139
Oscillatory flow and mass transport in a coronary artery.....	147
Patient specific numerical simulation of flow in the human upper airways for assessing the effect of nasal surgery.....	153
CFD simulations of turbulent flow in the human upper airways	163
OIL & GAS APPLICATIONS	169
Estimation of flow rates and parameters in two-phase stratified and slug flow by an ensemble Kalman filter	171
Direct numerical simulation of proppant transport in a narrow channel for hydraulic fracturing application	179
Multiphase direct numerical simulations (DNS) of oil-water flows through homogeneous porous rocks	185
CFD erosion modelling of blind tees	191
Shape factors inclusion in a one-dimensional, transient two-fluid model for stratified and slug flow simulations in pipes	201
Gas-liquid two-phase flow behavior in terrain-inclined pipelines for wet natural gas transportation	207

NUMERICS, METHODS & CODE DEVELOPMENT	213
Innovative computing for industrially-relevant multiphase flows	215
Development of GPU parallel multiphase flow solver for turbulent slurry flows in cyclone.....	223
Immersed boundary method for the compressible Navier–Stokes equations using high order summation-by-parts difference operators	233
Direct numerical simulation of coupled heat and mass transfer in fluid-solid systems	243
A simulation concept for generic simulation of multi-material flow, using staggered Cartesian grids.....	253
A cartesian cut-cell method, based on formal volume averaging of mass, momentum equations.....	265
SOFT: a framework for semantic interoperability of scientific software	273
 POPULATION BALANCE	 279
Combined multifluid-population balance method for polydisperse multiphase flows	281
A multifluid-PBE model for a slurry bubble column with bubble size dependent velocity, weight fractions and temperature.....	285
CFD simulation of the droplet size distribution of liquid-liquid emulsions in stirred tank reactors	295
Towards a CFD model for boiling flows: validation of QMOM predictions with TOPFLOW experiments	301
Numerical simulations of turbulent liquid-liquid dispersions with quadrature-based moment methods.....	309
Simulation of dispersion of immiscible fluids in a turbulent couette flow	317
Simulation of gas-liquid flows in separators - a Lagrangian approach.....	325
CFD modelling to predict mass transfer in pulsed sieve plate extraction columns	335
 BREAKUP & COALESCENCE	 343
Experimental and numerical study on single droplet breakage in turbulent flow	345
Improved collision modelling for liquid metal droplets in a copper slag cleaning process	355
Modelling of bubble dynamics in slag during its hot stage engineering.....	365
Controlled coalescence with local front reconstruction method	373
 BUBBLY FLOWS	 381
Modelling of fluid dynamics, mass transfer and chemical reaction in bubbly flows	383
Stochastic DSMC model for large scale dense bubbly flows.....	391
On the surfacing mechanism of bubble plumes from subsea gas release.....	399
Bubble generated turbulence in two fluid simulation of bubbly flow	405
 HEAT TRANSFER	 413
CFD-simulation of boiling in a heated pipe including flow pattern transitions using a multi-field concept	415
The pear-shaped fate of an ice melting front	423
Flow dynamics studies for flexible operation of continuous casters (flow flex cc).....	431
An Euler-Euler model for gas-liquid flows in a coil wound heat exchanger.....	441
 NON-NEWTONIAN FLOWS.....	 449
Viscoelastic flow simulations in disordered porous media	451
Tire rubber extrudate swell simulation and verification with experiments	459
Front-tracking simulations of bubbles rising in non-Newtonian fluids.....	469
A 2D sediment bed morphodynamics model for turbulent, non-Newtonian, particle-loaded flows.....	479

METALLURGICAL APPLICATIONS.....	491
Experimental modelling of metallurgical processes	493
State of the art: macroscopic modelling approaches for the description of multiphysics phenomena within the electroslag remelting process	499
LES-VOF simulation of turbulent interfacial flow in the continuous casting mold	507
CFD-DEM modelling of blast furnace tapping	515
Multiphase flow modelling of furnace tapholes	521
Numerical predictions of the shape and size of the raceway zone in a blast furnace.....	531
Modelling and measurements in the aluminium industry - Where are the obstacles?	541
Modelling of chemical reactions in metallurgical processes.....	549
Using CFD analysis to optimise top submerged lance furnace geometries	555
Numerical analysis of the temperature distribution in a martensic stainless steel strip during hardening.....	565
Validation of a rapid slag viscosity measurement by CFD.....	575
Solidification modeling with user defined function in ANSYS Fluent.....	583
Cleaning of polycyclic aromatic hydrocarbons (PAH) obtained from ferroalloys plant.....	587
Granular flow described by fictitious fluids: a suitable methodology for process simulations	593
A multiscale numerical approach of the dripping slag in the coke bed zone of a pilot scale Si-Mn furnace.....	599
INDUSTRIAL APPLICATIONS	605
Use of CFD as a design tool for a phosphoric acid plant cooling pond	607
Numerical evaluation of co-firing solid recovered fuel with petroleum coke in a cement rotary kiln: Influence of fuel moisture	613
Experimental and CFD investigation of fractal distributor on a novel plate and frame ion-exchanger	621
COMBUSTION	631
CFD modeling of a commercial-size circle-draft biomass gasifier.....	633
Numerical study of coal particle gasification up to Reynolds numbers of 1000.....	641
Modelling combustion of pulverized coal and alternative carbon materials in the blast furnace raceway	647
Combustion chamber scaling for energy recovery from furnace process gas: waste to value	657
PACKED BED.....	665
Comparison of particle-resolved direct numerical simulation and 1D modelling of catalytic reactions in a packed bed	667
Numerical investigation of particle types influence on packed bed adsorber behaviour	675
CFD based study of dense medium drum separation processes	683
A multi-domain 1D particle-reactor model for packed bed reactor applications.....	689
SPECIES TRANSPORT & INTERFACES	699
Modelling and numerical simulation of surface active species transport - reaction in welding processes	701
Multiscale approach to fully resolved boundary layers using adaptive grids.....	709
Implementation, demonstration and validation of a user-defined wall function for direct precipitation fouling in Ansys Fluent.....	717

FREE SURFACE FLOW & WAVES	727
Unresolved CFD-DEM in environmental engineering: submarine slope stability and other applications.....	729
Influence of the upstream cylinder and wave breaking point on the breaking wave forces on the downstream cylinder	735
Recent developments for the computation of the necessary submergence of pump intakes with free surfaces	743
Parallel multiphase flow software for solving the Navier-Stokes equations	752
 PARTICLE METHODS	 759
A numerical approach to model aggregate restructuring in shear flow using DEM in Lattice-Boltzmann simulations	761
Adaptive coarse-graining for large-scale DEM simulations.....	773
Novel efficient hybrid-DEM collision integration scheme.....	779
Implementing the kinetic theory of granular flows into the Lagrangian dense discrete phase model.....	785
Importance of the different fluid forces on particle dispersion in fluid phase resonance mixers	791
Large scale modelling of bubble formation and growth in a supersaturated liquid.....	798
 FUNDAMENTAL FLUID DYNAMICS	 807
Flow past a yawed cylinder of finite length using a fictitious domain method	809
A numerical evaluation of the effect of the electro-magnetic force on bubble flow in aluminium smelting process.....	819
A DNS study of droplet spreading and penetration on a porous medium.....	825
From linear to nonlinear: Transient growth in confined magnetohydrodynamic flows.....	831

UNRESOLVED CFD-DEM IN ENVIRONMENTAL ENGINEERING: SUBMARINE SLOPE STABILITY AND OTHER APPLICATIONS

Alice HAGER^{1*}, Manuela KANITZ², Jürgen GRABE², Christoph KLOSS¹, Christoph GONIVA¹

¹ DCS Computing GmbH, 4020 Linz, AUSTRIA

² TUHH, Institute of Geotechnical Engineering and Construction Management, 21079 Hamburg, GERMANY

* E-mail: alice.hager@dcs-computing.com

ABSTRACT

When installing gravity foundations for offshore structures such as wind power stations or oil platforms, the seabed needs to be excavated for providing enough stability. To minimize the impact on the surrounding fauna and the installation costs, steep but stable slopes are desired. The work presented is done in a research project on the numerical investigation of the stability of submarine slopes, particularly under the impact of influences like material removal or wave-induced disturbances.

The method used in the current project is coupled CFD-DEM: while the dynamics of the fluid phase (water and in some cases water and air) are handled with computational fluid dynamics (CFD), the soil is modelled by spheres, whose motion is calculated with a discrete element method (DEM). Force models are used for considering the particles' effect on the fluid and vice versa, a void fraction field accounts for the volume of the particles on the CFD side. Due to the high number of particles in the domain only unresolved CFD-DEM (cf., e.g. Zhou (2010)) is suitable: in this case the particles are smaller than the cells of the CFD mesh.

In the presented work the investigations concentrated on the validation of the CFD-DEM models against small-scale experiments that were conducted by the authors. In a first step, the used materials were characterized and a lubrication force model was implemented. Furthermore, some basic investigations on the topic of dilatancy were carried out. Then an experimental setup and an according simulation were compared. In addition to that a three phase (air, water, particles) solver was used to depict the effect of surface waves onto the particle bed.

For the calculations CFDEM@coupling was used. CFDEM@coupling is an Open Source software for coupled CFD-DEM simulations. It uses the CFD framework of the Open Source CFD code OpenFOAM® and the DEM framework of the Open Source code LIGGGHTS®. Both CFDEM@coupling and LIGGGHTS® have been presented before (cf., e.g. Goniva et al. (2012), Kloss et al. (2012)), the used model equations were validated against analytical solutions and literature.

Keywords: Lagrangian methods, granular flows, unresolved CFD-DEM.

NOMENCLATURE

Greek Symbols

- α Volume fraction.
- ρ Mass density, [kg/m³].
- μ Dynamic viscosity, [kg/m.s].
- ω Angular velocity, [rad/s].

Latin Symbols

- F Force, [N].
- g Gravitational acceleration, [m/s²].
- h Minimal surface distance of two particles, [m].
- m Mass, [kg].
- p Pressure, [Pa].
- r Particle radius, [m].
- U Velocity, [m/s].
- v Velocity [m/s].
- R_{sl} Solid-liquid interaction force, [kg/(m.s)²].
- T Torque, [N.m].

Sub/superscripts

- f Fluid
- i Index i
- j Index j
- p Particle
- w Wall

INTRODUCTION

The reasons for producing under water slopes are manifold: they occur when sand or gravel is harvested as well as when foundations for off-shore wind power plants or oil platforms are required. For both economic and ecological reasons it is desirable to build steep and yet stable slopes. The stability of the slopes and their formation can either be influenced by the production process itself (e.g. grab or suction dredging) or environmental phenomena such as surface waves. A joint research project between TUHH and DCS Computing GmbH aims on modelling different phenomena at the soil-water interface. The presented contents were developed within the course of this project.

First, the implementation and validation of a lubrication force model is discussed and the capability of depicting

dilatancy is demonstrated. Modelling the effect of dilatancy is crucial for simulating saturated sand beds. Due to shear loading, dilatancy leads to a local hardening of the soil bed as the pore volume increases. This effect causes a negative excess pore water pressure and hence suction occurs until the inflowing pore water fills the volume between the soil grains. In a second part two application cases are considered:

- (i) the effect of suction dredging on an underwater slope is investigated numerically and compared to experimental data
- (ii) a three-phase solver is used to proof the feasibility of simulating wave induced disturbances on particle beds

The presented work is realized with the Open Source software packages LIGGGHTS® (Kloss et al. (2012)) and CFDEM®coupling (Goniva et al. (2012)).

MODEL DESCRIPTION

A coupled CFD-DEM model was used to compute the dynamics of the fluid and particle phases and their interaction. On the CFD side the computational domain was discretised and a finite volume solver was applied. For the granular phase a Lagrangian method was used, in which each particle was considered individually.

Governing equations

The governing equations for the presented CFD-DEM method are the volume averaged Navier-Stokes equations:

Continuity equation

$$\frac{\partial \alpha_f \rho_f}{\partial t} + \nabla \cdot (\alpha_f \rho_f U) = 0 \quad (1)$$

Momentum equation

$$\frac{\partial \alpha_f \rho_f U}{\partial t} + \nabla \cdot (\alpha_f \rho_f U U) = -\alpha_f \nabla p + R_{sl} + \alpha_f \rho_f g \quad (2)$$

The term R_{sl} denotes the fluid-structure interaction force density, which is computed with the aid of particle-data. In the DEM method used for the calculation of the particle phase, the trajectory of each particle is calculated separately, using Newton's second law:

$$m_p \frac{dU_p}{dt} = \sum F_{p,p} + \sum F_{p,w} + m_p g + F_p + F_f \quad (3)$$

$$I_p \frac{d\omega_p}{dt} = T_p \quad (4)$$

The right-hand side of equation (3) consists of the particle-particle and particle-wall interaction forces, for their calculation a soft-sphere approach was used. The term $m_p g$ represents the mass-force due to gravity and F_p stands for the pressure forces. The term F_f represents the forces exerted by the fluid. In the current case the drag force, the pressure gradient force and the viscous force were identified as dominating forces. The drag force model by Koch and Hill (2001) was used. As the particle-particle contacts take place under water also a lubrication force model was implemented and validated.

Lubrication force model

If two particles collide within a fluid the displacement of the fluid in the gap between them causes a force – the lubrication force. While for many processes where fluids with relatively low viscosities are present the effect of the force is negligible, it might be worth considering in the presented applications. The classical formulation of the lubrication force is given as

$$F_{\text{lub}} = -6 \cdot \pi \cdot \mu \cdot r_{ij}^2 \cdot \left(\frac{1}{h} \right) \cdot v_n, \quad (5)$$

where r_{ij} is the effective radius defined as

$$\frac{1}{r_{ij}} = \frac{1}{r_i} + \frac{1}{r_j}, \quad (6)$$

with i and j denoting arbitrary different particles. h is the minimal surface distance between two particles and one has to make sure that the lubrication force does not become infinitely large when the particles are in contact with each other. Therefore, the radius of action of the lubrication force was limited to a certain minimal distance between two particles. This approach was also used by Sun and Xiao (2016) in their implementation.

Validation of the lubrication force model

In the considered cases particle-particle interactions have by far more impact than particle-wall interactions. Therefore, also the validation test-case focused on particle-particle interaction: a sphere was fixed at a specific position while another sphere was released “directly” above it, with different initial velocities and for different fluid viscosities.

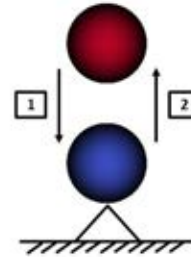


Figure 1: Basic test case for the validation of the lubrication force model.

During the simulation, the velocity of the moving particle was measured right before and after the collision. The velocity before the impact, v_1 , was used to calculate the Stokes number as following (cf., e.g. Tomac (2013)):

$$St = \frac{m v_1}{6 \pi \mu r^2}. \quad (7)$$

The coefficient of restitution was determined as

$$c_{\text{rest}} = \frac{v_2}{v_1}. \quad (8)$$

In the diagram in Fig. 2 the coefficients of restitution for different fluid viscosities are displayed over the according Stokes numbers. The results of the implemented model are compared to values presented by Zhang et al. (2005) and by Tomac and Gutierrez (2013), a good accordance could be obtained.

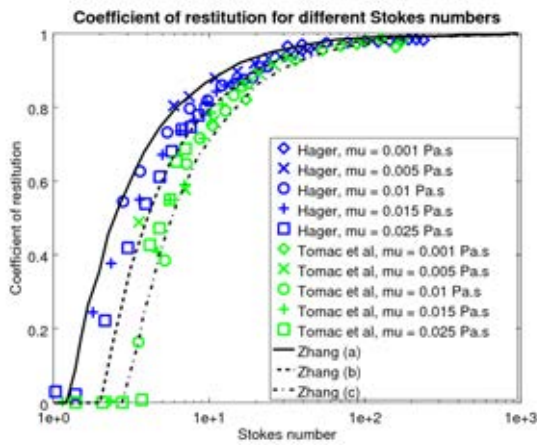


Figure 2: Validation of the lubrication force model.

Depiction of dilatancy

The qualitative test for demonstrating the capability of depicting dilatancy was the following: three packings with the same volume but of different volume fractions were generated (0.75, 0.80, 0.85), the packings consisted of about 3000 particles. For keeping the particle packing in place, fixed side walls were chosen, while the upper and the lower wall were moving: for the top wall a so-called servo wall was used that exerted a constant pressure force on the bed, while the lower wall was moved horizontally. The force of the servo wall was chosen such that the particle bed was at an equilibrium at the beginning of the test.

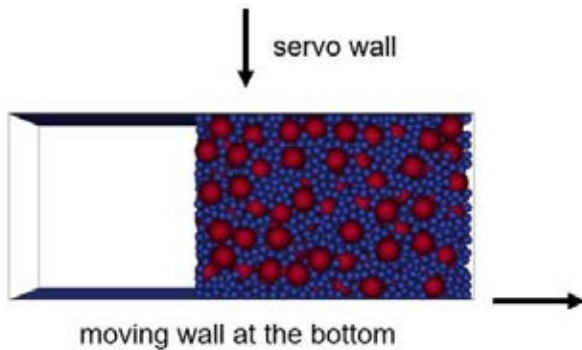


Figure 3: Setup of the dilatancy test case.

As soon as the motion of the lower wall started, the particles started to re-arrange and thus expand. The expansion showed in a lifting of the top plate, increasing with increasing initial volume fraction (cf. Fig. 4).

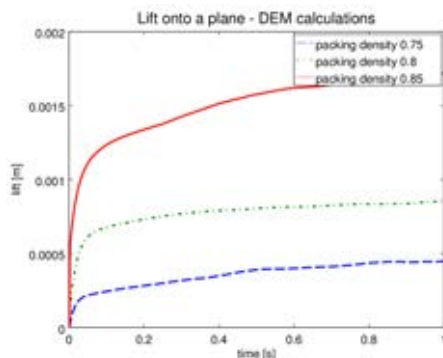


Figure 4: Displacement of the upper wall for different packing densities.

With this simple test the ability to depict dilatancy could be shown.

FORMATION OF UNDER WATER SLOPES

When extracting sand and gravel from under-water regions one often uses suction dredgers. In this process pumps deliver sand/gravel and water mixtures. Grabe (2005) presented a detailed small-scale experimental study of the effect of grab dredging onto the formation of an under-water slope. The material used in the experiments was described as sand with a critical angle of friction of 34° .

Material calibration

Accurate simulations require detailed knowledge of the used material. The material parameters can be divided in two groups: Some quantities can be measured directly, like for example density, grain size distribution or the volume fraction, while the set of model parameters needs to be determined otherwise. These describe the physical behaviour of the bulk material. Examples are the coefficient of friction and the coefficient of rolling friction. For obtaining these coefficients, a set of three well-established calibration experiments and simulations was conducted:

1. The shear cell test

The coefficient of friction between the particles can be determined with a shear cell. For the experiments Jenike shear testers with cylinder diameters of about 10 cm were used. These devices consist of an upper and a lower hollow cylinder which are filled with the granular material. One of these cylinders is fixed, the other one can be displaced linearly in horizontal direction. For the tests a weight is placed on top of the granular material (exerting a normal force) and the non-fixed cylinder is moved slowly in horizontal direction. The resulting shear stress is measured and compared to the shear stresses obtained by the ring shear cell test in the simulation.



Figure 5: Calibrating the particle-particle coefficient of friction with shear cell experiments and simulations.

2. Inclined plate

This experiment serves the purpose of determining the coefficient of rolling/sliding friction between particles and a wall. A small sample of grains is placed on a plate which is then inclined gradually. From a certain inclination (target angle) particles start to roll or slide down the plate, depending on whether the sliding or rolling friction is stronger.



Figure 6: Inclined plate test.

In the simulation particles are placed on a horizontal plate and the gravity vector is chosen such that the target angle is obtained. One then looks for the smallest coefficient of rolling friction, for which the particles are at rest (cf., Fig. 7).

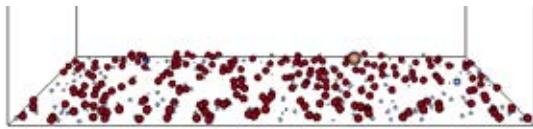


Figure 7: Simulation of the inclined plate test.

3. Angle of repose

The angle of repose experiment is used to determine the particle-particle coefficient of rolling friction. The setup used for the test is the following: a hollow cylinder is placed on an elevated circular plate with the same diameter as the cylinder. The cylinder is filled with granular material and the lifted slowly and uniformly, which leads to the formation of a conical heap. In the simulation the coefficient of rolling friction is varied until the angle of the cone matches the result of the experiment (cf., Fig. 8).

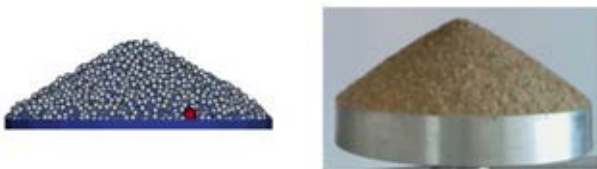


Figure 8: The particle-particle coefficient of rolling friction dominates the formation of the angle of repose.

The grain size distribution was identified by sieving, the average coefficient of restitution was determined with drop tests.

Simulation setup

As the experiment was carried out in a small scale it was possible to simulate the process at original scale. A slope with an inclination angle of 30° (smaller than the critical angle) was generated (cf., Fig. 9). The simulation domain had a length of 40 cm and a height of 25 cm. The maximal height of the particle bed was 20 cm. For reducing the number of particles in the simulation domain periodic boundary conditions were used for the front and back wall both on the CFD and the DEM side. The complete set of velocity and pressure boundary conditions is summed up in Table 1. The bed was initialized with a pure DEM calculation.

	velocity (U, m/s)	pressure (p, Pa)
top	zeroGradient	fixedValue (0)
bottom	fixedValue (0,0,0)	zeroGradient
front/back	cyclic	cyclic
left wall	fixedValue, (0,0,0)	zeroGradient
right wall	zeroGradient	fixedValue (0)

Table 1: Velocity and pressure boundary conditions for the CFD calculation.

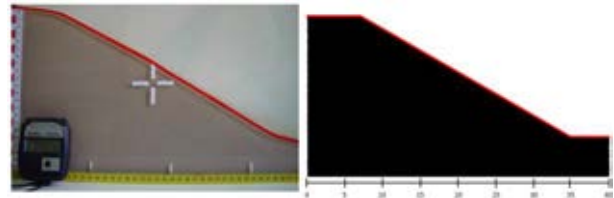


Figure 9: Initial slope (experiment vs. simulation)

In the current investigation, the focus lay on the formation of the slope after the extraction of the particles, thus the particles were removed at once in the simulation (cf., Fig. 10).

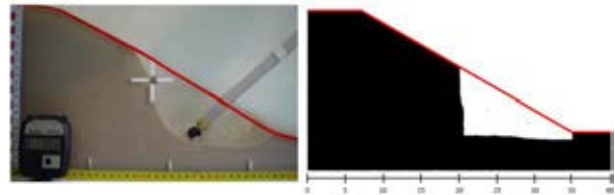


Figure 10: Slope after suction dredging is completed.

Results

After the removal of the particles in the dredging region the coupled CFD-DEM calculation was launched and the settling process started. In Fig. 11 two images of unstable slopes during the settling process in experiment and simulation are compared.

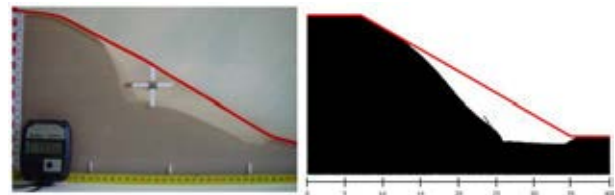


Figure 11: Slope during settling process (unstable).

It can be observed that while the reformation of the slope is ongoing, it develops slightly different in simulation and experiment. Furthermore, the time scale is smaller in the simulation, i.e. the changes of the slope occur faster. This difference could stem from an under-representation of the pore pressure in the simulation, but will be subject to future investigations as well. However, as can be seen in Fig. 12, in both cases the resulting final slope has an inclination of about 34° , which is the critical angle of the sand used.

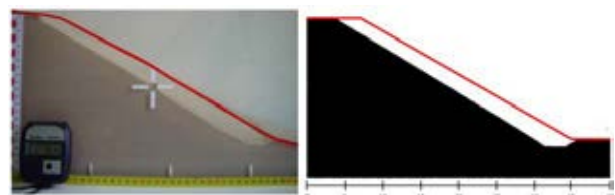


Figure 12: Final slope.

WAVE INDUCED DISTURBANCES

The purpose of this application example was to show the feasibility of combining a three-phase solver (two-phase fluid and particles) with the wave generation toolbox

waves2foam (cf., Jacobsen et al. (2011)). waves2foam uses a zone in which the waves are generated. For making sure that the manipulations of the equations due to wave generation and those due to the presence of a granular phase do not interfere, a separate wave generation zone without particles was used (cf., Fig. 13). The granular material used for these calculations were 3mm glass beads, their properties were determined as described in the previous section. The total length of the CFD domain was 1.2 m, whereas the first 0.4 m were used as wave generation zone. The height of the domain was 0.2 m and due to the application of periodic boundary conditions a domain depth of 0.05 m could be used.

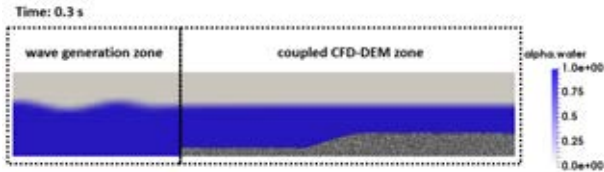


Figure 13: The waves were generated in a zone without particles.

As can be seen in Fig. 14 the waves propagate through the whole domain, their shape is influenced by the ascending sea bed.

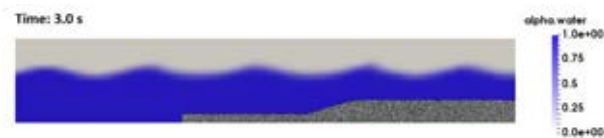


Figure 14: The waves propagate through the whole domain; the particle bed influences their shape.

When considering the force which the fluid exerts on the particle bed (here termed as drag force), the influence of the waves onto the soil becomes visible (cf. Fig. 15). The higher/lower force values onto the particles are in direct relation with the fluid pressure field ρ_{rgh} (total pressure minus hydrostatic pressure): it is lower underneath wave troughs and higher in wave crest areas and acts similar as a lift force.

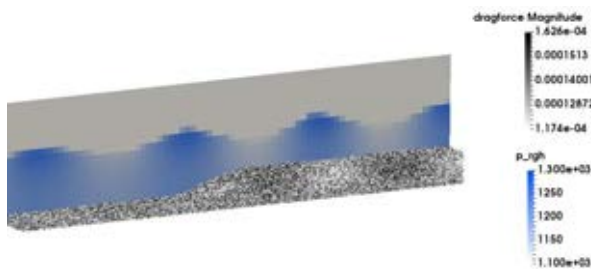


Figure 15: underneath the wave troughs the drag force onto the particles is increased while it is decreased underneath the wave crests.

These calculations showed that it is possible to apply wave boundary conditions to three phase problems within CFDEM@coupling. The observed behaviour matches the expectations and future experiments will be used for validation.

CONCLUSION AND OUTLOOK

A lubrication force model was implemented and validated and it was demonstrated that dilatancy can be depicted. The simulation of a suction dredging case was in good accordance with the experiments. Furthermore, the feasibility and plausibility of the combination of a three-phase solver and the toolbox waves2foam was shown by a small-scale test case, experiments for validating the new solver are planned.

ACKNOWLEDGEMENTS

The authors are grateful to the Austrian Science Fund (FWF) for supporting this research under project I 2257 (DACH project).

REFERENCES

- GONIVA, C., KLOSS, C., KUIPERS, J.A.M. and PIRKER, S., (2012), "Influence of rolling friction modelling on single spout fluidized bed simulations", *Particuology*, **10** (5), 582-591.
- GRABE, J., (2005), "Phänomene an der Grenzschicht zwischen Wasser und Boden", *Grenzschicht Wasser und Boden – Phänomene und Ansätze, Veröffentlichungen des Instituts für Geotechnik und Baubetrieb der TU Hamburg-Harburg*, **9**, 3-30.
- JACOBSEN, N.G., FUHRMAN, D.R. and FREDSSØE, J., (2011), "A wave generation toolbox for the open-source CFD library: OpenFOAM®", *Int. J. Numer. Meth. Fluids*, **70** (9), 1073-1088.
- KLOSS, C., GONIVA, C., HAGER, A., AMBERGER, S. and PIRKER, S., (2012), "Models, algorithms and validation for opensource DEM and CFD-DEM", *Prog. Comput. Fluid. Dy., An. Int. J.* 2012, **12** (2/3), 140-152.
- KOCH, I. and HILL, R.J., (2001), "Inertial effects in suspension and porous-media flows", *Ann. Rev. of Fluid Mechanics*, **33**, 619-647.
- OpenCFD Ltd., (2004-2017), "OpenFOAM – The Open Source CFD-toolbox", URL: www.openfoam.com.
- SUN, R. and XIAO, H., (2016), "SediFOAM: A general-purpose, open-source CFD-DEM solver for particle-laden flow with emphasis on sediment transport", *Comput. Geosci.*, **89**, 207-219.
- TAESERI, D., LAUE, J., OTSUBO, M. and TOWHATA, I., (2016), "New mitigation method for pipeline uplift during seismic event", *Geotechnical Research*, **3** (2), 54-64.
- TOMAC, I. and GUTIERREZ, M., (2013), "Discrete element modelling of non-linear submerged particle collisions", *Granul. Matter*, **15**, 759-769.
- ZHANG, W., NODA, R. and HORIO, M., (2005), "Evaluation of lubrication force on colliding particles for DEM simulation of fluidized beds", *Powder Technol.*, **158**, 92-101.
- ZHOU, Z.Y., KUANG, S.B., CHU, K.W. and YU, A.B., (2010), "Discrete particle simulation of particle-fluid flow: model formulations and their applicability", *J. Fluid Mech.*, **661**, 482-510.