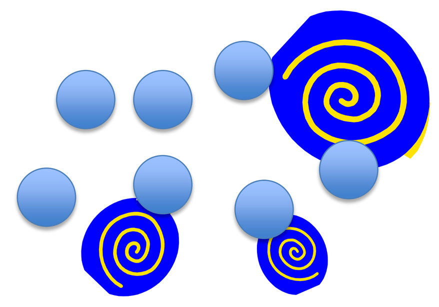
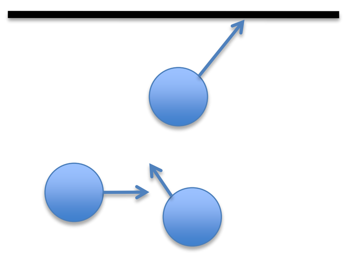
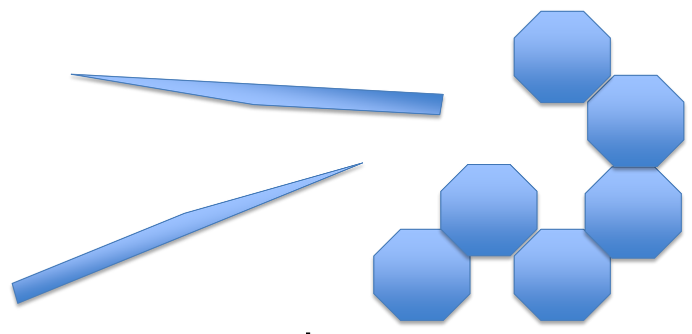
Numerical simulation of fluid-particle interactions in dense particulate flows

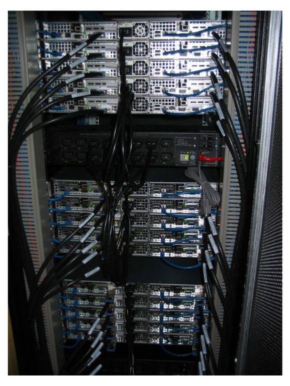
Dominique Thévenin
Fluid Dynamics, Faculty of Process and Systems Engineering, Otto von Guericke University Magdeburg

Introduction. Particulate flows are found in most applications of the GK and lead to many modeling challenges, mainly due to:


fully coupled evolution fluid⟷particles


impact of collision models


complex structure of particles & agglomerates


numerical cost of associated simulations

Problem Definition

Derive suitable physical models, then simulate numerically dense particulate flows for particles and agglomerates of arbitrary shape while taking into account resulting turbulence modifications

Objectives

(1) Optimally combine existing experience and tools concerning
a) Discrete Element Method/CFD
b) Direct Numerical Simulation
c) Lattice Boltzmann
to meet this challenge
(2) Compare with experiments, evaluate the results

Cooperation

J. Tomas– U Magdeburg (MD)
Dense particulate flows, Collision models

G. Warnecke – U MD
Population balance models

F. Scheffler– U MD
Flow in porous media

E. Tsotsas – U MD
Flow through arbitrary structures

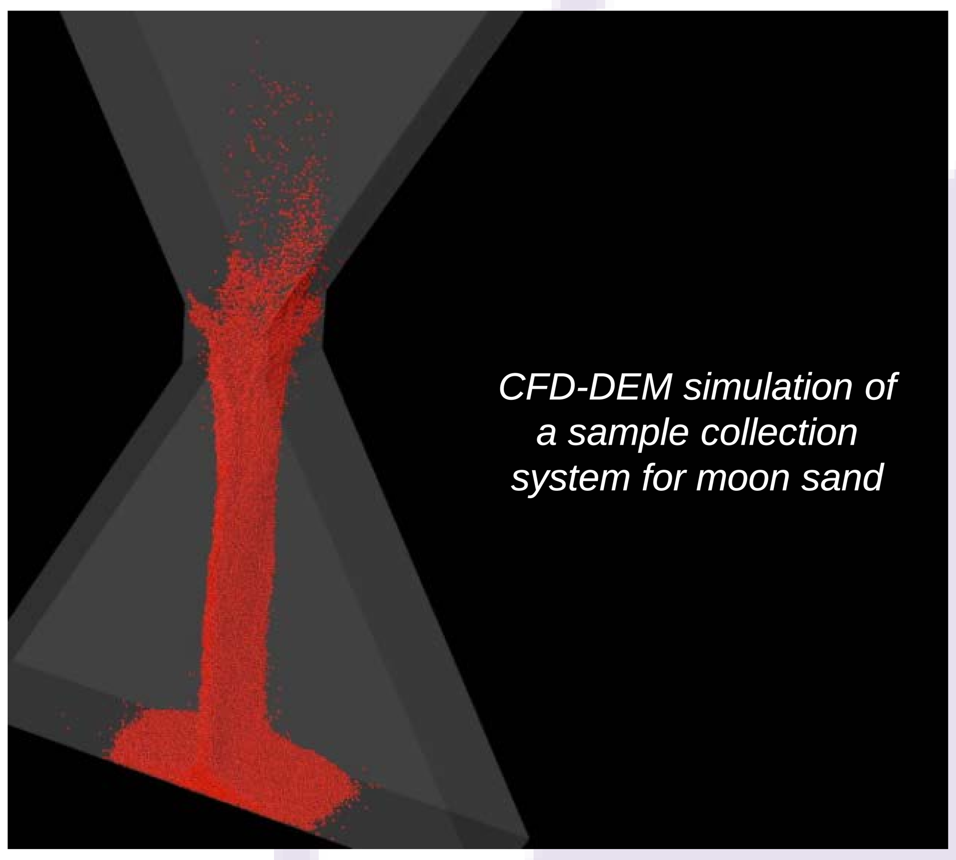
L. Tobiska – U MD
Numerical methods

H. Altenbach– U MD
Material models for DEM

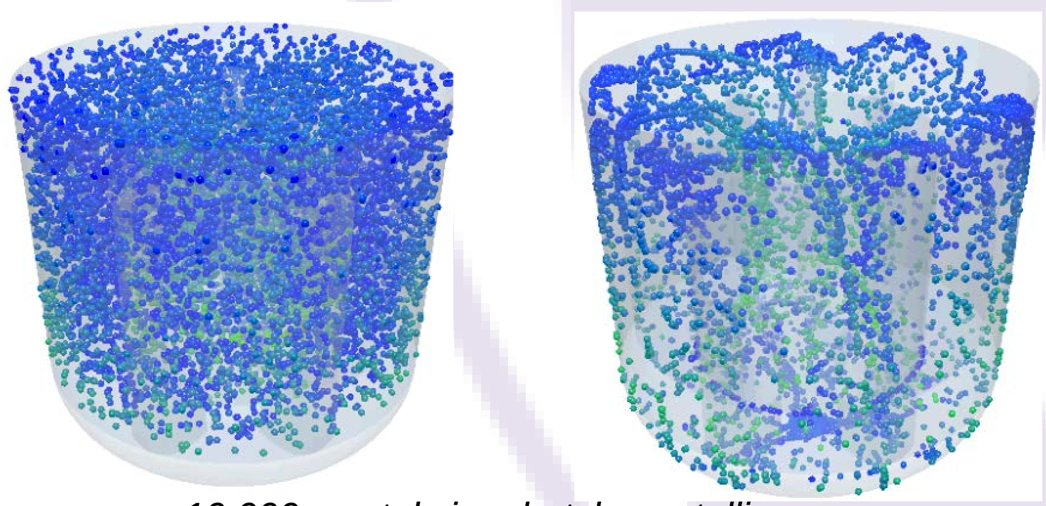
D. Marchisio – Politec. Torino
Homogenization, statistical description

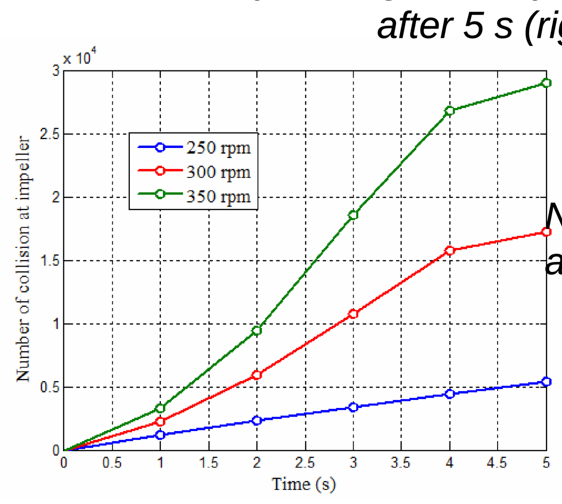
Discrete Element Method

- **Most suitable approach to model the solid phase and the structural interactions (particle-particle and particle-wall)**
- Usually coupled to a simple description of the flow (Reynolds-Averaged Navier-Stokes, RANS)
- Experience with commercial (ANSYS-Fluent & EDEM) as well as with open-source solutions (OpenFOAM & LIGGGHTS)


CFD-DEM simulation of a sample collection system for moon sand

- Numerical costs growing rapidly with number of particles/of collisions and quality of description


10 000 crystals in a batch crystallizer: initially homogeneously distributed (left), after 5 s (right)



Number of collisions vs. time as function of impeller speed

Direct Numerical Simulation

- **Most suitable approach to investigate the modulation of turbulence properties due to interaction with the particles**
- Models available: flow tracers, point particles (1-way and 2-way coupling), fully resolved particles (2-way and 4-way coupling)
- A 3rd generation DNS tool is available in our group for this purpose (4th order in space, 3rd order in time)

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho \mathbf{v}) = S_{mass},$$

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla\cdot(\rho \mathbf{v}\mathbf{v}) = -\nabla p + \nabla\cdot\tau + \rho \mathbf{F} + S_{mom},$$

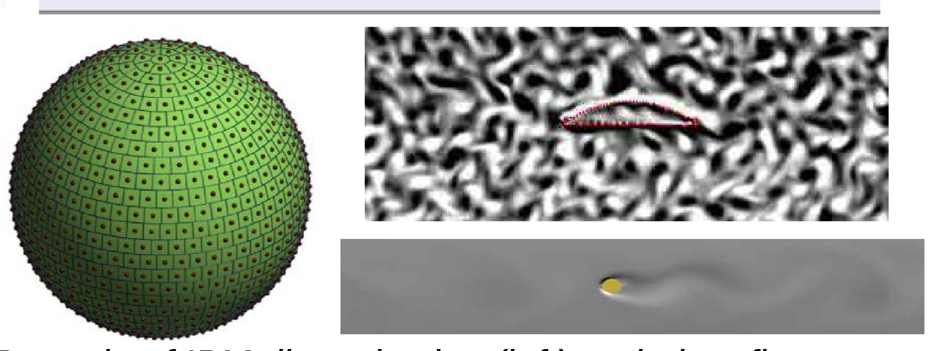
$$\frac{\partial(\rho e_t)}{\partial t} + \nabla\cdot(\rho \mathbf{v} e_t) = -\nabla\cdot\left[-\lambda\nabla T + \sum_{k=1}^{N} h_k Y_k \mathbf{V}_k\right] + \rho\sum_{k=1}^{N} Y_k f_{k,i}(\mathbf{v}+\mathbf{V}_k) + S_{energy}$$

$$\frac{\partial(\rho Y_k)}{\partial t} + \nabla\cdot(\rho \mathbf{v} Y_k) = -\nabla\cdot(\rho Y_k \mathbf{V}_k) + \dot\omega_k + S_{mass},$$

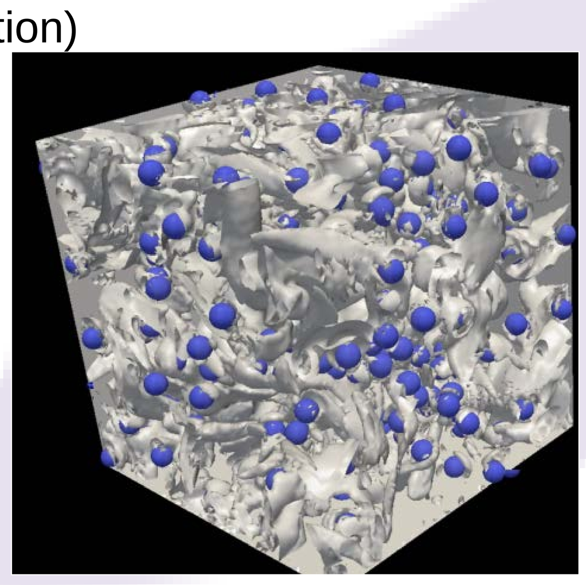
- Particles and agglomerates of arbitrary shape are described in the DNS by direct-force Immersed Boundary Method (IBM)

$$V_c^m(\rho_p^m - \rho)\frac{dU_{i,m}}{dt} = -\rho\sum_l F(X_l^m)\Delta V_l^m + \sum_{j=1, j\neq m}^{N_p} F_R^{(m,j)}$$

$$I_c^m\frac{d\Omega}{dt} = -\rho\sum_l (X_l^m - X_c^m)\times F(X_l^m)\Delta V_l^m + \rho\frac{d}{dt}\int_{\vartheta_h}((x - X_c^m)\times u)\,dx$$


Example of IBM discretization (left), turbulent flow around a lens (top right) and around a sphere (bottom right)

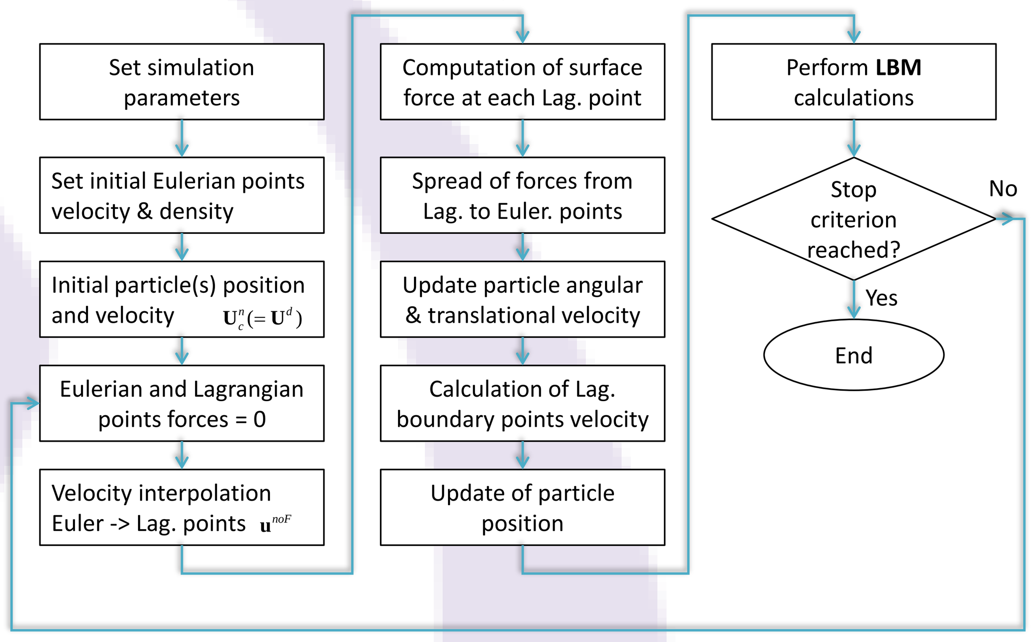
- Different collision models are implemented (hard sphere, repulsive velocity barrier, repulsive potential force, lubrication)

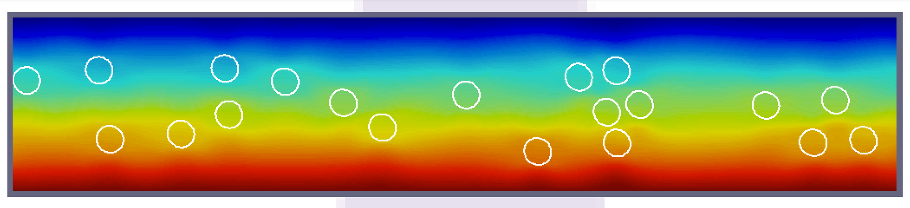

Four-way coupled DNS of turbulent particulate flow (mass loading 23%, particle diameter = 16 times Kolmogorov size)

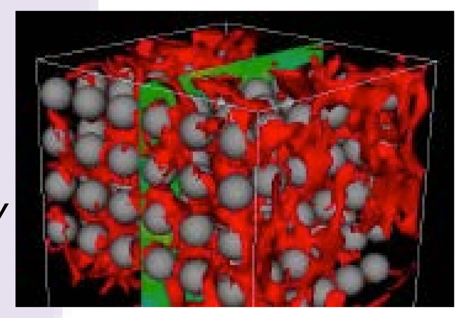
Lattice Boltzmann Simulation

- **Most suitable approach to simulate flow in porous media or in complex structures**
- A newly-written LB solver with multi-relaxation times (MRT) is available for this purpose
- As in DNS, a direct-force Immersed Boundary Method (IBM) is used to describe particles and agglomerates of arbitrary shape

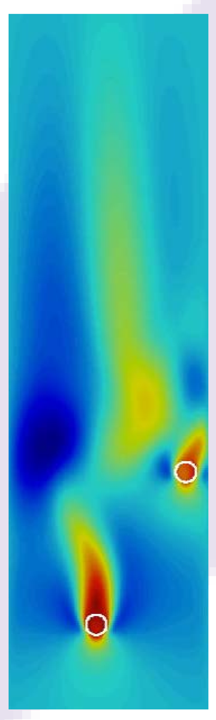
$$f_i(\mathbf{x}+\mathbf{c}_i\Delta t, t+\Delta t) = f_i(\mathbf{x},t) - \frac{\Delta t}{\tau}\left[f_i(\mathbf{x},t) - f_i^{eq}(\mathbf{x},t)\right] + F_i(\mathbf{x},t)\Delta t$$

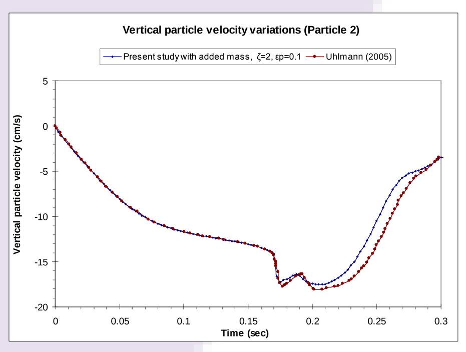


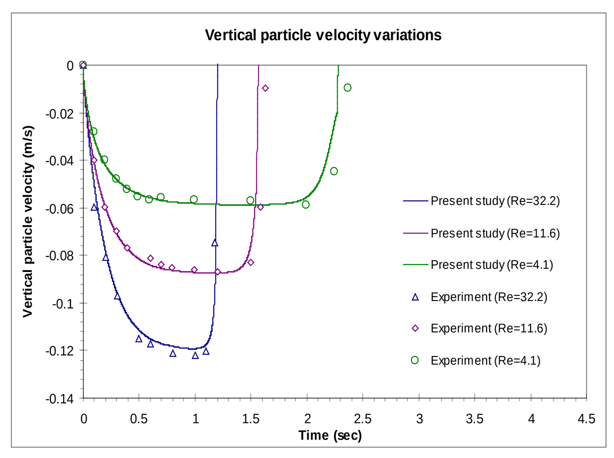

Hydraulic transport in a channel


Flow through a regularly packed bed

- Collision model: Feng & Michaelides (2005)




Standard benchmarks: DKT



Particle sedimenting in a channel