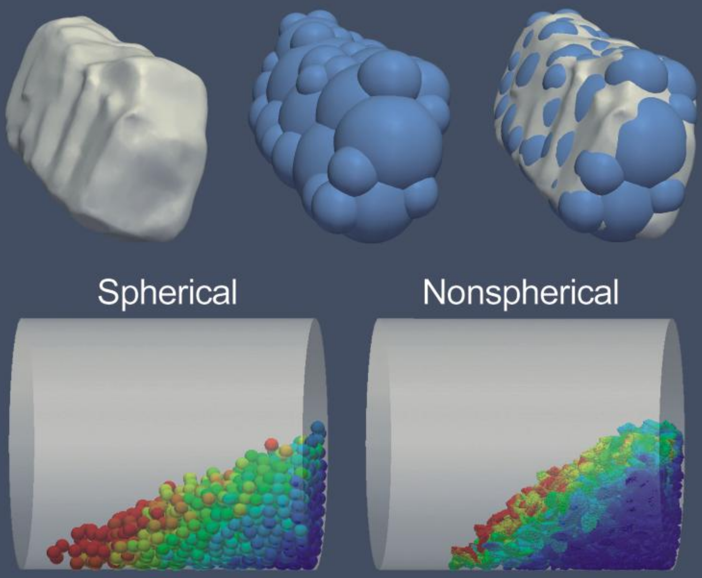


Christian-Doppler Laboratory
on Particulate Flow Modelling

Annual Report | **2011**

CHRISTIAN-DOPPLER LABORATORY ON PARTICULATE FLOW MODELLING

T +43 (0)732/2468 6477 | **F** +43 (0)732/2468 6462 | **W** http://www.particulate-flow.at

P | Altenbergerstrasse 69, 4040 Linz, Austria

Front cover: Non-sphericity has a significant influence on particle handling. In this example the shape of a real coke particle (which has been obtained by 3d laser scan) has been approximated by a couple of glued spheres. The resulting ensemble of spheres can be readily introduced into classical Discrete Element Models (DEM)

EDITORIAL 1

Dear Readers,

In its third year of operation the CD-Lab on Particulate Flow Modelling is supported by seven industrial partners with a maximum funded budget of nearly 600k€ a year. Additional research is financed by alternative funding programs or by direct bilateral research projects so that by 2012 our CD-Lab will grow to a team of 15 researchers.

As a consequence of this growth we have organized our team into two working groups – Rock'n'Roll and 'Dust'n'Dirt – which focus on grain based and continuum based particle models respectively. In addition a cross sectional experimental group assists for validation. Consulting requests will be handled by a separate spin-off company. For the future an endowed Particulate Flow Modelling professorship is planned.

On the scientific side the topic of Particulate Flow Modelling remains highly exciting. Thanks to our publications, awards and open source activities at www.cfdem.com with more than 1000 registered users we have gained worldwide attention. I have the feeling that by now we are playing in the scientific upper league.

With these introducing words I wish you a pleasant reading!

Sincerely,

Stefan Pirker | stefan.pirker@jku.at

EDITORIAL 2 | ROCK ´N´ ROLL

Dear Readers,

Pausing a moment and having a look back at the recent years of the CD-Lab on Particulate Flow Modelling we see plenty of stories of success:

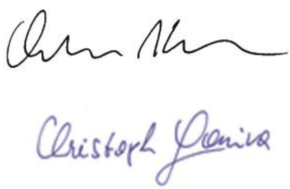
PhD theses were performed successfully and now find application in real industrial plant operations, new colleagues could be attracted, interesting publications were placed and contacts to many excellent researchers all over the world could be established. As an extraordinary highlight the broad international response to our open source DEM project (LIGGGHTS) and the CFD-DEM coupling (CFDEM coupling) should be mentioned.

Following the vision of creating a reliable, fast and multi-purpose software tool for granular and coupled fluid-granular systems enormous development work has been performed. Our aspiration to provide a trustworthy “workbench” for researchers and scientists consequently leads to sound testing and validation work performed on a regular basis.

As a clear commitment to sustainability and quality of our work we have decided to allow space for code development within a spin-off company to be founded January 2012. This step will allow us to keep the focus on academic work within the CD-Lab and perform profound code development within the spin-off.

Sincerely,

Christoph Kloss | Christoph Goniva



christoph.kloss@jku.at | christoph.goniva@jku.at

EDITORIAL 3 | DUST ´N´ DIRT

Dear Readers,

One year ago I was assigned as team leader of the Dust'n'Dirt group. In the last year this group once again has grown by two PhD students, who are funded by two new industrial partners (Borealis and Siemens Mining). Andreas Aigner will study fluidized and moving bed experimentally and numerically with particle based methods (CFDEM) and Gerhard Holzinger will analyse the separation of mixtures by floatation.

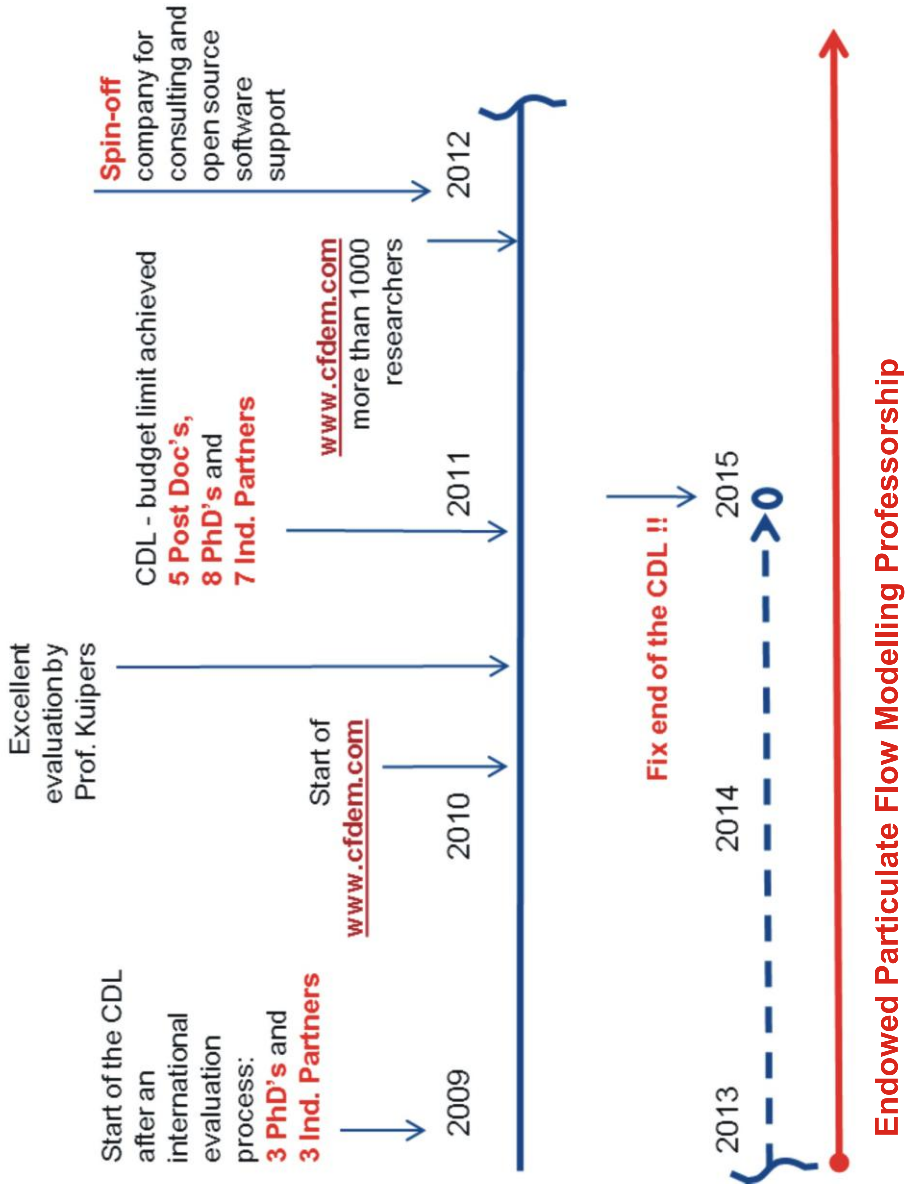
During this year I have observed remarkable progress, which I want to emphasize by the following examples. (1) The hybrid model is now ready for industrial scale applications. (2) The development of a new diffusor shows a pressure recovery of up to 30% in a cyclone. (3) New data processing routines allow a better comparison between coal injection experiments and corresponding simulations. (4) The coupling of Lattice Boltzmann and Finite Volume simulation indicates a tremendous speedup to conventional FV models. (5) By analytical calculations new boundary conditions for continuum particle model have been derived, which are a significant improvement to the state of the art.

Finally, these examples show that it is important to me to sustain the three main pillars of the CD-Laboratory, i.e. (1) an analytical analysis of the problem, (2) numerical modelling and (3) experimental validation. I am confident that with this approach we are on the right way!

Sincerely,

Simon Schneiderbauer | simon.schneiderbauer@jku.at

CD-LAB'S TIMELINE



CONTENTS

ROCK ´N´ ROLL

DUST ´N´ DIRT

EXPERIMENTS AND ADD TOPICS

ROCK ´N´ ROLL | LIGGGHTS

We want LIGGGHTS to be a high-quality code in order to deliver high-quality work to our industrial partners. Our work is increasingly trusted by the scientific and the open source community. As a result, LIGGGHTS will be part of the official Ubuntu Linux packages as of the next release in spring 2012. A few highlights of the year 2011 are given below.

Firstly, the topic of **picturing non-sphericity** plays an important role. The multi-sphere or clump modelling capability has been enhanced and parallelized. An example (a granular avalanche) can be seen in Fig. 1.

Experimental validation and quality control play a tremendous role in proving a sound simulation code. Fig. 2. shows the comparison of simulation results and experimental results of hopper flow-rate for 24 cases and two versions of LIGGGHTS (1.2.7 and 1.4.4). As part of our test harness initiative, we periodically re-run these tests to make sure we pinpoint bugs and ensure validity of our simulation models.

Last, but not least the applicability of our simulation methods depend on their **speed, efficiency and parallelization**. Fig. 3 shows the strong (fixed size) scalability of LIGGGHTS for the same hopper discharge. Being the first DEM code available to the public to introduce an MPI load-balancing technique, we can push these limits further.

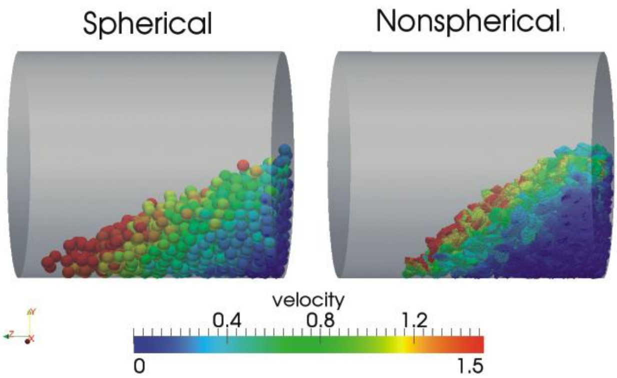


Fig. 1: Comparison of granular avalanches of spherical (left) and non-spherical particles (right) simulated with LIGGGHTS.

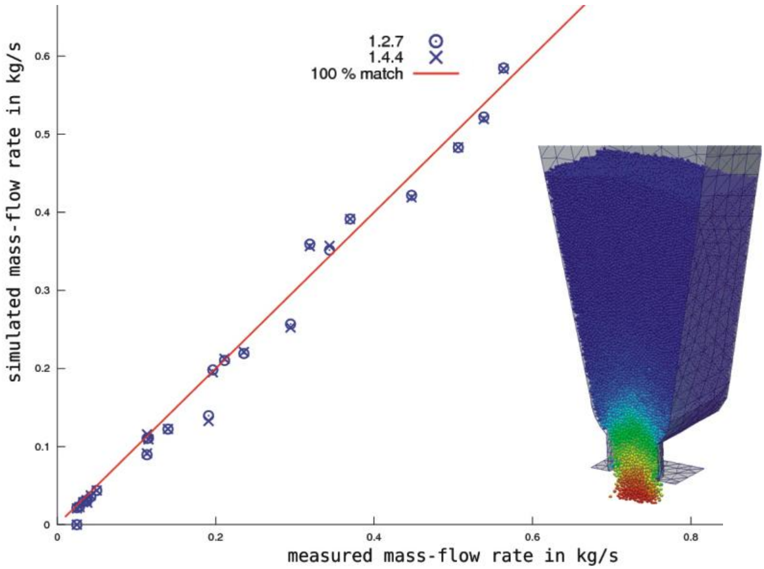


Fig. 2: Experimental validation and version consistency of LIGGGHTS for hopper discharge mass flow rate

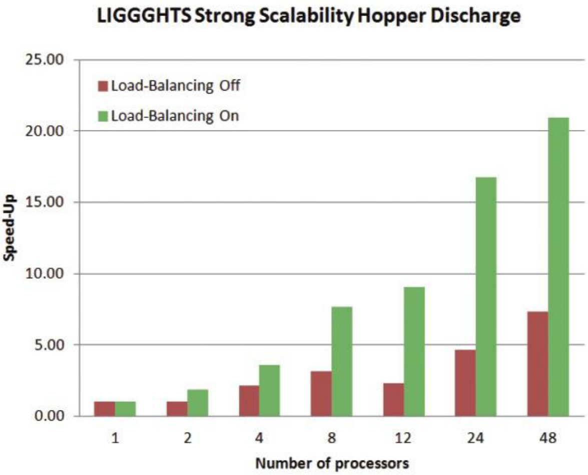


Fig. 3: Strong scalability with and without dynamic load-balancing (hopper discharge)

Christoph Kloss | christoph.kloss@jku.at

ROCK ´N´ ROLL | NON-SPHERICITY, TEST HARNESS

Multisphere Method

Most objects that are simulated using the DEM are non-spherical and have rolling and frictional behavior that differs from the behavior of spheres and thus can only be modelled to a certain extent using spheres. LIGGGHTS however offers the capability to integrate multiple bonded, overlapping particles, this is called multisphere method.

Now it is possible to use data obtained by laser scans to approximate arbitrary non-convex objects by multiple overlapping spheres. The approximation preserves the volume of the object which means that simulations taking advantage of the CFD-DEM coupling of LIGGGHTS and Open FOAM experience realistic volume displacement of the simulated fluid.
The more spheres one uses to approximate an object the more accurate the approximation gets, statistically. Since guessing how many spheres need to be used is not a good way minimize the number of spheres and with it the additional CPU time needed for a given simulation, a tool has been developed to easily determine the optimal number of spheres for an approximation. This tool is based upon the fact that many approximations involving different numbers of spheres can be calculated in a short time, thus allowing for quality-comparisons w.r.t. different sphere counts.

Test Harness

The test harness that has been announced in last years annual report is an automated quality assurance tool for the LIGGGHTS code to ensure the integrity of the software and the numerical models. It has been developed to automatically run test-simulations stored in a database and to analyze the results obtained.
The test harness enables us to detect more bugs before new releases, compare different models, validate models via standard experiments, easily test and analyze scalability and to effortlessly compare input-scripts to pinpoint bugs that have eventually been found.

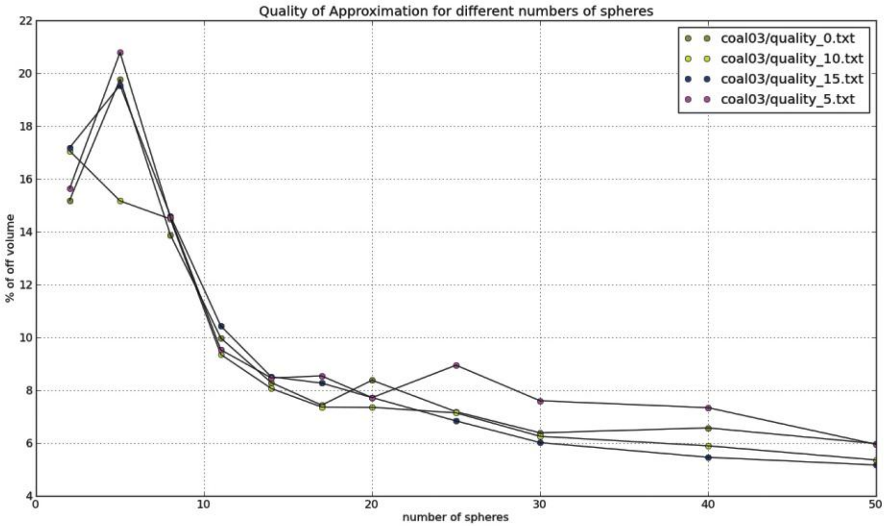


Fig. 1.: Plot of the error of approximations with varying overlap of spheres and number of spheres

Fig. 2.: Laserscan of a stone and it's approximation by 9 and 30 spheres

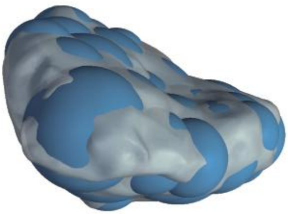
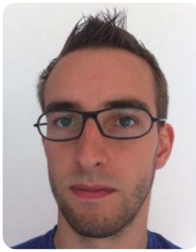


Fig. 3.: Approximation of a concave particle by 30 spheres

Stefan Amberger Supervision: Kloss | **Michael Friedl** Supervision: Kloss, Goniva



stefan.amberger@jku.at

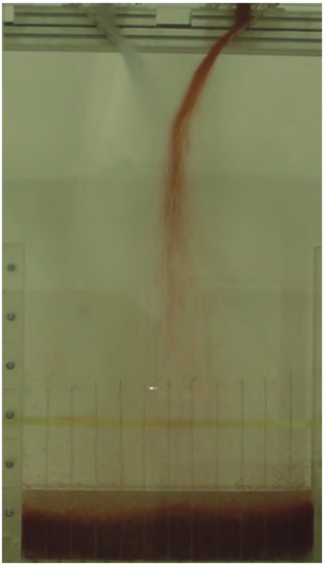
ROCK ´N´ ROLL | GRANULAR MIXING

Mixing of granular materials is an important field of study for a wide range of industrial applications. Many processes are charged by pouring granular base substances into reactors, ovens or other industrial facilities. To control their operation, the control over the spatial distribution of these substances is important. The present studies aim at predicting such distributions by numerical simulation.

Experiments with two discharging hoppers were performed. The hoppers were filled with glass granulate of different diameters and allowed to fully discharge. In the resulting bed, the spatial distribution of each particle species was then measured. These experiments were used as verification data for simulations.

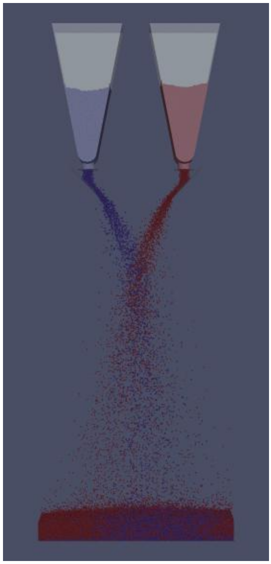
DEM simulations were performed with the same geometries as in the experiments. Again, the spatial distribution of the particles in the bed was analysed. Principal features of the experimental results, such as a concentration of the bigger, heavier particles in the middle of the bed were recovered in the simulation results. Thus, prediction of distributions in more complex geometries can be assumed to be reliable.

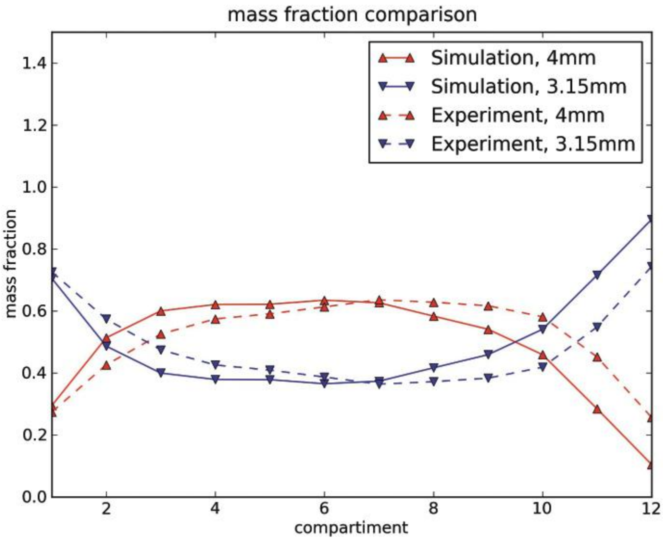
Of course in real world applications the granular media to be poured into industrial facilities are rarely spherical. Therefore, further experimental and numerical studies with non-spherical particles will follow to validate our models.



Left: The experimental setup with the two colliding particle species coming from hoppers (not shown).

Right: A screenshot of a simulation run





Preliminary results for the spatial distribution of two different particle diameters

Supervision: Christoph Kloss

Philippe Seil | philippe.seil@jku.at

ROCK ´N´ ROLL | CFDEM

Having a glace at a recent publication (Zhou et al. (2010), JFM) we read: "Coupled particle–fluid flow can be observed in almost all types of particulate processes which are widely used in industry. Understanding the fundamentals … is of paramount importance to the formulation of strategies for process development and control."

This motivates following the way started early 2010, developing an open source tool for coupled CFD-DEM simulations.
Digging a bit deeper we find: „However, the origin and the applicability of these models are not clearly understood." So there is still work to be done, and we want to meet this challenge!

Within the last year we made great progress in developing a multi-purpose CFD-DEM coupling. Within one framework we can handle a huge variety of flow problems. This multi-disciplinarity on one hand helps gaining a broad understanding of fluid-granular flow and on the other hand raises the demand of comprehensive physical modelling rather than "quick fix" for one problem.

Together with our partners from industry and academia we have been conducting studies on e.g. dust propagation in industrial plants (Fig. 1), flow situation in rotating ball mills (Fig. 2), local erosion phenomena in river structures (Fig. 5), fluidization of fluid granular systems such as blast furnace or chemical reactors (Fig. 3), particle-bubble interaction in flotation cells (Fig. 4), process control by surge conveyors, and several others.

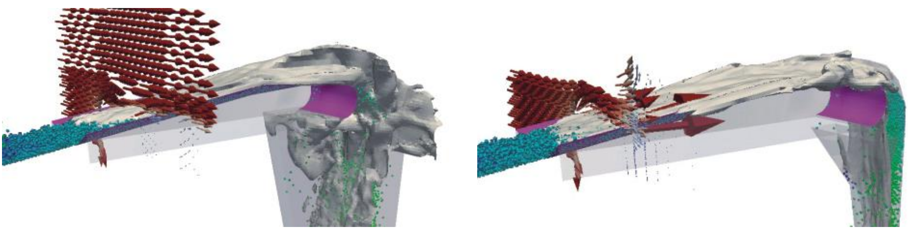


Fig. 1: Optimization of dust emission at a transfer chute. Original design (left) and optimised design (right).

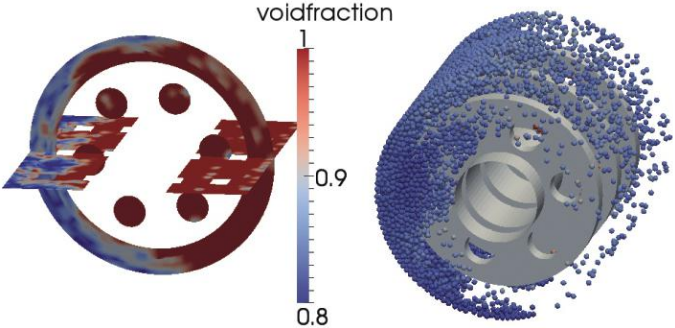


Fig. 2: Local ball-loading (left) and flow pattern (right) in a rotating ball mill.

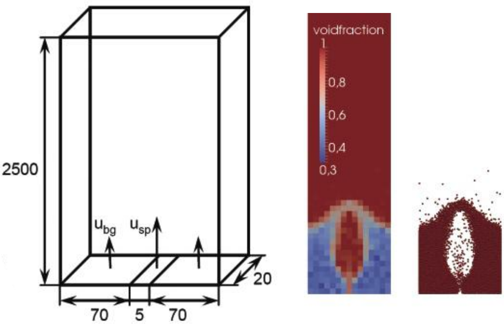


Fig. 3: Flow in spout fluidized bed reactor.

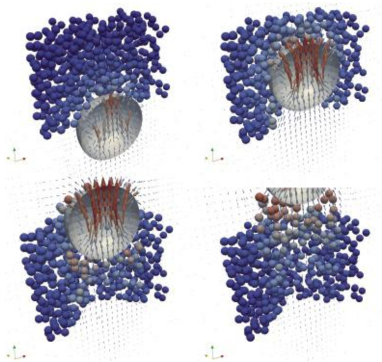


Fig. 4: Bubble-particle interaction in a flotation cell.

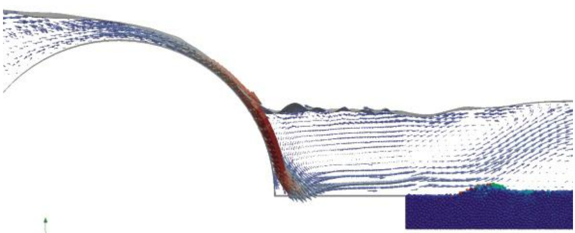


Fig. 5: Local erosion pattern behind a weir.

Christoph Goniva | christoph.goniva@jku.at

A resolved CFD-DEM method within CFDEM

Another application example within the framework of CFDEM is the so-called immersed boundary method. This resolved method is used for depicting the interaction between solid bodies and a fluid phase where the bodies are significantly bigger than the fluid-cells.

As customary for the CFDEM coupling, the motion of the solid is calculated with the DEM code LIGGGHTS and the computations of the dynamics of the fluid phase are done with OpenFOAM® (cf. Fig. 1). The calculations are carried out side-by-side, and after a certain number of time steps data is exchanged: the CFD solver obtains the information of the location and velocity of the solids while LIGGGHTS gets the drag force acting onto each body.

The particularity of this method consists of the fact that both phases are represented by only one velocity and pressure field. Even though the particles move through the domain, no mesh-deformation is necessary.
As already mentioned, the represented bodies must be bigger than the resolution of the CFD-grid. It was shown that at least 8 cells per diameter are needed in order to be able to display the dynamics of the interaction accurately. In many cases this requirement leads to a very high number of cells. In order to keep the number of cells as small as possible dynamic local mesh refinement can be applied. This tool already exists in OpenFOAM® and is a particularly efficient method for minimizing the number of mesh cells (cf. Fig. 3).
Various validation examples, published recently by our group, show the correctness of the implementation (cf. Fig. 2, Fig. 4).
The next goal lies in the parallelization of the method, such that it becomes applicable to large scale problems.

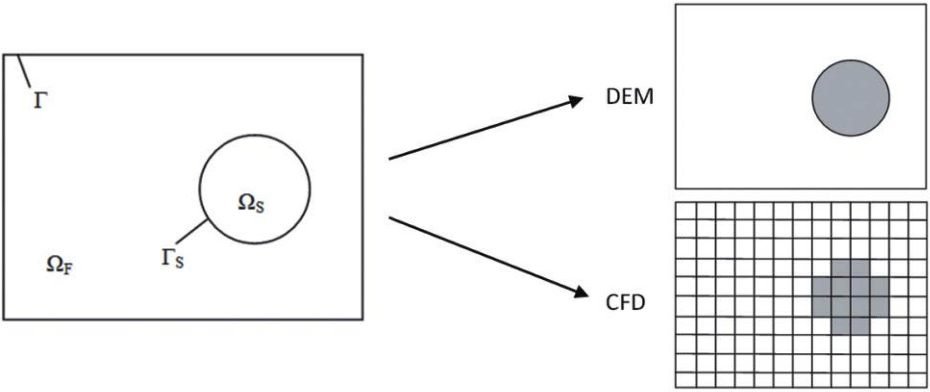


Fig. 1: Particle in the fluid domain (left), representation of the particle in DEM (top) and CFD (bottom)

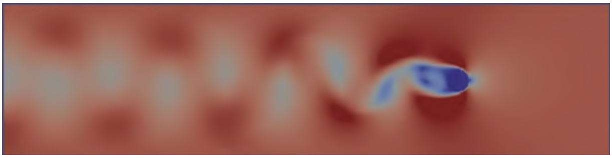


Figure 2: Karmán-vortex street developing in the wake of a cylinder

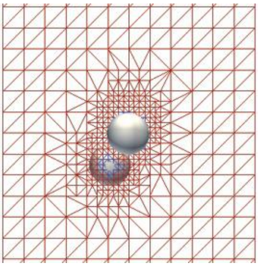


Figure 3: Dynamic local mesh refinement

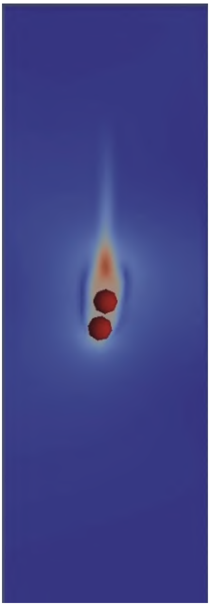


Figure 4: Settling of two particles

Supervision: Christoph Goniva

Alice Hager | alice.hager@jku.at

ROCK ´N´ ROLL | HOPPER AND SCREW

Plastics manufacturing, steelmaking, sinter techniques, … – granular media are involved in a wide range of modern production processes. Hence hoppers and screw conveyers are very important components in such industrial facilities and their continuous development and especially the possibility of simulating them are in the interest of the scientific community and our industrial partners.

The DEM-simulation software LIGGGHTS as well as the coupling to finite volume methods CFDEM already provide tools that cope very well with these complex problems. In course of the collaboration with Borealis we investigate improved models and implement this new features to our software toolbox. In case of hopper flows we implement new approaches for material properties like stickiness, in order to locate and avoid stagnant areas in the process. On the other hand the simulation of a screw conveyer including strong gas flow is a challenge for the CFDEM coupling.

To show the validity of the models we compare the simulation results with measurement data available in literature or 'in-house' experiments. Preliminary results are encouraging, although we investigate further model improvements.

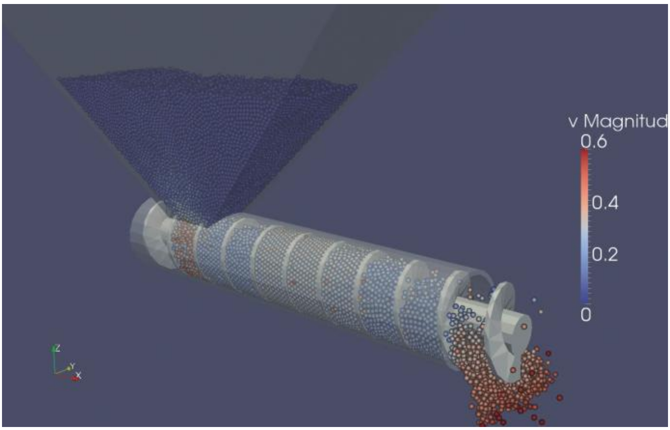


Fig.1: DEM simulation of a hopper and screw conveyer

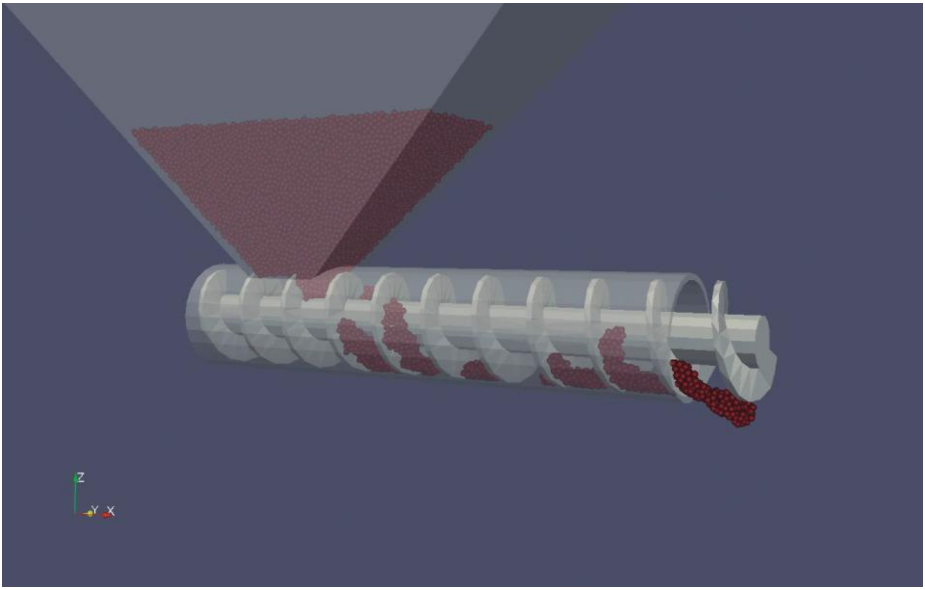


Fig. 2: Hopper + screw conveyer simulation done with LIGGGHTS. In addition a cohesion model was implemented which causes reduced particle flow rates or even a totally clogged outlet.

Supervision: Schneiderbauer

Andreas Aigner | andreas.aigner@jku.at

DUST ´N´ DIRT | EULERIAN PARTICLE MODEL

Fluidized and moving beds are widely used in chemical and process industries. Computational fluid dynamics is a valuable technique for predicting the flow behavior of these systems, as it is necessary for scale-up, design, or optimization. While state of the art continuum models capture the behaviour of interparticle-collisions dominated fluidized beds adequately, they fail to predict the flow in friction dominated moving beds. For example, these models show a decreasing mass flow rate from a hopper with decreasing bed height in the inertial stress regime. Additionally, the current modelling of the solids wall shear stresses does not predict Coulomb friction in the limit of all particles slide at impact correctly.

In cooperation with the Borealis AG a comprehensive continuum approach for the numerical modelling of fluidized and moving beds is under development. In this first year the focus of research was laid on the deficiencies mentioned above.

Firstly, we were able to deduce a novel approach for the wall boundary conditions for the particulate phase, which accounts correctly for both collisional regimes, i.e. sliding and non-sliding. Utilizing these boundary improved the accordance of the numerical model to the measurements significantly in a single-spout fluidized bed (Figure 1). Secondly, a general frictional model was developed, which shows a constant mass flow rate from hoppers in the inertial stress regime (Figure 2). Dilatation of the moving bed in highly sheared areas is also precisely predicted by the model. Finally, it was shown that the numerical modelling of combined fluidized bed and moving bed reactors with spillways and screws within one simulation is feasible (Figure 3). This allows to focus on potential regime changes of the particle flow (fluidization in the moving bed or de-fluidization in the fluidized bed) that might pose risks to a smooth process operation.

The next steps include additional validation by inhouse fluidized bed and moving bed experiments. In a long term objective it should be feasible to scale up the model to plant scale.

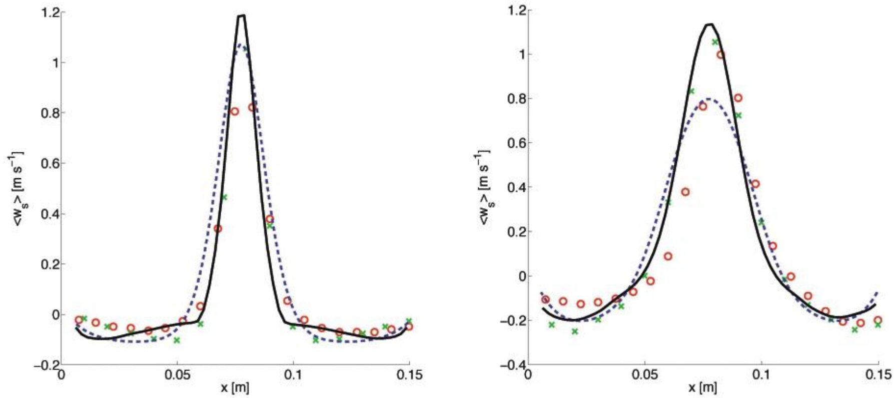


Fig. 1: Sensitivity of the time averaged vertical solids velocity <w_S> to the formulation of the wall boundary conditions: - - - state of the art; — CD-Lab . ○ represent PIV and × PEPT measurements (van Buijtenen et al., 2011).

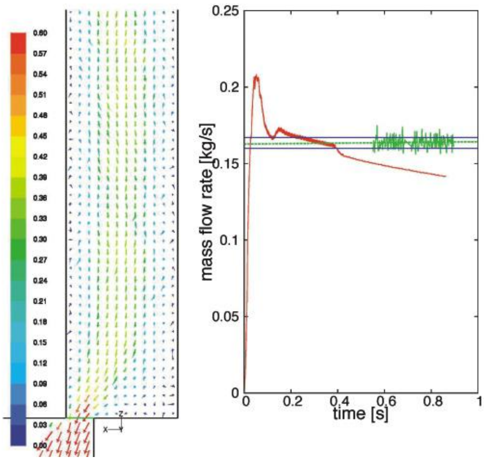


Fig. 2: Left: Discharge of glass spheres from a bin. **Right:** mass flow rate from the bin: state of the art model (red), CD-Lab (Green); confidence interval of measurement (blue).

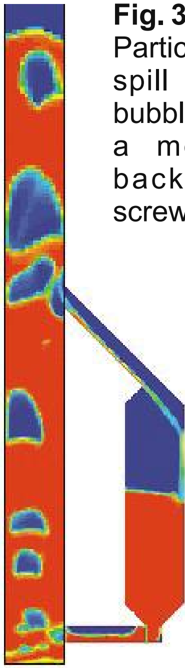
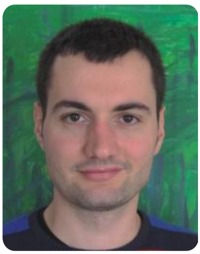


Fig. 3: Regime changes: Particle transport over a spill way (top) from bubbling fluidized bed to a moving bed and backtransport by a screw (bottom).



Simon Schneiderbauer | simon.schneiderbauer@jku.at

DUST ´N´ DIRT | HYBRID PARTICLE MODEL

General

The hybrid particle model, which is developed in cooperation with the Polysius AG, is a numerical model for the simulation of particle conveying and separation systems. The basic version of the hybrid model (EUgran+) is augmented by additional physics, for example: drag law valid in particle strands and particle agglomeration.

The hybrid model

The hybrid model combines an Eulerian granular approach and a Lagrangian approach. On the one hand, physical effects are computed in the model which demands lower computational effort. On the other hand, this approach allows to include Lagrangian information in continuum models.

Benefits of the hybrid model:

- Valid in high and low mass loaded regimes
- Handles particle-particle interaction
- Poly-disperse particle
- High computational efficiency

Highlights

The EUgran+ model could be validated by our labor cyclone and double looping, shown in Fig. 1 and Fig. 2. The numerical results, at the cyclone, show good agreement with experiments and analytics.
Highlight of my year 2011 was the conference of Multiphase Flow on the island Kos. It was a great conference, with very interesting topics. I could see that the scientific work at the Christian Doppler Laboratory on Particulate Flow Modelling is at a very high level.

Actual and future work:

The main points for the hybrid model to give accurate results, could be identified and is still under development. The two important points are:

- Development of a simple agglomeration model, to capture global effects of agglomeration
- Development of a drag model which allows the creation of particle strands, as can be seen in cyclones

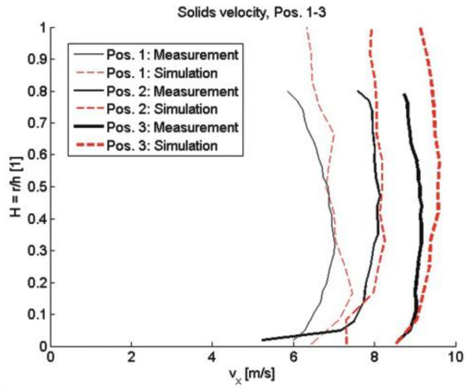


Fig. 1: Velocities at 3 positions after double looping

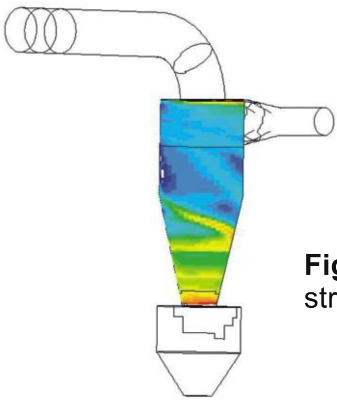


Fig. 2: Particle strand in our cyclone

David Schellander | david.schellander@jku.at

DUST ´N´ DIRT | HYBRID TURBULENCE MODEL

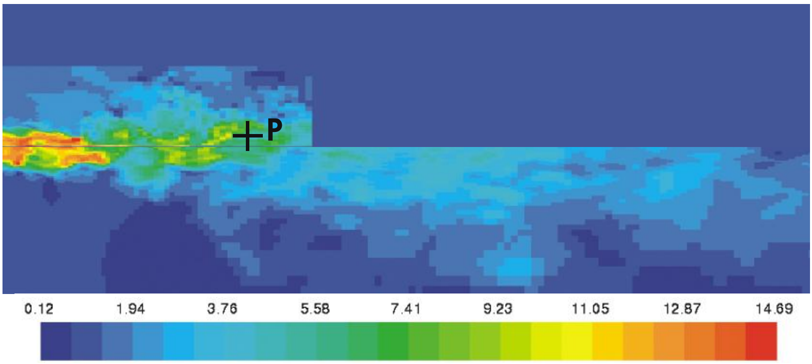
Thanks to an ambitious team of experienced senior scientists, who forward independent research and also supervision, I found some time for my own research. Developing new numerical models is a pleasing luxury beside all this administrative and organizing stuff that comes along with my position as head of the CD-Lab.

In a multitude of flow situations the unsteady behaviour of coherent turbulent structures or vortices determines the critical core phenomenon of the process under consideration. In continuous casting of steel the interaction between irregularly occurring vortices at the interface between metal and slag might trigger unfavourable slag entrainment. In pulverized coal injection the interaction between the particle jet and the attacking turbulent eddies determine jet dispersion and consequently reaction turnover.

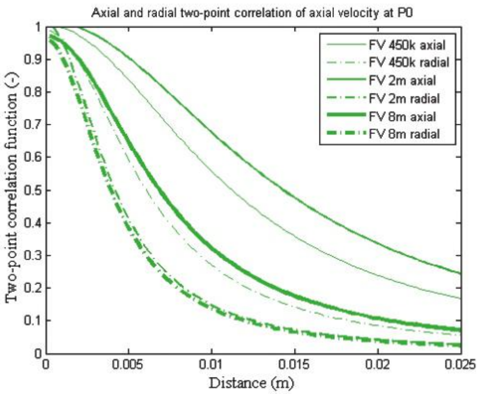
Numerically resolving these energy containing vortices requires a fine grid resolution and enormous computational resources. In the present research effort a vision of a hybrid turbulence model is tested. A Lattice Boltzmann (LB) based flow model is embedded into a Finite Volume (FV) based flow simulation. For this combination a LB solver had to be added to a conventional CFD software. New boundary conditions had to be developed for the communication between the worlds of Boltzmann and Navier-Stokes.

In order to validate this new approach a round turbulent jet has been considered. In a series of simulations the influence of grid resolution and corona communication condition on the results of this hybrid turbulence model has been investigated.

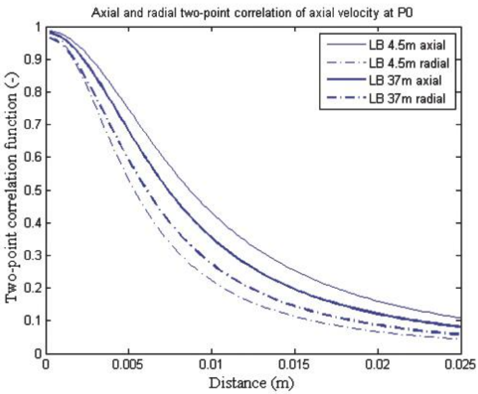
First results indicate that this hybrid turbulence model could speed up simulations tremendously if compared to conventional FV based models. At the same time the results of the hybrid model agree with corresponding expensive FV simulations with a fine grid resolution.



Conventional CFD simulation (bottom) with embedded Lattice-Boltzmann simulation (top)



Evaluation of turbulent length scales at point P – conventional simulations (green, based on 450k , 2m and 8m cells) exhibit explicit grid dependencies while the hybrid turbulence model (blue, based on 4.5m and 37m cells) is nearly independent on its resolution. Note, that the 37m hybrid model is significantly faster than the 8m conventional model.



Stefan Pirker | stefan.pirker@jku.at

Minimizing the pressure drop

The kinetic energy stored in the tangential component of the vortex motion is lost due to downstream dissipation. This is also the reason for decreasing pressure loss of the cyclone with increased mass loading and/or wall roughness: In this case the swirling motion is not as strong and the losses due to subsequent dissipation are smaller.

The attempts to regain the kinetic energy from the swirl and thus to minimize the pressure loss of the cyclone are almost as old as cyclones themselves. Most configurations aim to decrease pressure loss by installing diverse kinds of rectifying vanes or other built-in components inside the cyclone body or the vortex finder. Some improvements can also be made by changing the vortex finder geometry or position. However, there are certain types of cyclones which work under harsh conditions (high temperatures, danger of corrosion etc.), where such vanes or similar built-in components inside the cyclone body or the vortex finder should be avoided as far as possible. For this reason, a pressure recovery type diffuser on the top of the cyclone body is considered.

Furthermore, in cases of gas-particle flows with high humidity level there is a danger of settling and sticking down of particles on horizontal planes. For this reason the horizontal planes within the apparatus should be avoided as well. Therefore the second configuration has no horizontal planes. The radial diffuser on top of the cyclone body is arranged under the angle of 30° with respect to the horizontal. The two considered diffuser geometries are shown in Fig. 1 (see next page).

Both numerical simulations and experimental measurements are used to find the optimal dimensions of the diffuser. With the diffuser pictured on the next page up to 30% pressure recovery is possible (Fig. 2).

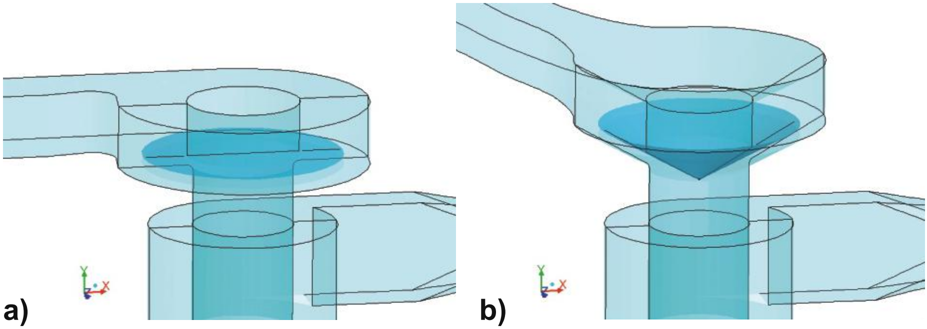


Fig. 1: Radial diffuser geometries, **a)** “horizontal”, **b)** “inclined”

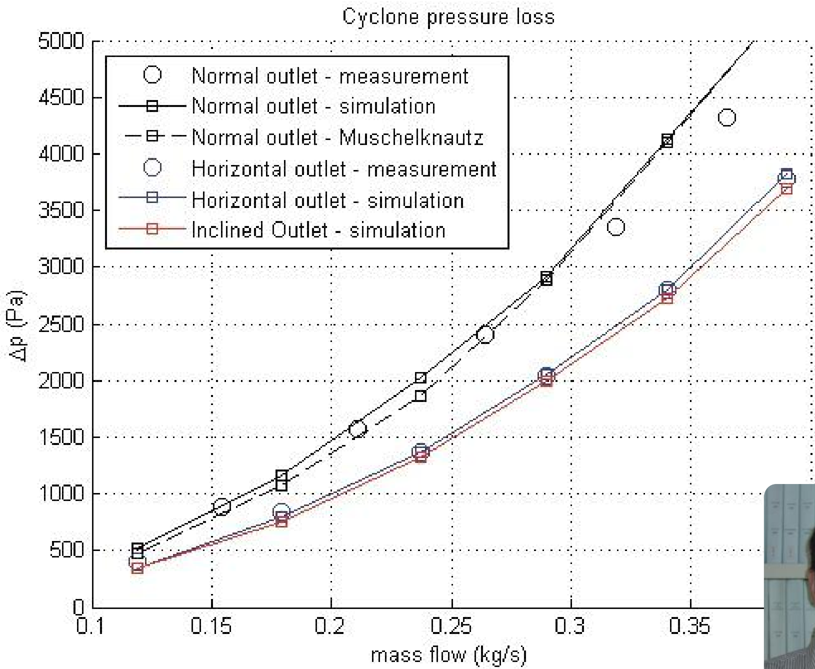


Fig. 2: Cyclone pressure loss

Damir Kahrimanovic | damir.k@jku.at

EXPERIMENTS | POWDER INJECTION

Due to the continuous growth of the CD laboratory there is also an increasing demand on experimental data. The activities in our fluid mechanics laboratory have therefore experienced a significant rise in the last three years. It is a strong commitment that all new projects within the CD Lab and associated research activities contain dedicated experimental tasks and I am looking forward to new challenges and exciting experiments.

High Density Powder Injection

In collaboration with voestalpine Stahl Donawitz GmbH (VASD) the injection of pulverized coal into a blast furnace should be investigated. It is believed that the dispersion behaviour directly behind the injection lance determines the combustion efficiency. To get a better understanding of the injection process a lab-scale experiment under cold conditions was designed to investigate fluid flow and particle dispersion directly downstream the injection lance.

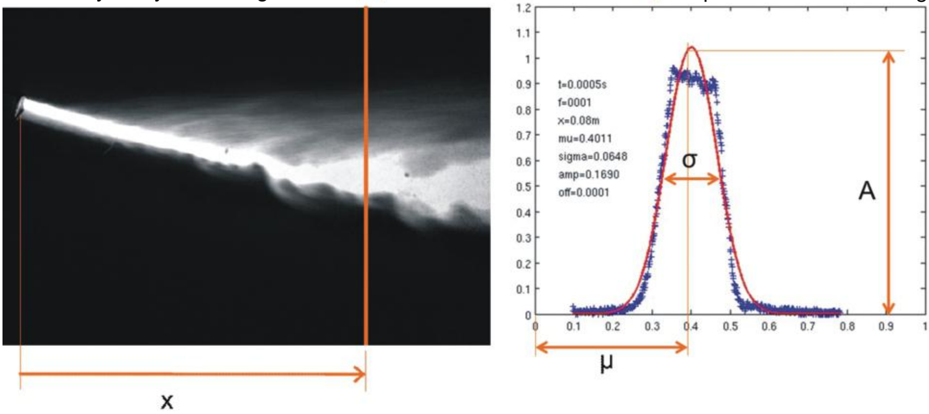
In a second approach the same flow situation is pictured by numerical simulations. Thereby, a Finite-Volume based calculation of the gas phase with a hybrid Eulerian-Lagrangian particle model is coupled with a Lattice-Boltzmann (LB) based LES simulation which serves as a subgrid model to resolve flow turbulence correctly.

From an academic point of view it is now interesting how the huge amount of data produced by both, high-speed camera and LES simulations, can be compared on a qualitative and quantitative basis. Therefore different ways of data processing methods not relying on any physical quantity are investigated.

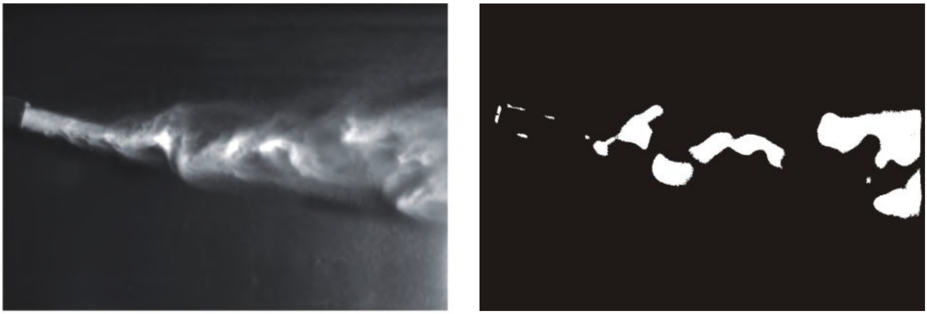
For the practical application the collected knowledge is now used to sketch new nozzle designs to improve dispersion behaviour.

Luminosity analysis of image

optimizer for curve fitting



Particle cluster detection



Pneumatic test rig

Stefan Puttinger | stefan.puttinger@jku.at

EXPERIMENTS | LIQUID SLOSHING

In collaboration with Siemens VAI, voestalpine and RHI within the K1Met project we investigated the phenomenon of sloshing in a partially filled vessel. Because these oscillating movement can cause resonance effects and in further consequence structural weaknesses of the whole suspension system.

Our study focused on a small-scale experiment which should be capable of reproducing the physical key phenomenon – the resonance phenomenon, the beat.

In course of our research we could show experimentally the dependence of the beat on the mechanical eigen-frequency as well as the fluid wave frequency. Moreover, analytical estimates and numerical predictions correlate well in principle with the experiments with respect to sloshing mode frequencies and the fluid-structure resonance phenomenon.

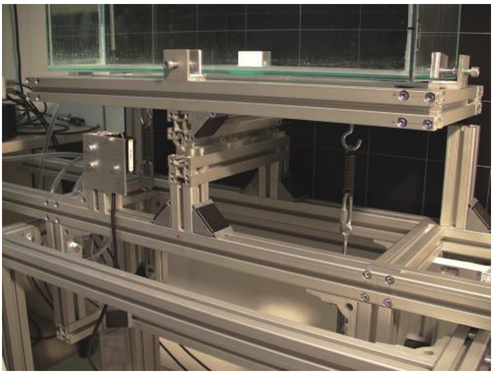


Fig. 1: Tank with seesaw and sensors

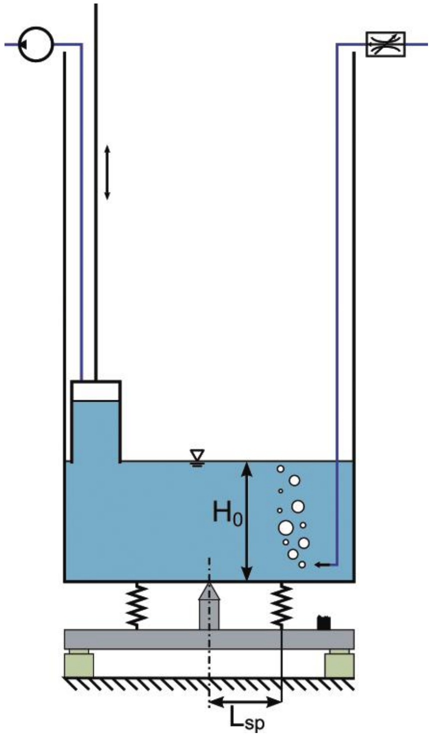


Fig. 2: Sketch of the experimental set up

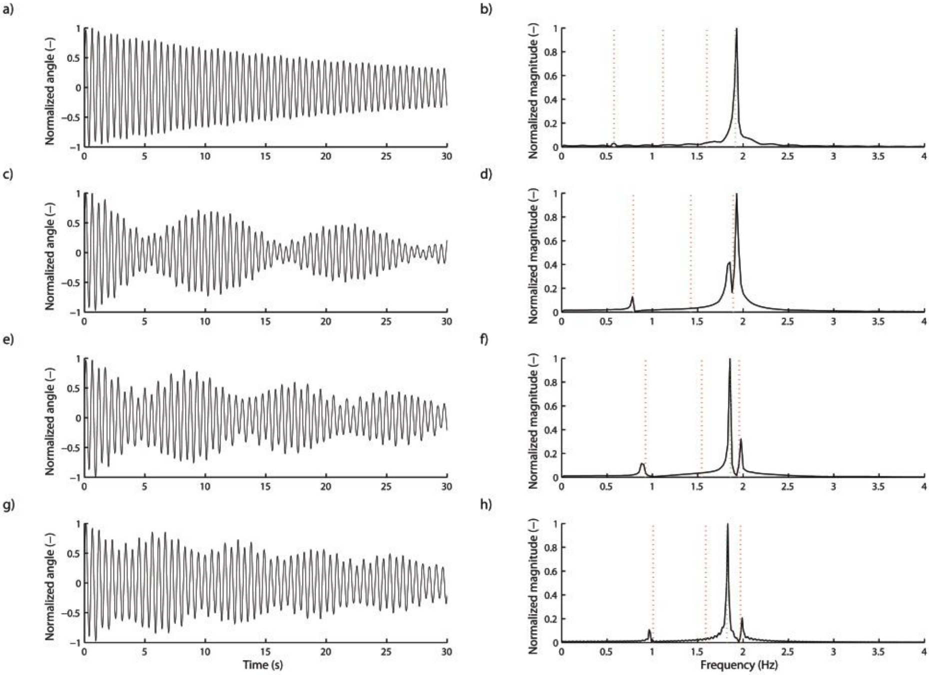


Fig. 3: Experimentally obtained monitors of tilting angle (left) and corresponding frequencies (right) for initial excitation-induced sloshing against the fill level (top to bottom: fill levell is 50, 100, 150 and 200 mm). In addition, the analytically obtained characteristic sloshing frequencies (red) and the structural eigenfrequency (green) are indicated by vertical lines.

Supervision: Stefan Puttinger

Andreas Aigner | andreas.aigner@jku.at

ADD TOPICS | SEDIMENT TRANSPORT

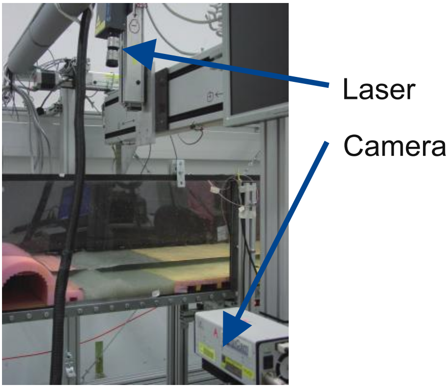
Sediment transport is a big issue within environmental engineering. It deals with problems such as soil erosion as well as the drift of snow leading to the formation of snow cornices and further snow deposits at leewards mountainsides and the ultimate formation of avalanches. In addition, erosion by floods or river erosion forced due to hydraulic structures is a further actual topic.

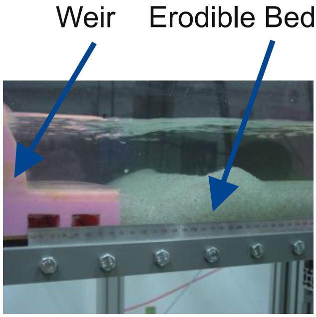
River erosion caused by massive obstacles, like hydroelectric power plants and weirs, effect on the one hand river ecosystems and have on the other hand a non negligible effect on the plant efficiency. Therefore, extensive experience and physical insight into the hydraulics, sediment transport and morphological processes is crucial.

In order to get a deeper understanding experimental and numerical investigations of local scour development downstream of a weir were performed. An experiment setup with an Acoustic Doppler Velocimetry (ADV) and Particle Image Velocimetry (PIV) for flow and a flush mounted Constant-Temperature Anemometry (CTA) for wall shear stress measurements was developed. Furthermore, a spatial and temporal erosion detection system with PIV was developed.

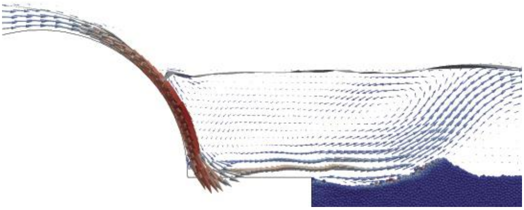
River morphology is composed of complicated interactions of turbulent flow, the free water surface, particle motion and its resulting bed configurations. In order to deal with this, a 4-way coupled Open Source CFD-DEM VOF Solver was developed, which can deal with various turbulence models. The obtained numerical results were compared with the experimental data and show a good match. As the Model bases on a general approach, it can easily be adapted to other sediment transport problems.

Particle image velocimetry (PIV)

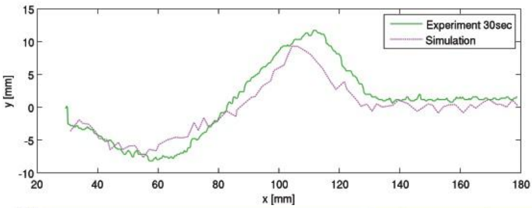




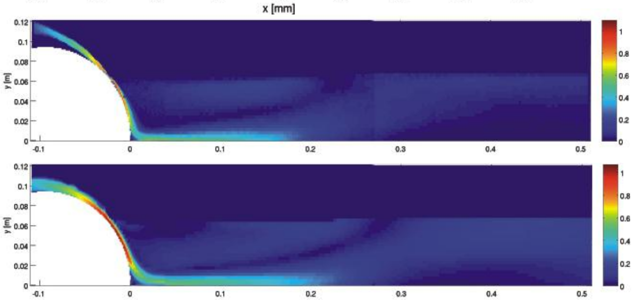
Experiment setup



CFD-DEM simulation



Comparison of the erosion pattern of experimental data and CFD simulation



Comparison of the velocity field (flow over a weir) of experimental data and CFD simulation

Supervision:
Goniva, Puttinger, Schneiderbauer

Klemens Gruber | klemens.g@gmx.net

ADD TOPICS | DIE CASTING

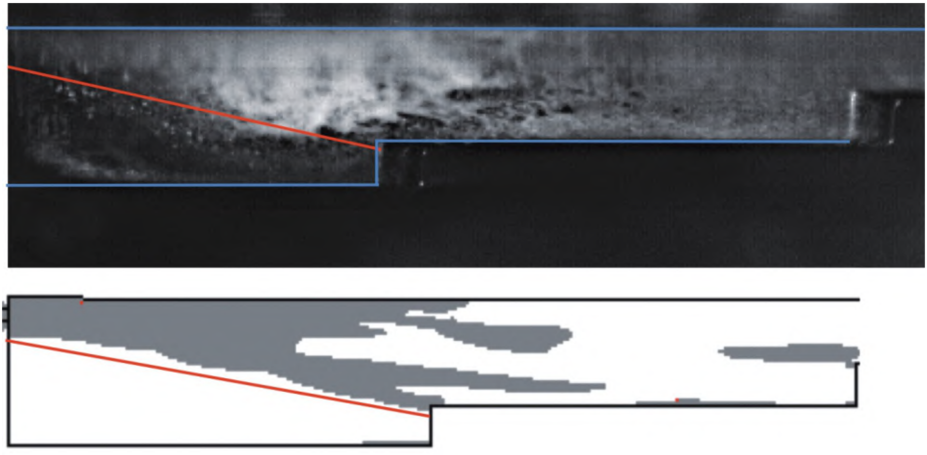
Due to its advanced degree of automation and its high productivity the high-pressure die-casting process is commercially the dominant technique for the production of aluminium die casting parts, for example, transmission housings, engine blocks and even small components like brackets, oil pump cases and water pumps.

Two main difficulties arising during the casting process are considered. Due to the high casting speed a free jet is formed at the ingate. At the surface of the jet droplets of liquid aluminium are formed. Depending on the shear forces between jet and the surrounding gas the jet disintegrates or atomizes. However, both lead to an increase of the surface ratio for the liquid aluminium, where oxidation may start. On the one hand, the collision of two disintegrated parts of the oxidized aluminium jet may cause cold runs. On the other hand, a high degree of atomization may increase the porosity of the final casting.

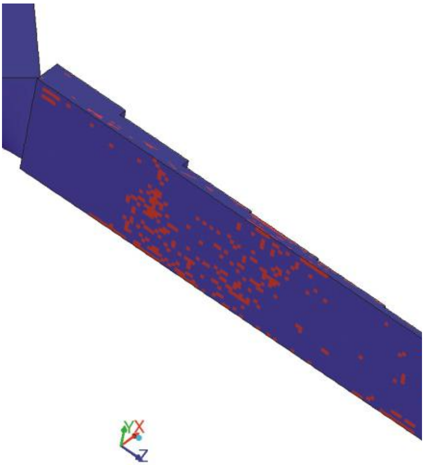
In cooperation with the **LKR** in Ranshofen the flow and filling characteristics during injection of liquid aluminum during high pressure die casting was studied threefoldly: a) analytically, b) experimentally and c) numerically. It has been shown analytically and experimentally that the geometry of the nozzle at the ingate significantly influences the process of the drop formation at the surface of the free jet of liquid aluminum. Numerically, the drop formation cannot be resolved with an acceptable computational effort. However, the simulation of high-pressure die-casting delivers an appropriate measure of the casting process and is able to predict cold runs.

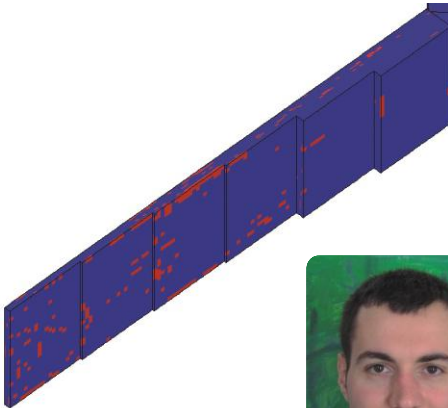
The feasibility of water modelling to experimentally simulate the high pressure die casting process has been introduced providing the basis to validate numerical models.

To conclude, this work leads to a better understanding of this challenging casting technique. The scientific findings introduce investigation techniques enabling casting professionals to optimize their casting products depending on the special demands on the final casting.

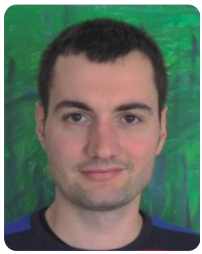


Comparison of experimental and numerical water models.





Comparison of experimental
and numerical water models.



Simon Schneiderbauer | simon.schneiderbauer@jku.at

INDUSTRIAL PARTNERS

Industrial plants such as COREX® and FINEX® are no place for experiments and yet continuous innovations are expected. Therefore we need the development und concurrent validation of fundamental models for the simulation of dense particulate flows. The simulation of particulate flows requires new techniques in experimental, analytical, mathematical and computer matters. For this purpose we rely on this CD-Lab. as our scientific partner.

Georg Aichinger | Siemens VAI Metals Technologies GmbH

The competence and expertise of the CD Lab allows us to explore complex fluid dynamic processes, which are otherwise hidden due to the harsh conditions of iron- and steelmaking. For us as an industry partner it is crucial to validate the results of the simulations with laboratory scale tests to achieve useful knowledge, which can lead to an optimization of industrial plants and processes.

Hugo Stocker | voestalpine Donawitz GmbH & Co KG

In cement plants and applications for the minerals industry a lot of fine particles like dust and powders are produced, transported and separated. As a turn-key supplier for this industry it is our aim to understand the physics behind the processes for improving our equipments and machineries furthermore. One focus in doing so lies in modern simulation techniques like CFD. Our goal is to be better than the commercially available state of the art. For us the cooperation with the CD-Lab on particulate flow modeling seems to bet the key to the actual state of scientific research.

Ulrich Voss | Polysius AG

In Gegenstromreaktoren wie dem Hochofen kann die Gasströmung nicht losgelöst von der Partikelströmung betrachtet werden. Auf diesem Gebiet hat das CD-Labor mit der Kopplung von CFD- und DEM-Tools Bahnbrechendes geleistet. Die Anwendung von CFDEM in der Darstellung der Prozesse ist für uns als ressourcenintensive Industrie ein Instrument, um das uns die Mitbewerber beneiden. Auch hier ist das CD-Labor wieder "einen Schritt voraus".

Christoph Feilmayr | Voestalpine Stahl GmbH

Granular flows can be found throughout different production steps in the refractory industry. Examples are the feeding of the green mix into all different kinds of moulds or the flow of pellets when they are burnt in a shaft kiln. To get an deeper insight, especially for process steps with an limited accessibility modelling is an important tool. In that sense we found the CD Laboratory to be the right academic partner.

Gernot Hackl | RHI AG

Since the CD-Lab resides at the upmost forefront in numerical and physical modeling of flow dynamics of particulate materials it allows a powder metallurgical company to study complex phenomena during the processing of such materials in a state where the processing steps physically are even not existing but stay in a conceptual phase. Mechanism-based understanding of the extraordinary flow behavior of refractory metal powders plays a crucial role in the design of such processing steps which is substantially being pushed forward by the CD-Lab within our collaboration.

Arno Plankensteiner | Plansee SE

AWARDS AND SELECTED PUBLICATIONS

MEC Award - PhD thesis with the Highest Impact on Industry: awarded to Christoph KLOSS for his PhD "LIGGGHTS – A New Open Source DEM Code Applied to the Corex Process" (17.11.2011)

Erwin Wenzel Preis – best PhD thesis: awarded to Simon SCHNEIDERBAUER for his PhD "Modellierung der Schneedrift im alpinen Gelände unter Zuhilfenahme lokaler Erosions- und Sedimentationsmodelle" (14.11.2011)

HAGER A., KLOSS C., PIRKER S. and GONIVA C. (2011) Efficient Realization on a Resolved CFD-DEM Method within an Open Source Framework, OpenSource CFD Conf. Proc., Paris, France.

PUTTINGER S., PIRKER S., KAHRIMANOVIC D. STOCKER H. and HABERMANN A. (2011) Dispersing of a highly laden particle jet in a lab-scale pulverized coal injection (PCI) experiment, Steelsim 2011 proc., Düsseldorf, Germany.

KLOSS C., GONIVA C., AMBERGER S. and PIRKER S. (2011) LIGGGHTS Open Source DEM: Models, Features, Parallelism and Quality Assurance, 8th Int. Conf. on CFD in Oil & Gas, Met. and Proc. Ind., SINTEF/NTNU, Trondheim Norway.

GONIVA C., KLOSS C., HAGER A., WIERINK G. and PIRKER S. (2011) A Multi-Purpose Open Source CFD-DEM Approach, 8th Int. Conf. on CFD in Oil & Gas, Met. and Proc. Ind., SINTEF/NTNU, Trondheim Norway.

PIRKER S., AIGNER A. and WIMMER G. (2011), Experimental and numerical investigation of sloshing phenomena in a spring mounted rectangular tank, Chemical Engineering Science, Vol. 68, pp. 143-50.

KAHRIMANOVIC D., AICHINGER G., PLAUL F. and PIRKER S. (2011), Numerical simulation of roughness effects inside a brick-lined cyclone separator, Int. J. of CFD Case Studies, Vol.9, pp. 5-17.

Imprint:

CHRISTIAN-DOPPLER LABORATORY ON PARTICULATE FLOW MODELLING
T +43 (0)732/2468 6477 | **F** +43 (0)732/2468 6462 | **W** http://www.particulate-flow.at

P | Altenbergerstrasse 69 | 4040 Linz | Austria