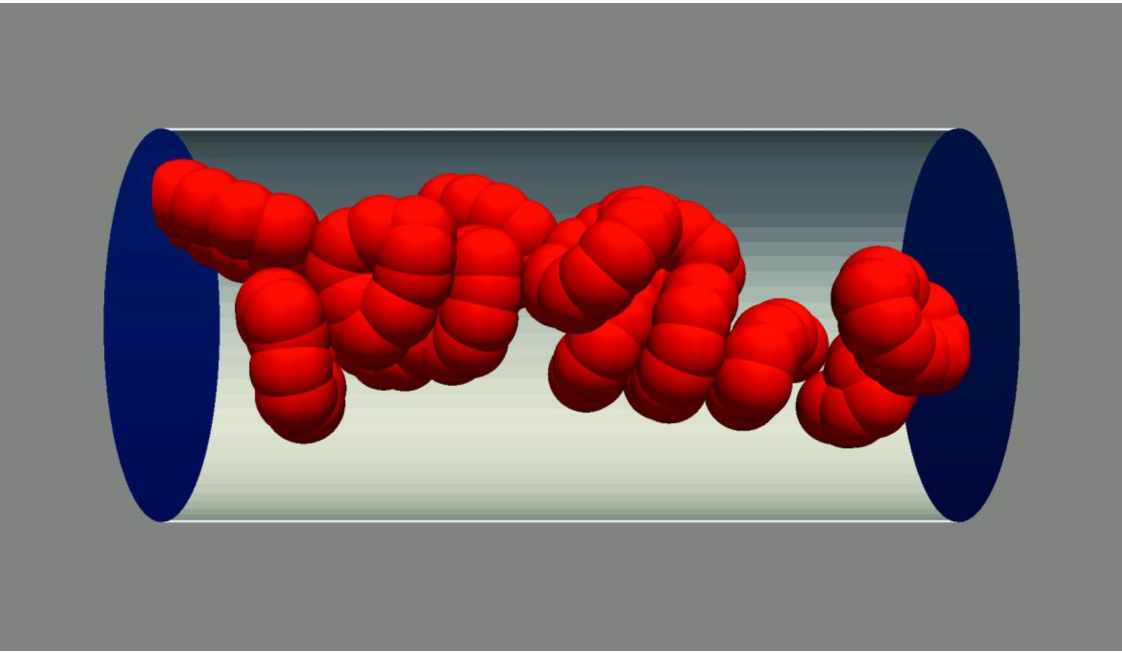


JKU | Department of Particulate Flow Modelling

Annual Report | 2019

JKU DEPARTMENT OF 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: A reduced-order digital model of deformable Red Blood Cells (RBCs) – it has been a huge step from its algorithmic implementation towards a sound physics-based validation – something which earned us our first journal publication in bioengineering © A. Balachandran Nair & M. Saeedipour

EDITORIAL

Dear Readers,

What is digitalization? In the past years we got more and more involved in discussions about this trendy buzzword. What's the potential of digitalization in an industrial context and how could we at PFM contribute to this subject?

Essentially, every numerical simulation can be considered as a digital model of the corresponding real-world flow or process. However, in nearly all cases, these digital models run orders of magnitude slower than their real-world's counterpart.

With the development of recurrence CFD (rCFD), we see the possibility to overcome this common limitation. Here, we are able to convert digital models into digital shadows, which run concurrently and synchronized with the corresponding real-world processes. Naturally, this real-time capability opens new perspectives for the application of high-resolution simulations in online process observation and control.

At the same time, we must not forget about physics – every digital model prediction needs a thorough physics-based correction, and this holds even more for fancy reduced-order models such as rCFD.

With these introducing words I wish you a pleasant reading!

Sincerely,

Stefan Pirker | stefan.pirker@jku.at

EDITORIAL

Dear Readers,

The time has passed so quickly that this year we have already celebrated half-time of the CD-Laboratory.

Currently, the focus of the CD-Laboratory is laid on the generalization of turbulence modelling of multiphase flows. Especially, in gas-particle flows (fluidized beds, raceway) as well as interfacial flows (mold flow) an additional source of turbulence arises, which is stemming from the interaction between the phases. On the one hand, particle clusters and on the other hand, surface tension are important turbulence production mechanisms, which require accurate modelling to capture the flow behaviour in large-scale processes.

A second focus of this CD-Laboratory is recurrence CFD, which is a novel approach constituting a liable method towards accessing slow phenomena in extremely large systems.

Other projects include the efficient numerical simulation of the transport and mixing of cohesive particles, which additionally exhibit a wide particle size distribution, as well as liquid-liquid flows.

Finally, I want to thank my team for their great work and their engagement and I am looking forward to the second half of this CD-Laboratory!

Sincerely,

Simon Schneiderbauer | simon.schneiderbauer@jku.at

CONTENTS

EDITORIAL | MICRO

Dear Readers,

During the past year, we experienced an amazing journey in the field of modelling multiphase flows from microscale modelling of biological flows towards highly-resolved interfacial turbulence.

We have expanded our research towards the biomedical applications of computational fluid dynamics. Together with our industrial partners, we investigate the multiphase flows in different bio-microfluidic systems such as the blood flow transport in micro-scale environments (Figure 1) as well as the colliding atomization of two micro liquid jets in the respiratory inhalers (Figure 2). For the former and within the **Achuth B. Nair**'s PhD project, we proposed a new reduced-order model for deformable particles which is applicable to study the red blood cell dynamics suspended in blood plasma in the context of resolved CFD-DEM coupling.

Our research focus on subgrid modelling for two-phase large eddy simulation has further extended by proposing a new approach for the functional subgrid closure models. In this approach, the interfacial work done by the liquid-liquid or liquid-gas interfaces is included in the subgrid eddy viscosity in the form of a correction term. Consequently, all the eddy-viscosity-based SGS models for the two-phase LES will include the effect of surface tension on the smaller scales. This approach improves the two-phase LES of interfacial flows involving turbulence-interface interactions, fragmentation, and coalescence (Figure 3).

Let's have a tour on our modelling activities in the complex multiphase systems within the past year.

Sincerely,

Mahdi Saeedipour | mahdi.saeedipour@jku.at

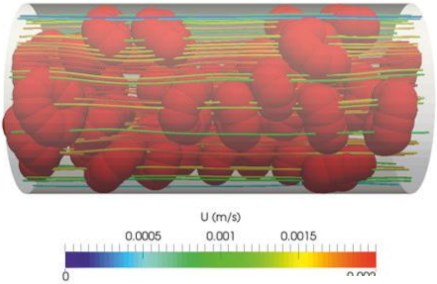


Fig.1: Blood flow transport in a micro-channel with the hematocrite of 0.45. The red blood cell dynamics are coupled with the blood plasma using a reduced order model for the deformable particles in the context of resolved CFD-DEM.

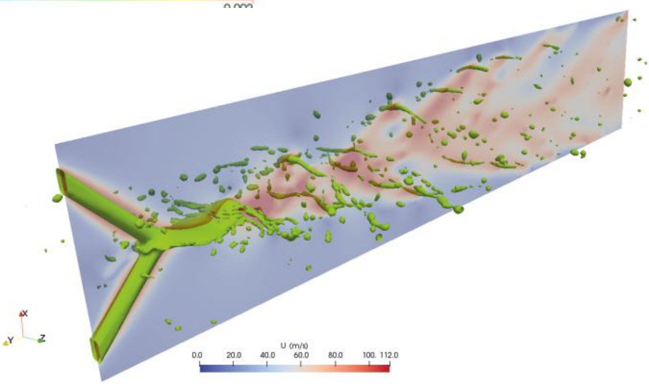


Fig.2: A snapshot of the colliding atomization of two micro liquid jets with the middle plane velocity contours and iso-contour of α=0.5. Such snapshots are then used for numerical characterization of the spray.

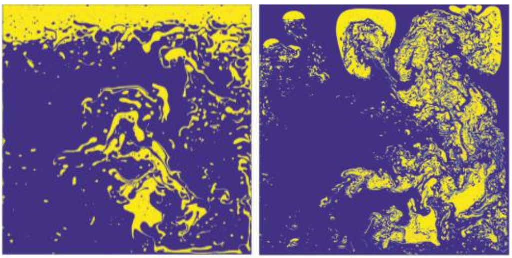


Fig.3: Two snapshots of highly resolved simulation of the liquid-liquid phase inversion problem with Re = 2214 (left) and Re= 20,160 (right) on the 2048 x 2048 grid. These results are used to compute the new SGS models for the LES of interface-turbulence interactions.

MICRO | REDUCED-ORDER MODELLING OF RED BLOOD CELLS

The human body is a treasure trove filled with mysteries that have fascinated the scientific world for centuries. From finding cures and vaccinations for deadly diseases to understanding the function of various organs and cells in our body, there are several areas of prime research interest. One such field is hematology or the study of blood and its complex rheological and physiological properties.

With the advent of computers, it has become feasible to understand blood rheological in a faster and safer manner. Computational methods have been developed in the past decades dedicated to providing solutions to the complex behaviour of blood on a macroscopic, as well as microscopic scale.

Microscale simulations have gained prominence due to their ability to provide details of activities on a cellular level. However, it is still a challenge to balance between the related computational costs and accuracy for such simulations. In our recent publication [1], we have proposed a new approach to model red blood cell suspensions in plasma based on resolved CFD-DEM. Here, a red blood cell is represented as a cluster of overlapping spherical particles interconnected by flexible bonds. The static and dynamic behaviour of the model has been tested showing good accuracy with available literature.

Further tests such as the Wheeler test, shear deformation of an RBC in Couette flow, were performed to study the performance of the model in a more dynamic scenario showing good quantitative agreement with previous works (Figure 1). Moreover, the behaviour of an RBC in a stenosized microchannel as in Figure 2 was performed to study the shape recovery of RBCs. The model was able to replicate the shape memory behaviour of RBCs which is a qualitative measure of RBC viscoelasticity. Additionally, the application of micro-focusing of RBCs using constriction was simulated for real-life microfluidic application such as separation of biologicals cells from plasma. (Figure 3). The RBCs were concentrated in the center of the flow chamber leading to the formation of an extended cell-free layer downstream. The presented ROM-RBC can be utilized similarly to study more complex scenarios in the future.

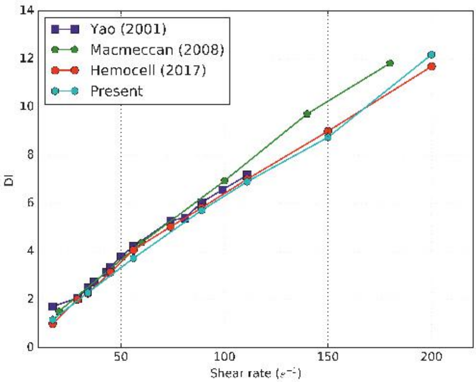


Fig.1: Deformation Index for the Wheeler experiment.

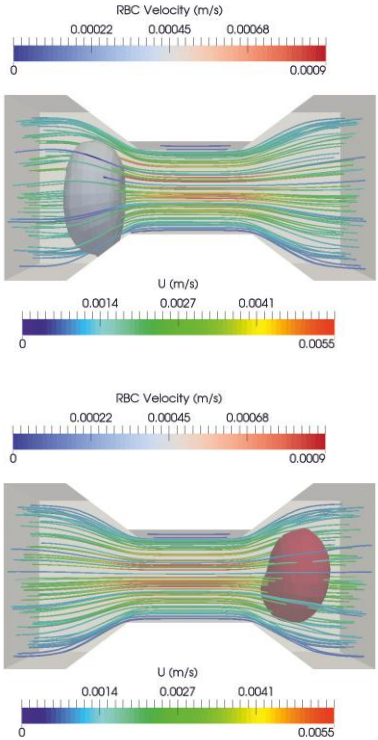


Fig.2: The passage of an RBC through a stenosized microchannel based on the ROM-RBC model.

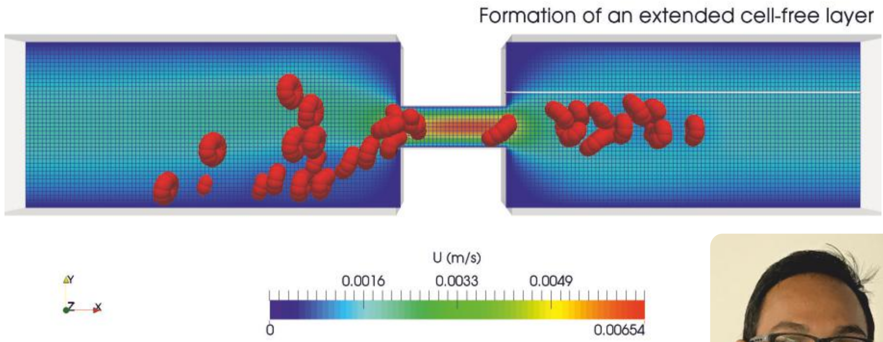


Fig.3: The formation of an extended cell-free depletion layer downstream of the constriction.

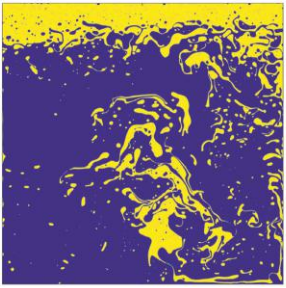
Achuth Nair Balachandran Nair | achuth.balachandran_nair@jku.at

MICRO | A NEW APPROACH TO INLCLUDE SURFACE TENSION IN THE SUBGRID EDDY VISCOSITY FOR THE TWO-PHASE LES

Turbulent two-phase flows feature different mechanisms for the production and dissipation of turbulent kinetic energy compared to the single-phase flows. However, this difference is usually neglected in developing eddy viscosity-based subgrid scale (SGS) models for the two-phase large eddy simulation (LES). In our recent article [1], we improved this shortcoming for the LES of interfacial flows within the context of one-fluid formulation. A new approach is presented for the two-phase LES to include the surface tension, which is a production mechanism for the kinetic energy in the small scale motions, into the subgrid eddy viscosity model.

We follow the Favre-filtered governing equations of interfacial flows based on the volume of fluid (VOF) approach and derive the transport equation for the turbulent kinetic energy to include the effect of surface tension. Then, we propose a new form for the eddy viscosity based on the mixing length assumption which includes an additional production mechanism of turbulent kinetic energy stemming from the interfacial work done by the surface tension. The proposed model for eddy viscosity is employed to close all the SGS terms appearing in the filtered continuity and momentum equations.

The model performance is evaluated utilizing the a-priori filtering of the fine grid simulation of phase inversion problem. To test the generality of the model at different physical conditions, two different density ratios were considered for the fine grid simulation which leads to two different Re numbers. The results highlight a significant improvement of the eddy viscosity-based SGS models in the prediction of the turbulent kinetic energy for the small unresolved scales particularly for the regions of low shear (Figure 1). This study provides a proper perspective for future SGS models in the context of large eddy simulation of two-phase flows and will be used for modelling liquid-liquid and liquid-gas interfacial flows involving interface deformation, rupture, and coalescence. We further continue this line of research with an a-posteriori implementation of the SGS models.



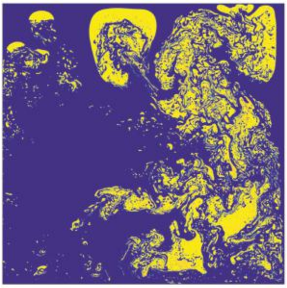
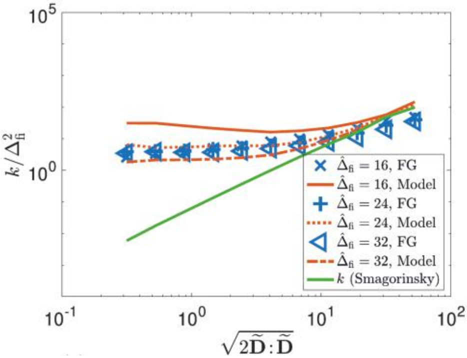
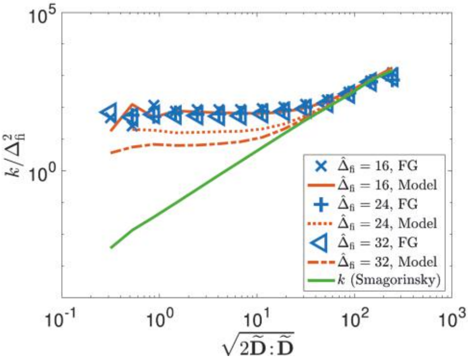


Fig.1: Turbulent kinetic energy as a function of the resolved strain rate for different filter sizes for the liquid-liquid phase inversion problem [1]. The symbols in each plot indicate the filtered fine grid simulation while the red solid and dashed lines correspond to the corrected eddy-viscosity model with different filter sizes. The green line represents the classical Smagorinsky model.





Mahdi Saeedipour | Simon Schneiderbauer

[1] Saeedipour, M., Schneiderbauer, S., A new approach to include surface tension in the subgrid eddy viscosity for the two-phase LES *International Journal of Multiphase flow*, 121, 103128. 2019.

mahdi.saeedipour@jku.at | simon.schneiderbauer@jku.at

MICRO | ATOMIZATION OF TWO COLLIDING MICRO LIQUID JETS IN A RESPIRATORY INHALER

Turbulent two-phase flows feature different mechanisms for the production and dissipation of turbulent kinetic energy compared to the single-phase flows. However, this difference is usually neglected in developing eddy viscosity-based subgrid scale (SGS) models for the two-phase large eddy simulation (LES). In our recent article [1], we improved this shortcoming for the LES of interfacial flows within the context of one-fluid formulation. A new approach is presented for the two-phase LES to include the surface tension, which is a production mechanism for the kinetic energy in the small scale motions, into the subgrid eddy viscosity model.

We follow the Favre-filtered governing equations of interfacial flows based on the volume of fluid (VOF) approach and derive the transport equation for the turbulent kinetic energy to include the effect of surface tension. Then, we propose a new form for the eddy viscosity based on the mixing length assumption which includes an additional production mechanism of turbulent kinetic energy stemming from the interfacial work done by the surface tension. The proposed model for eddy viscosity is employed to close all the SGS terms appearing in the filtered continuity and momentum equations.

The model performance is evaluated utilizing the a-priori filtering of the fine grid simulation of phase inversion problem. To test the generality of the model at different physical conditions, two different density ratios were considered for the fine grid simulation which leads to two different Re numbers. The results highlight a significant improvement of the eddy viscosity-based SGS models in the prediction of the turbulent kinetic energy for the small unresolved scales particularly for the regions of low shear (Figure 1). This study provides a proper perspective for future SGS models in the context of large eddy simulation of two-phase flows and will be used for modelling liquid-liquid and liquid-gas interfacial flows involving interface deformation, rupture, and coalescence. We further continue this line of research with an a-posteriori implementation of the SGS models.

Particulate Flow Modelling

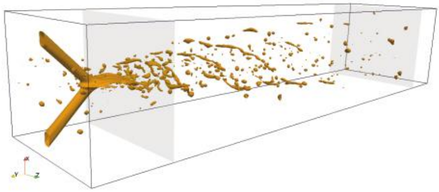


Fig.1: A 3D snapshot of the spray formation out of colliding atomization. The statistics of atomizing flow is obtained at the domain between the gray zones.

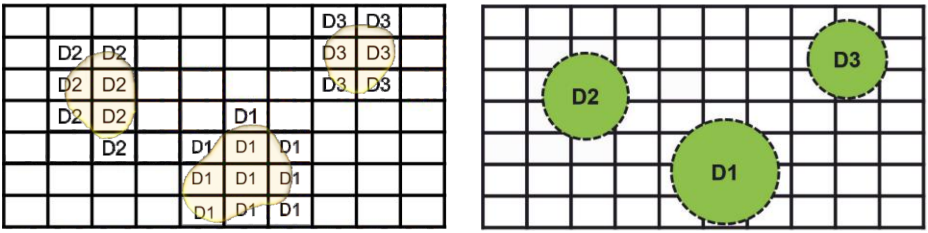


Fig.2: Schematic description of post-processing tool: the connected cells identification for each individual liquid structure (left), and the equivalent-volume droplet detected for each connected structure (right).

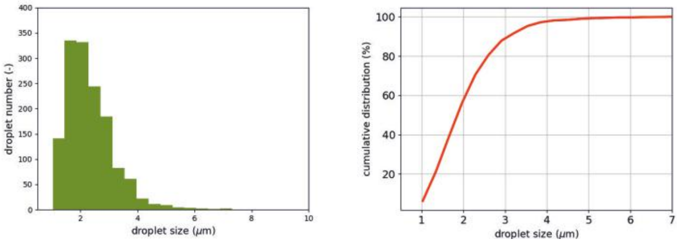


Fig.3: Size histogram and cumulative size distribution of the droplets after the quasi-steady state condition achieved.

[1] Saeedipour, M., Atomization of two colliding micro liquid jets in a respiratory inhaler: A computational study. *29th Conference on Liquid Atomization and Spray Systems*, 2019.

Mahdi Saeedipour | mahdi.saeedipour@jku.at

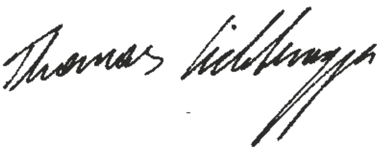
EDITORIAL | MESO

Dear Readers,

When we talk about mesoscopic (from the Greek word *mésos* = middle) research, we mean most generally a description of nature between the extremes of micro- and macroscales. In our field of multiphase flows, the importance of the spatial multiscale hierarchy and the specific role of mesoscopic physics are well recognized. While we have a very good understanding of the laws of physics in the microscopic regime, we often want to simulate large-scale processes. To this end, we need to account for small-scale behavior in terms of an approximate, mesoscopic description. This can either happen in terms of modeling (e.g. **Alija Vila**'s work on sub-scale channels in packed beds, cf. Fig. 1) or algorithmic (e.g. **Daniel Queteschiner**'s contribution on locally resolved, coarse-grained simulations, cf. Fig. 2) developments.

The temporal aspect of mesoscale science to build bridges between short-term, microscopic and long-term, macroscopic dynamics is less well understood. While the description of spatial sub-scale structures usually leads to correction terms to the underlying, microscopic equations of motion, such an approach is less fruitful for the temporal evolution. To the best of my knowledge, there is absolutely no consensus how the laws of physics for mesoscopic time scales could look like. However, maybe we do not even have to find it out! Over the last few years, data-driven approaches have spread in various scientific disciplines and might fundamentally change our field, too. Since we can generate high-fidelity data for various types of flow, we might be able to train a computer to understand their mesoscopic evolution. The value of such an approach will be substantial, but it will be a long way to go even though we have already taken first steps with recurrence CFD. Dear Readers, I hope we can walk some part of it together!

Sincerly,



Thomas Lichtenegger | thomas.lichtenegger@jku.at

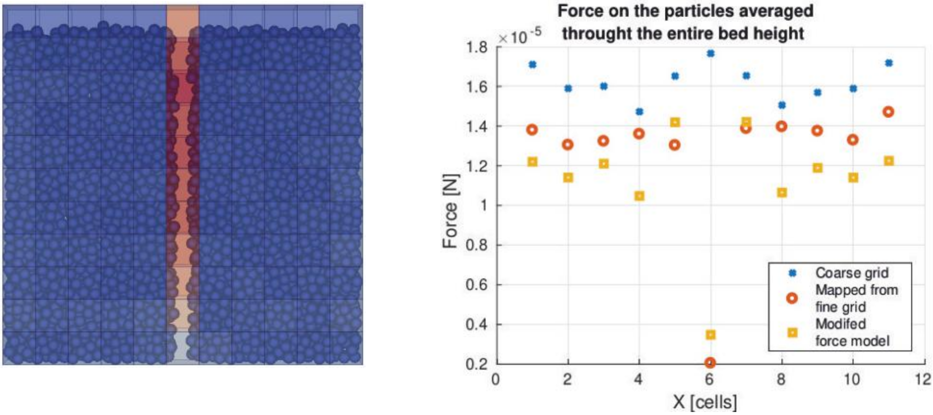
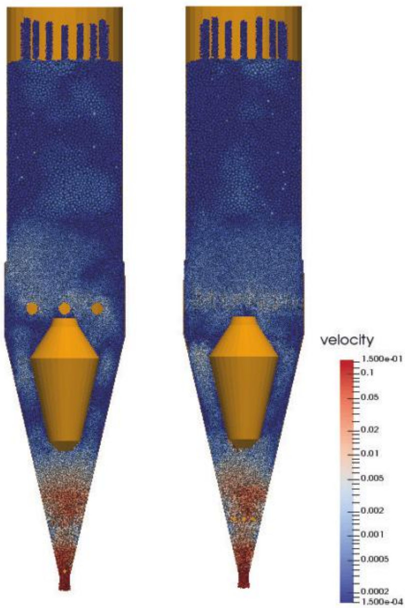


Fig.1: Gas flow through a packed bed with a channel. If a coarse grid is used, the heterogeneity is not resolved and the forces acting on the particles are overestimated. However, the influence of the channel can be accounted for in an implicit fashion in the calculation of the forcing term.

Fig.2: Front and side view of a Midrex process simulation. While the upper part of the shaft furnace allows for massive coarse-graining, burden feeders in the mid and lower regions and the relatively small outlet require a finer resolution. Embedded DEM simulations enable combinations of fine- and coarse-grained domains.



MESO | EXTENDING THE MULTI-LEVEL COARSE-GRAIN MODEL OF THE DEM TO UNRESOLVED CFD-DEM SIMULATIONS

In a wide field of applications, the discrete element model (DEM) in combination with computational fluid dynamics (CFD) has proven to be a useful tool to study granular flows. However, a major shortcoming of simulation methods involving the DEM is the high computational cost. The increase of simulation runtime with increasing number of particles hinders the usage of DEM and CFD-DEM simulations for large-scale systems.

The coarse-grain (CG) model of the DEM lowers the computational demand by using coarser (pseudo) particles to represent a number of original particles. However, due to the violation of geometric similarity, this approach fails to capture effects that intrinsically depend on particle size.

To overcome this issue, we have introduced a method which concurrently couples multiple coarse-grain levels to adjust the resolution of the simulation as required. Spatially confined sub-domains of finer scale are embedded into coarser representations of the system and are coupled by exchanging volumetric properties of the granular flow.

We have extended this multi-level coarse-grain (MLCG) model to unresolved CFD-DEM simulations where the DEM part is typically taking up a large fraction of the computational resources. On the CFD side, each coarse-grain level of the granular model is treated separately using appropriately scaled drag laws and a suitable mesh resolution. The DEM component links up the different CG levels.

Figure 1 shows the DEM part of the fluid flow through a packed particle bed, where the fluid is entering from the bottom. One can observe the particle velocities resulting from the drag forces. In the MLCG case, the forces were calculated separately for each CG level. Figure 2 compares the void fraction fields obtained by coupling the same CFD setup to a reference-size DEM simulation, a MLCG DEM simulation and a conventional CG DEM simulation. In the MLCG setup, the Eulerian fields based on particle properties are basically a composition of fine- and coarse-grained DEM data.

Technically, for the most part, the different CG levels are hidden from the fluid solver and its sub-models such that little to no adjustment is required compared to the standard CFD-DEM implementation. The communication between the CFD and DEM parts is realized via MPI, where the CFD side sre 3.while the coarse-grain levels on the DEM side run in disjoint child communicators ("simulation worlds") as illustrated in Figure 3.

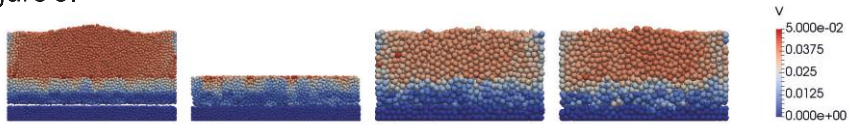


Fig.1: Fluid flow through a packed particle bed. The solid phase of the reference simulation is shown on the left, a conventional coarse-grained simulation on the right, and the fine- and coarse-scale parts of a coupled MLCG simulation in the middle.

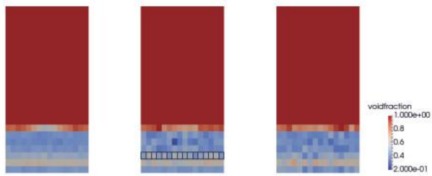


Fig.2: Volume fraction of the fluid phase for the simulation shown in Fig. 1. All three simulations use the same setup for the fluid phase. Coupling to the reference particle simulation (left) results in the smoothest Eulerian field. Coupling to the CG particle simulation results in a coarser void fraction field (right). In the MLCG case, the fine-scale particles are used to calculate the void fraction field below the MLCG transition zone (black cells), whereas the computation for the cells above is based on the coarse-grained particles.

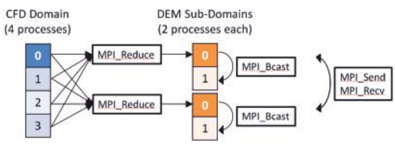


Fig.3: Communication path from a CFD simulation running on 4 processes to a DEM simulation with 2 CG levels, each of which uses 2 processes.

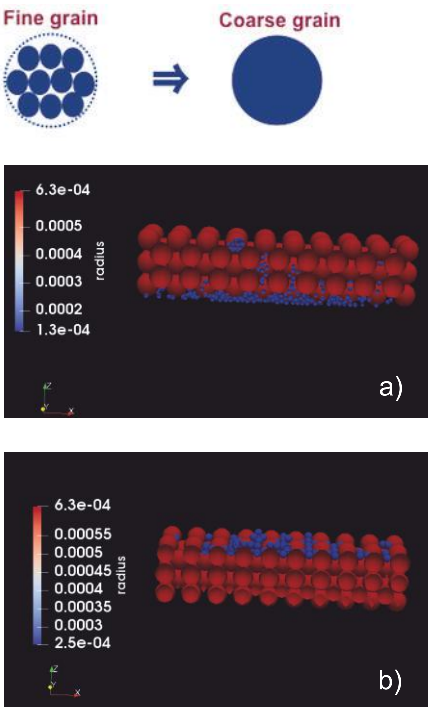
Daniel Queteschiner | daniel.queteschiner@jku.at

MESO | REVISED COARSE GRAIN APPROACH FOR THE SIMULATION OF HOPPER FLOW WITH POLY-DISPERSE PARTICLES

Discrete element method (DEM) is a widely used numerical method to model dynamics of granular flow in various industrial applications. But at the same time, it has certain limitations in terms of computational cost with respect to time and memory consumption. Therefore, we use an alternative method to simulate poly-dispersed hopper flow. We implement a hybridized coarse grain method by decomposing the particle mixture into two sub categories with respect to their size as well as the numerical methods used to model their corresponding dynamics. All the particles are of spherical size at the moment but they have different size ratio. Dynamics of larger particles are calculated using DEM and dynamics of the smaller particles are calculated implementing the coarse grain approach where each coarse grain consists a cluster of small particles. This approach reduces number of particles in a significant amount which in return, optimizes the computational cost. But, this method changes geometrical size distribution of particles and hence influence the original particle dynamics (Fig1). We apply a revised coarse-graining approach using a corrected diameter for the collisions of parcel containing small particles and the large DEM particles (Fig 2) to get the correct segregation of particles. Thereafter, a quantitative volume fraction (VoF) comparison with the reference solution is introduced (Fig. 3). For a particle size ratio up to 1:4, using this approach we are able to reduce the computational time by 94% of the time required for reference fine grain DEM simulation.

Our goal is to get the detail insight of the effect of size ratio and the corresponding coarse grain ratio on the overall dynamics of particle mixture in order to get a better understanding of customized optimization of computational cost associated with its corresponding simulation.

Particulate Flow Modelling



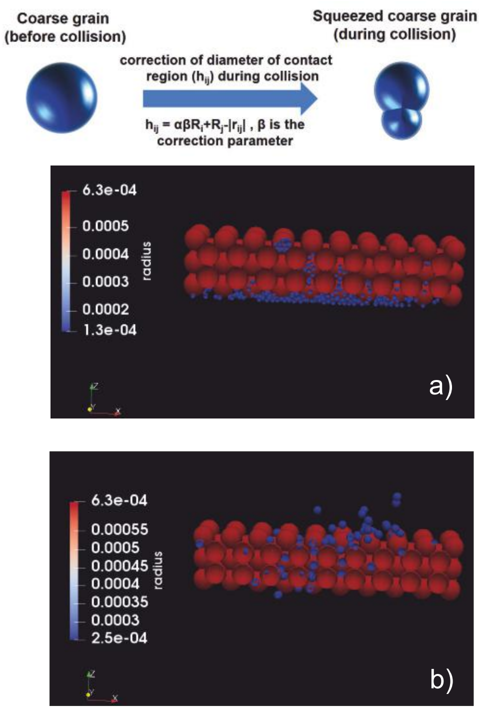


Fig.1: (a) Fine grain small particles can percolate and (b) Coarse grained particle parcel can't percolate due to inadequate pore space.

Fig.2: (a) Fine grain small particles can percolate and (b) Revised coarse grained particle parcel can percolate through the stabilized and densely packed bed of large particles.

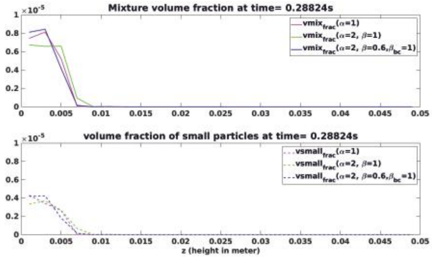


Fig.3: Volume fraction comparison of particle mixture and of small particles between coarse grain and reference fine grain simulation.

Suranita Kanjilal | suranita.kanjilal@jku.at

MESO | DIRECT REDUCTION OF IRON ORES IN FLUIDIZED BEDS

The reduction of iron ores has gained much interest in the past decades as it is a core process in the iron- and steel-making industries. According to WorldSteel Association the global steel industry uses approximately 2 billion tonnes of iron ore, that is reduced to usable iron through technologies as blast furnaces and fluidized bed reactors. In order to improve our understanding on the reactions taking place and the parameters affecting them, numerical methods have to be utilized, due to the harsh conditions inside these reactors that make physical measurements extremely difficult.

We implemented a mathematical model that represents the indirect reduction of iron ore into the CFD-DEM coupling library and carried out validations against available experimental data. For initial validation, a single hematite ore was reduced with various gas compositions, ranging from only CO and N_2 to a gas mixture of CO, CO_2, H_2, H_2O and N_2. We determined that the numerical results achieve the same reduction degree at the same time with experimental data and found out that the model overestimates the reduction of hematite and magnetite due to the sharp interface assumption. Nonetheless, the model is able to correctly predict the rate determining reaction, wüstite to iron, thus achieving the same reduction degree with experiments.

Afterwards, we investigated the influence of the kinetic parameters such as per-layer activation energies, E_a, and pre-exponential factors, k_0. Increasing the activation energies for hematite, doesn't influence the reduction degree as the reduction occurs rapidly. The same is not valid for magnetite, in which an increase of E_a from 70 kJ/mol to 100 kJ/mol slows the reduction 27%. As the rate determining reaction, increasing wüstite E_a from 68 kJ/mol to 100 kJ/mol decreases the reduction rate around 44%, as illustrated in Fig. 1. In comparison, increasing the pre-exponential factor results in a faster reduction.

Lastly, numerical simulations for a lab-scale polydisperse fluidized bed in which convective and conductive heat transfer along with heat arising due to reactions are taken into account, are carried out.

The reaction conditions are defined so that only one reaction step can proceed. Fig. 2 illustrates comparison of reduction of magnetite and wüstite for various particle sizes against experimental data in a fluidized bed.

Future research will work on applying the model to industrial scale fluidized beds, where a parcel approach that assembles sub-sets of particles into parcels might be followed. As coarse graining and reduction rate scaling are already implemented, other factors to decrease computational time might be considered.

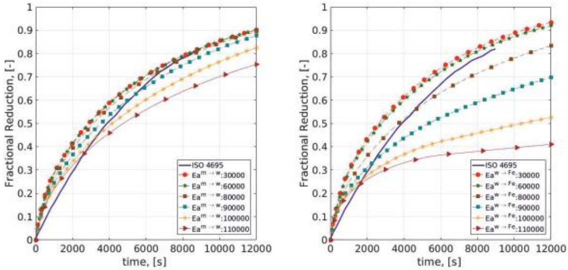


Fig.1 (above): Fractional reduction for different activation energies where blue line represent the experimental results and the marked lines various activation energies that can be found in literature (left) for magnetite layer and (right) for the rate determining wüstite layer.

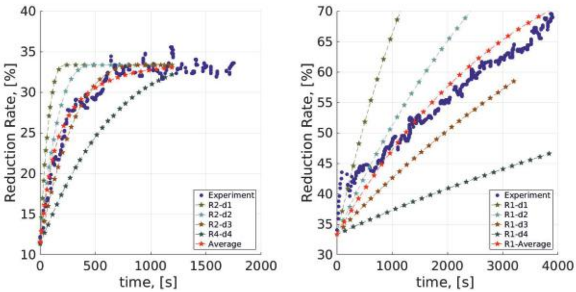


Fig.2 (above): Reduction rate for iron ore particles with different diameters for the reduction from magnetite to wüstite (left) and wüstite to iron (right). The scattered dots representing th experimental data and the marked lines the CFD-DEM results.

M. Efe Kinaci | mustafa_efe.kinaci@jku.at

MESO | RECURRENCE CFD APPLICATION IN TURBULENT FLOWS

With the advances in computer power and computational fluid dynamics (CFD) techniques, it is feasible to solve more sophisticated problems in a wide range of spatial and temporal scales; however, we are limited to short-time simulations because of highly expensive calculations in complex processes. “Recurrence CFD” (rCFD) employs a system’s reappearing patterns for time-extrapolating its behavior and reduces the computational cost considerably.

In some cases, there is no apparent dominant periodicity and we need an approach to extract the coherent structures and examine their behavior. Here, we created a computational domain with 650000 cells and studied a confined turbulent jet (Fig. 1 (a)). We applied the method of proper orthogonal decomposition (POD) to capture the most energetic modes with the purpose of investigating their effect in rCFD analysis. Hence, we gathered a database with 1000 snapshots for a time range of 50 seconds to identify the most similar states for two cases. As it can be observed in Fig. 2 (left), there is no visible dominant spatial feature in the distance matrix. If we reconstruct the flow fields according to the first 5 modes comprising 80% energy of the system (Fig. 1 (b)), we can observe the existence of some dominant spatial features in Fig. 2 (right) by separating the coherent structures form the incoherent ones.

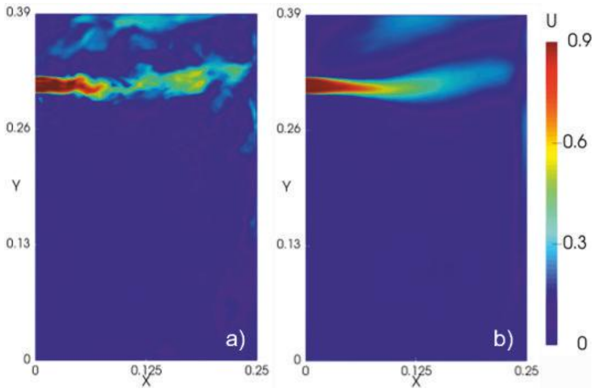


Fig.1: Contour plot of a) instantaneous velocity field and b) reconstructed field with first 5 modes at *t = 25 sec.* The velocity fields are plotted for the mid-plane of the jet.

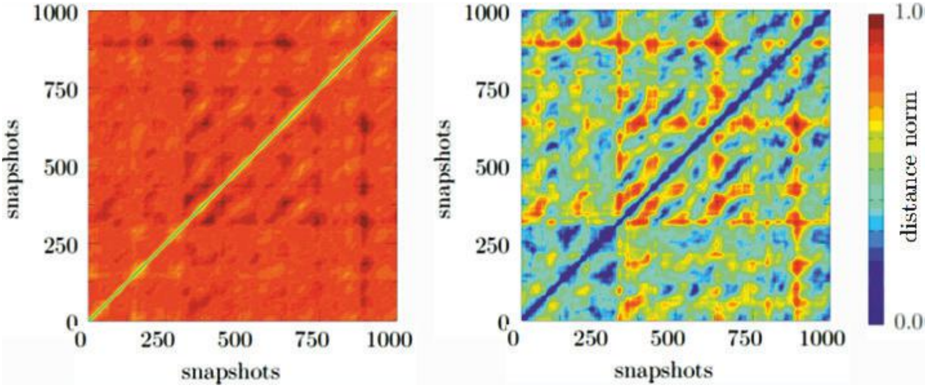


Fig.2: Distance matrices calculated according to velocity fields resulted from LES (left) and reconstructed fields with the first 5 modes (right). The recurrent patterns are visible for the most energetic modes.

As the next step, we carried out rCFD to examine a passive scalar transport. First we used the original, full velocity fields. Then, we created a database composed of reconstructed fields from the first 5 modes and the higher modes were modeled as turbulent diffusivity. Figure 3 depicts that we cannot compensate the effect of velocity fluctuations by a constant coefficient and we would need much larger number of modes. This underlines the importance of using full field information and not just a few leading modes.

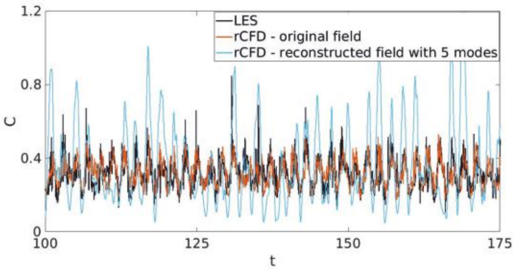


Fig.3: Tracing species over time at one selected point. rCFD with the original database is in very good agreement with LES. However, we need higher number of modes to reconstruct our velocity fields.

Sanaz Abbasi | sanaz.abbasi@jku.at

MESO | LONG-TERM INVESTIGATION OF HEAT TRANSFER IN MOVING BEDS USING TIME-EXTRAPOLATION

The most accurate models for granular systems compute the trajectories of each particle or parcel according to the forces acting on it from the surrounding grains and fluid. To obtain a numerically stable representation of the contacts between stiff materials, small time steps need to be chosen, which are often on the order of microseconds and below. Macroscopic flow behavior on the other hand is characterized by completely different time scales. One often encounters either recurrent (e.g. fluidized beds, turbulent flows; return times of seconds) or approximately steady-state (e.g. moving beds) motion. Especially if slow, long-lasting processes like heat transfer or certain types of chemical conversion take place, application of standard simulation tools becomes unfeasible.

For such scenarios, it is indispensable to decouple different time scales. One needs to obtain cheap approximations for the dynamics of the granular multiphase system, while long-term processes may be studied with conventional approaches. Over the last few years, we have developed a completely novel, data-assisted strategy that may be used for both recurrent (see contributions on recurrence CFD) and steady-state problems.

Recently, we have demonstrated how one can describe heat transfer in a **real-scale blast furnace** over the duration of 24 hours as sketched in **Fig. 1**. First, we obtain the time-averaged granular velocity and volume fraction fields (cf. **Fig. 2**) from massively coarse-grained CFD-DEM simulations over short durations at cold conditions. Next, we replace solid grains with non-interacting tracers that follow the stream lines with much larger time steps and correspondingly lower computational costs. Similarly, we increase fluid time steps by postulating that gas flow becomes steady within each granular step. Since we allow tracers to carry temperatures and to interact with the gas phase, we can study heat transfer over long durations.
While such an investigation would be limited to a few seconds with CFD-DEM, our approach allows us to **carry out simulations for 24 hours of process time with little costs of 120 hours CPU time on 8 cores**. Starting from a cold blast furnace, we can monitor its temperature evolution towards its final state shown in **Fig. 3**.

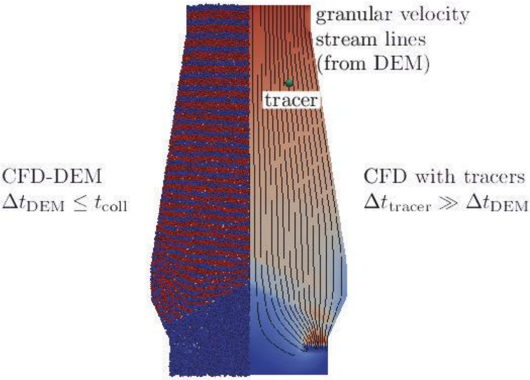


Fig.1: Simulation strategy for steadily moving particle beds. Once the time-averaged granular velocity has been obtained, solid grains may be replaced with non-interacting tracers (green) which simply follow stream lines.

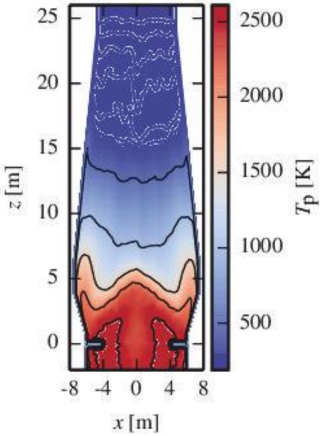


Fig.3: Final temperature-distribution in a full-scale blast furnace obtained after 24 hours. A significant temperature gradient is present across the cohesive zone.

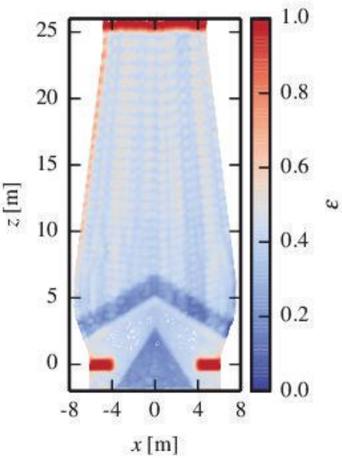


Fig.2: Particle volume fraction field. The predefined cohesive zone and deadman manifest with higher packing fractions while the raceways are almost void.

Thomas Lichtenegger | thomas.lichtenegger@jku.at

MESO | MODELING OF GAS-SOLID FLOW WITH COHESIVE POWDERS

Using a rheometer, we have measured the powder's resistance to flow when it is subjected to aeration. This procedure is additionally modelled with a coupled CFD-DEM simulation (see Figure 1). As shown in Figure 2, the energy required for a downwards movement of a rotating blade is reduced if the particle column is aerated. Airflow separates neighbouring particles and, in case of cohesive powders, a channel through the sample might be established.

Furthermore, in our die filling experiment where a feeding shoe is moved over the cavity, we have discovered that an initial bed loosening by aeration increases the powder flowability and filling performance (see Figure 3). Airflow introduces narrow gas channels throughout the powder bed, dividing it into respective sections that then can flow without forming arches. This phenomenon was also reproduced with unresolved CFD-DEM simulations with cohesive particles.

Fractures of particle assemblies can happen frequently for cohesive and non-spherical particles. Heterogeneity created in the configuration influences particle-particle interactions and can have a strong impact on the behaviour and the properties of a powder bulk (i.e. powder flowability).

In unresolved CFD-DEM simulations, the channel width might be smaller than the FV-mesh. This is referred to as a sub-grid heterogeneity. In our approach, we rely on DEM data to provide us with a finer resolution to identify such formations. Cells that contain particles with low coordination numbers are identified as they might contain a heterogeneity. Then, we calculate the approximated channel width and modify the force model accordingly.

Previously, we have performed resolved lattice-Boltzmann simulations of fluid flow through arrested particle configurations with a channel. The corresponding gas-particle momentum exchange and pressure drop were calculated. Simulations indicated a significant decrease in the overall pressure drop even for channel widths of only one particle diameter. This resolved lattice-Boltzmann simulation data can be used to deduce the drag force corrections for the unresolved CFD-DEM approach.

Particulate Flow Modelling

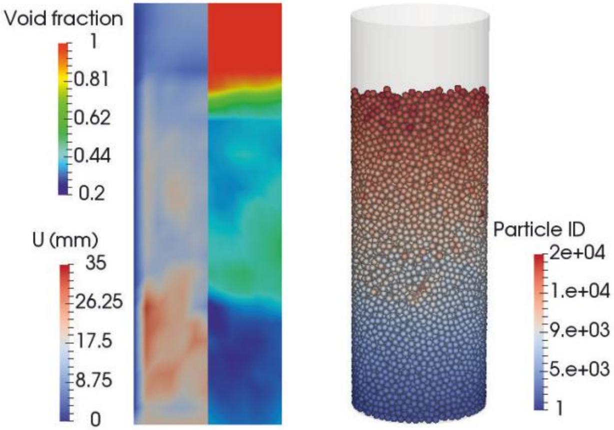


Fig.1: Coupled CFD-DEM simulation of the rheometer experiment. The air is introduced from the bottom of the vessel.

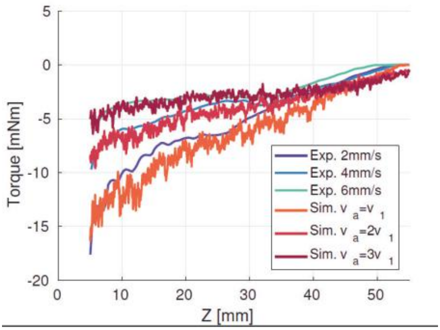
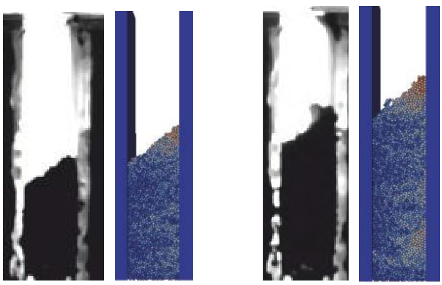


Fig.2: In rheometer aeration experiment, blade torque reduces with the air velocity. The trend is seen in our coupled CFD-DEM simulation.



a) No pre-aeration **b)** Pre-conditioning by aeration

Fig.3: More powder flows into the cavity if it is conditioned by aeration (experiment and simulation).

Alija Vila | alija.vila@jku.at

EDITORIAL | MACRO

Dear Readers,

The last year's focus was laid on the implementation of our recent advancements in the modeling of large-scale multiphase flows into opensource software. We proudly present the first openly available version of **`twoPhaseEulerTurbFoam`**, which includes two different approaches for the large eddy simulation (LES) of gas-particle flows. This solver can be downloaded directly from the departments github repository and is frequently updated.

Currently, we have have two ongoing international collaborations. On the one hand, the bottom outlet flushing process in water reservoirs (Figures 1 and 2) is investigated together with the Stellenbosch university in South Africa. This efforts have been recently published in the International Journal on Sediment Research. On the other hand, together with the Institut de Mecanique des Fluides de Toulouse (Prof. Simonin and Prof. Fede) we currently head to unveil the nature of turbulence in gas-particle flows (Figure 3).

Internally, the micro and macro groups joint efforts to bring further the modelling of turbulent multiphase flows, where we recently developed a novel sub-grid scale model accounting for the small unresolved scales of surface tension.

Finally, I want to thank my team members for their encouragement and their excellent work.

Sincerely,

Simon Schneiderbauer | simon.schneiderbauer@jku.at

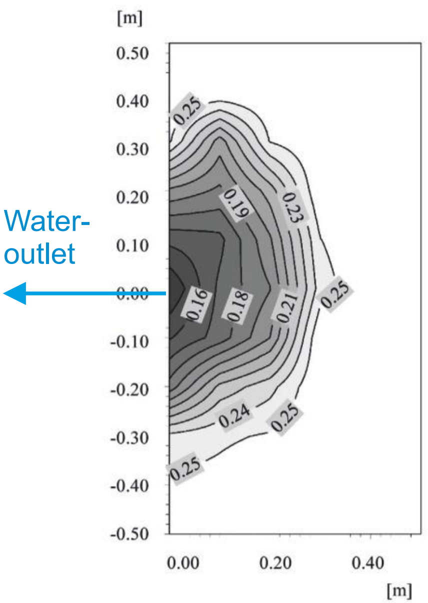


Fig.1: Exemplarily contour map of bed levels (scour hole) near the water outlet [1].

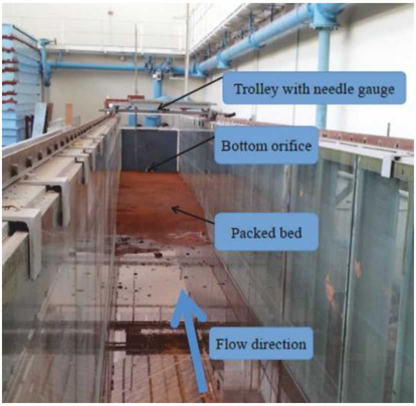


Fig.2: Experimental setup of bottom outlet flushing process in reservoirs [1].

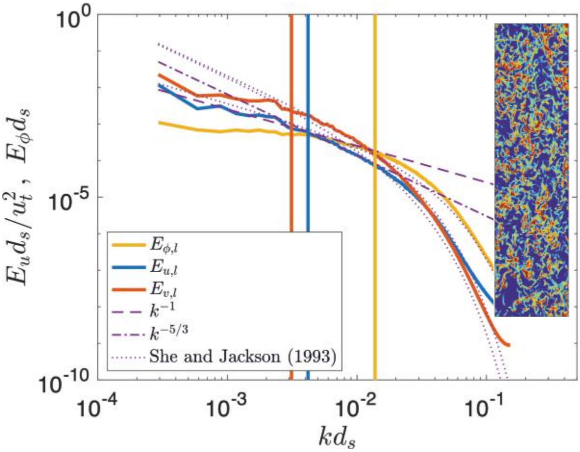


Fig.3: Normalized energy spectra for the solids volume fraction ϕ, the vertical solids velocity and the vertical gas velocity in the case of unbounded fluidization.

References:
Sawadogo, O., G.R. Basson, S. Schneiderbauer. Int. J. Sediment Res., 2019, 34 (5), 461–474.

MACRO | CLUSTER INDUCED TURBULENCE FOR MOMENTUM AND HEAT TRANSFER

In industrial scale gas-solid flows, like fluidized beds or risers, cluster are observed to form due to the momentum coupling between the gas- and particle phase in the presence of a mean body force, such as gravity. Gas-solid interface exchange models for i.e. momentum and heat are usually derived for homogeneously distributed particles in the flow. In applications, coarse numerical grids are used in order to render simulation of industrial sized reactors feasible. These coarse grids do not resolve the particle clusters, which can only be a few particle diameters wide. By using uncorrected drag models, the drag force acting on the particle cluster is severely overestimated, since the information about the heterogeneity of the flow is lost. This overestimation was shown to be up to 80% of the resolved drag force and leads, for example, to a severe overestimation of the bed expansion.

In our first project phase we were able to derive closure models for the unresolved terms in the momentum and internal energy balance equations in heterogeneous flows. Thereby, the important quantities were identified and modelled in the framework of multiphase turbulence modelling. The most important quantities were observed to be the drift velocity and the drift temperature, measures for the sub-grid heterogeneity of the flow. These quantities can be expressed as a function of the variance of the solid's volume fraction and either the turbulent kinetic or turbulent internal energy. Transport equations for these turbulent energies were derived and the models were evaluated in an a-priori study.

The next step was to implement the newly developed turbulence models in order to start an a-posteriori evaluation and validation against i.e. experimental data. A dynamic adjustment of the correlation coefficients present in the model equations was implemented using test-filters on the resolved quantities. In a preliminary comparison of a coarse-grid simulation with a grid-size, which is 8 times larger than that of the fine-grid simulation of the same Geldart A-type, homogeneously bubbling fluidized bed, we showed that the macroscopic flow properties, like the bed expansion (see Fig. 2) and volume fraction distribution over the height of the bed, as well as the variance of the volume fraction (see Fig. 1) could be correctly reproduced by our models.

Particulate Flow Modelling

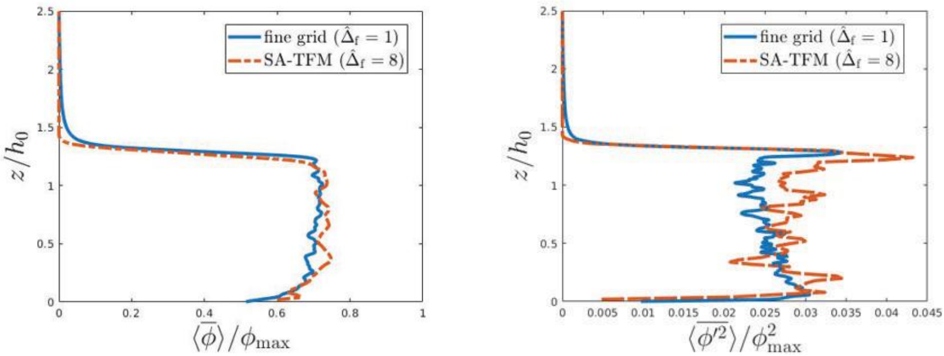


Fig.1: Comparison of the results of a fine-grid simulation, with a grid-size which resolves the heterogeneous particle clusters, to the predictions of the developed Spatially-averaged Two-fluid Model calculated on an 8-times coarser grid. Average volume fraction distribution (left) and variance of the volume fraction over the height of the bed (right). The averaging time was 2 s.

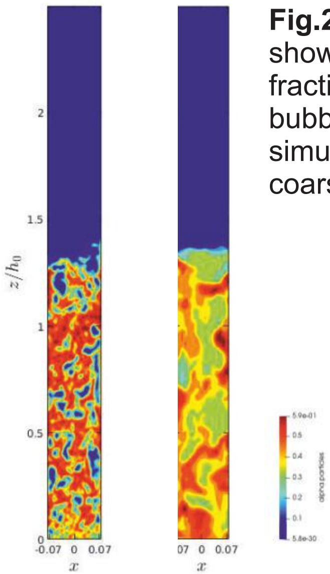


Fig.2: Snapshot of the flow field at time = 5 s, showing the spatial distribution of the solid's volume fraction in a slice view of the homogeneously bubbling Geldart A type fluidized bed. Left: fine-grid simulation, resolving particle clusters; right: 8-times coarser grid with SA-TFM drag correction.

Stefanie Rauchenzauner | stefanie.rauchenzauner@jku.at

MACRO | DEFLAGRATION

We at PFM are a coherent group of completely different scientists – each one with his/her individual focus and ideas of doing research. Our research on modelling deflagration events in a 20l bomb device is just another example of how much we can benefit from the interplay of these different perspectives.

From an experimental perspective, we want to see the real-world particle distribution inside the bomb prior to ignition. This distribution determines the subsequent process of local particle combustion which, in turn, controls the global propagation of the deflagration front. Here, we see the physics, which should be the base for any further modelling.

From a theoretical modelling perspective, we can deduce essential model requirements from these real-world observations. If the particle cloud forms streak-like coherent structures (see page 43), we have to account for these structures in our coarse-grained models (since we are not able to resolve this cluster formation with reasonable computational efforts). Hence, we have to come up with adequate sub-grid models addressing such non-resolved heterogeneities.

Finally, from a practical perspective, we should try to tackle these phenomena by an efficient and robust numerical simulation environment. Here, we learnt a lot from our developments in the realm of recurrence CFD. We can combine (coarse-grained) Lagrangian predictions with an (imperfect) Eulerian interpolation, only to correct the results by a physics-based balancing correction.

However, at some point we have to accept that we at PFM are missing an essential perspective. In this case we don't have the capacities to perform real post-ignition experiments. That's the point, where we look for scientific friends and we are more than happy that Harald Raupenstrauch and Christoph Spijker from Montan University Leoben will join forces with us in the upcoming year of this project.

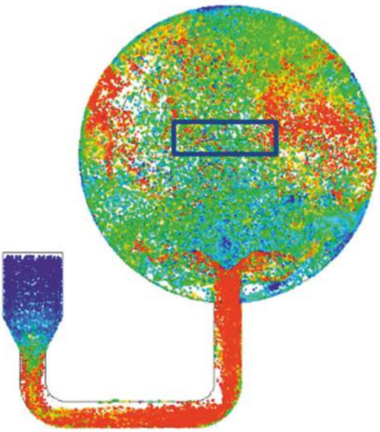


Fig.1: Simulation of bomb filling from a pressurized reservoir, particles coloured by velocity.

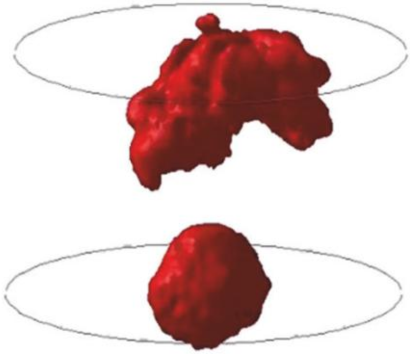


Fig.3: Instantaneous flame front for two different ignition delay times of 20 and 120 ms, both 5 ms after ignition, in the first case the effect of the initial flow is clearly visible.

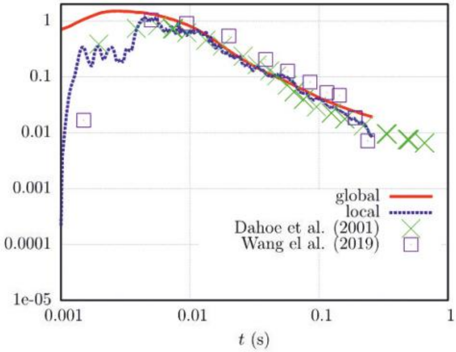
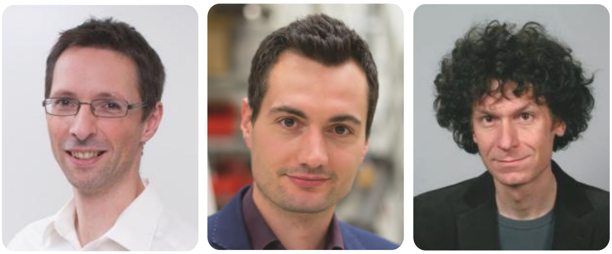


Fig.2: Decay of turbulence after filling of particles into the bomb; comparison with experiments.

Stefan Puttinger | Simon Schneiderbauer | Stefan Pirker



stefan.puttinger@jku.at | simon.schneiderbauer@jku.at | stefan.pirker@jku.at

MACRO | TOWARDS A FAST FLUIDIZED BED SIMULATION USING RECURRENCE CFD

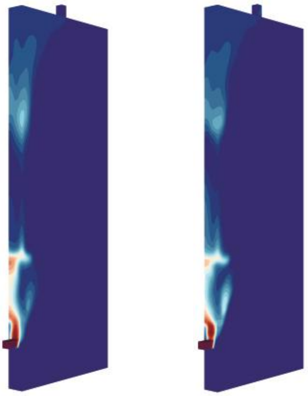
The novel approach of recurrence CFD (**rCFD**) constitutes a liable candidature towards accessing the slow phenomena in extremely large systems. It‘s essence is based on capturing the recurrent dynamics of a pseudo-periodic coherent flow and employ it for the fast transport modeling of such slow passive process.

In this regard, we adopt the so-called transport-based rCFD model † for the fast prediction of chemical species conversion and heat transfer in bubbling fluidized beds (FB) simulated using ANSYS/Fluent. Following this model, the dynamics are tracked on the gas and solid phase and stored, beforehand, as Lagrangian trajectories at each recurrent moment. Our primary focus was about how to retrieve all diffusion constitutents that a passive scalar can undergo during the convection from one recurrent moment to another. To do so, the ensemble oscillating dispersion of a scalar parcel, along this temporal jump, is assumed as a turbulent diffusion and approached basing on the turbulent kinetic energy of the tracked dynamics. We have applied this method for the fast modeling of species propagation on the gas (Fig.1) and solid (Fig.2) phase, and the interacting scalars transport such as temperature in FB (Fig.3). The rCFD obtained results were very promising with a computational speed-up ratio (*i.e.* cost by full CFD/cost by rCFD) up to 1600.

As future concerns, we will converge to model the active reactions between chemical species in FBs. Therewith, more consequences like the trigger of new solutions and heat sources will be considered.

Pirker & Lichtenegger, ChES, 188, 65-83, (2018).

Particulate Flow Modelling



Full CFD a) rCFD

b)

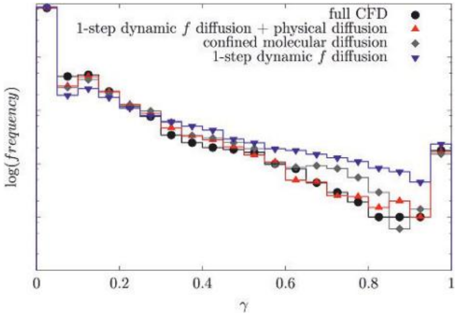


Fig.1: Gaseous species results (a) at $t = 6.2$ [s] with the histogram (b) at the same moment.

d)

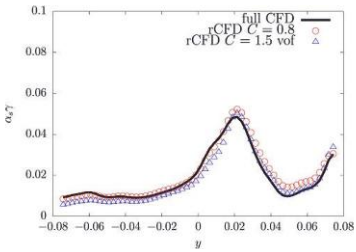
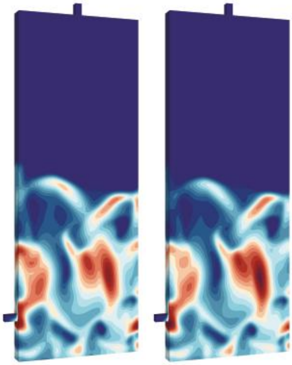


Fig.2: Solid species results (c) at t = 4.6 [s] with the mean profile (d) at height=0.03[m].



Full CFD c) rCFD

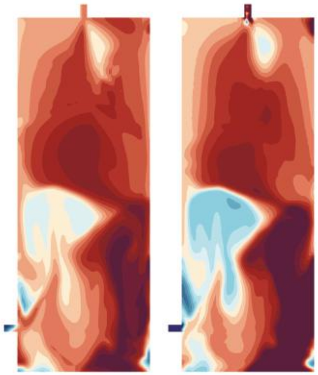
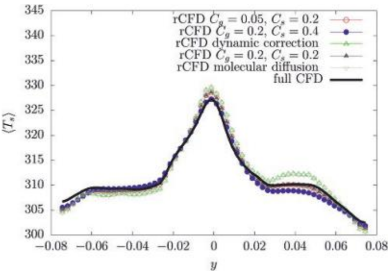


Fig.3: Solid temperature results (e) at t =4.6 [s] with the mean profile (f) at height=0.025[m].



Full CFD e) rCFD

f)

Firas Dabbagh | firas.dabbagh@jku.at

MACRO | APPLY RECURRENCE CFD TO SIMULATE POLLUTANT DISPERSION IN BUILT ENVIRONMENT

Air pollution in the urban environment has become a pressing issue all over the world over the last decades. It is very important to be able to provide more accurate simulations for near-field pollutant dispersion towards the provision of effective solutions for controlling and maintaining outdoor air quality. For the first time, transport-based rCFD is applied to simulate atmospheric pollutant dispersion around a building.

Beyond a pure application of an existing method, the transport-based rCFD is significantly further developed to accurately reproduce the pollutant dispersion. The existing (rudimentary) global one-step diffusion concept [1] is thoroughly revised and further developed, since the turbulent diffusivity plays a crucial role in the atmospheric dispersion process.

In terms of application, the pollutant dispersion around an isolated cubical building with a rooftop vent immersed in a neutrally-stratified atmospheric boundary layer is tested. For the time-averaged pollutant concentration, the results by full LES and rCFD are very close as shown by (i) concentration coefficient contours in a horizontal and a vertical plane (Fig. 1a-d); and (ii) 3D plume shapes based on iso-surfaces of mean concentration coefficient (Fig. 2a-d).

This work demonstrates the applicability and the potential of transport-based rCFD in the field of atmospheric pollutant dispersion in the built environment.

[1]Pirker & Lichtenegger, ChES, 188, 65-83, (2018).

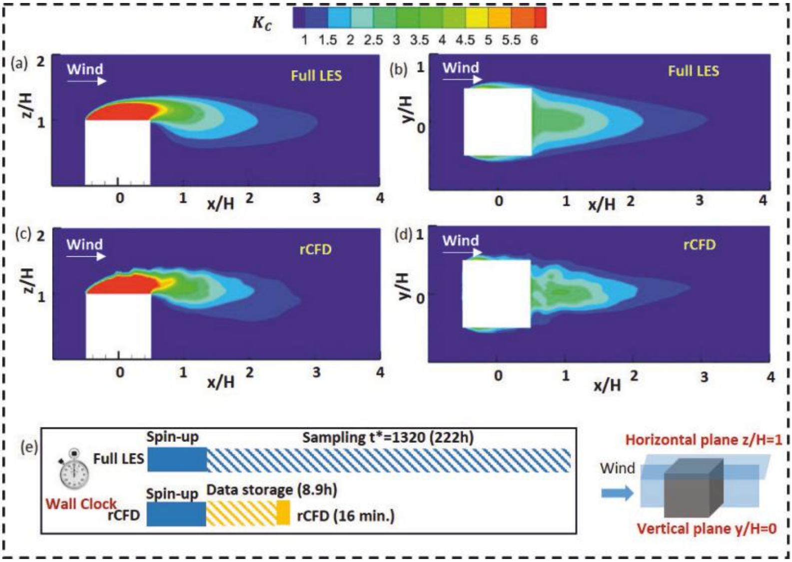


Fig.1: Comparison of mean concentration coefficient between full LES and rCFD in two planes (a)-(d), and (e) wall-clock time consumption.

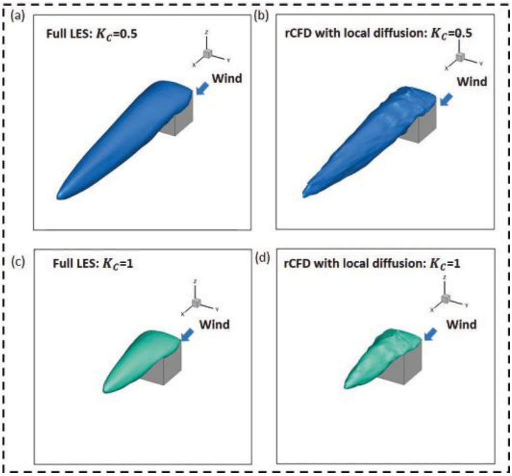


Fig.2: Iso-surfaces of time-averaged concentration coefficient.

Yaxing Du | yaxing.du@jku.at

MACRO | RECURRENCE CFD

Let's try to bring rCFD toward real industrial applications. And what is more 'industrial' than 170 tons of liquid steel circulating from a ladle into a vacuum chamber forth and back. Here high amount of gas are introduced into one upward snorkel which triggers a highly turbulent flow featuring a violently disturbed free surface.

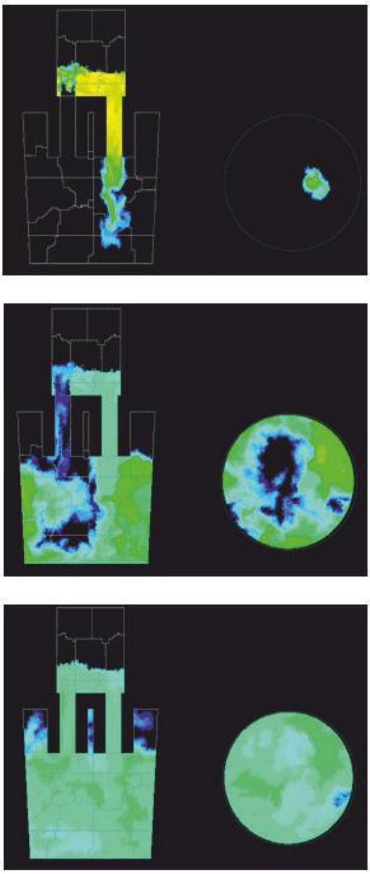
Modelling such a turbulent multiphase flow certainly is a challenge by its own. A feat, we achieved by combining a Lagrangian bubble swarm model with an Eulerian free surface model together with a scale-resolving turbulence model. But is there a change to address such a flow by means of recurrence CFD? Will we in the end be able to picture species mixing and homogenization in real-time?

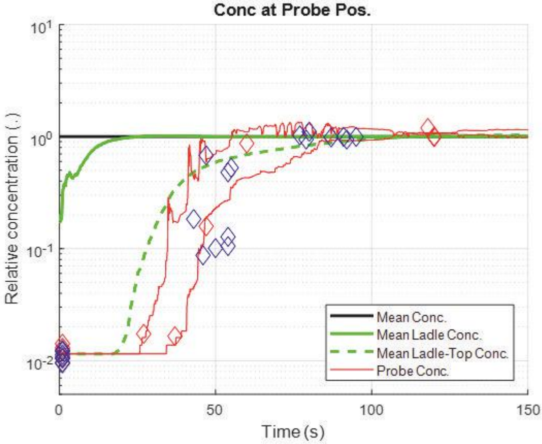
We started this quest by establishing a (multiphase) flow database by means of full CFD simulations based on a computational grid of nearly three million cells. On top of this database, we applied recurrence CFD in order to picture the process of titanium dissolution within the melt. rCFD simulations indicate that the homogenization of titanium concentration happens very fast, lasting only about one minute – a surprising finding, which our industrial partners could substantiate by specific plant trials.

While this information certainly is of interest by its own, we could further provide these results in real-time. So, also our rCFD simulation of homogenization lasted only about one minute (just remember, that's multiphase and turbulent species propagation on a 3M grid).

With this first successful application we start asking ourselves how we could employ rCFD as a digital shadow of an industrial process. And if we are able to implement a digital shadow running in parallel to the plant, what additional information we could deduce for an improved control of the plant. That's certainly an interesting topic, which will presumably accompany us within the upcoming years…

Particulate Flow Modelling





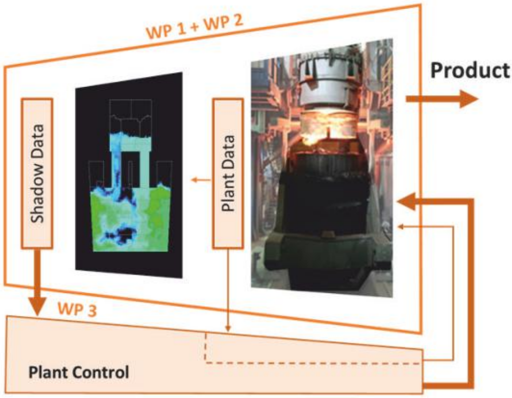


Fig. 1: Real-time simulation of Ti-homogenization in a RH plant (left).
Fig. 2: Comparison of (red lines) rCFD predictions with (symbols) point probes from plant trials (right, top).
Fig.3: Concept for including a digital shadow into an improved process control loop (right, bottom).

Stefan Pirker | stefan.pirker@jku.at

EDITORIAL IV | EXPERIMENTS & DATA ANALYSIS

Dear Readers,

developing something completely new is a demanding challenge and usually takes its time. Joint efforts of our colleagues from TU Graz/EMT, PFM and voestalpine finally lead to the first field capable capacitive tomography sensor which is in operation at the PCI plant of voestalpine Linz since May 2019. The first period of measurements brought a new insight into the behaviour of pneumatic powder conveying in a real industrial environment. Know-how transfer to our industrial partners has always been a strong commitment in our research activities. The 'Tomoflow' project is a perfect example that R&D engagements often do not pay off immediately but can bring you forward significantly on the long run.
Also the raceway monitoring project together with voestalpine Donawitz has finally brought an implementation of new data processing methods in the blast furnace process control system. We have collected data for more than one year and a now have a sound basis for long term statistics and the correlation with other process data.
Certainly, we also have a lot of ongoing lab activities and are currently setting up new experiments for the upcoming projects in funding period 2 of K1Met.

Sincerely,

Stefan Puttinger | stefan.puttinger@jku.at

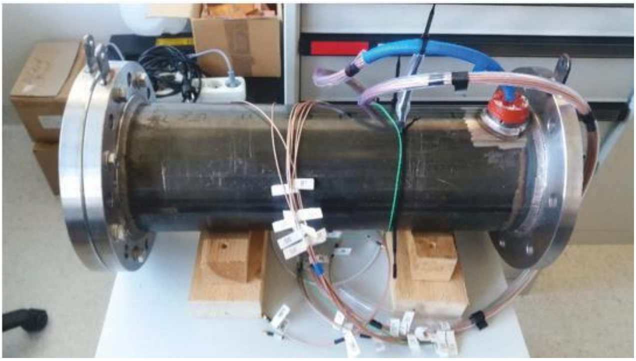


Fig.1: Electrical capacitance tomography (ECT) sensor before installation in the PCI plant of voestalpine.

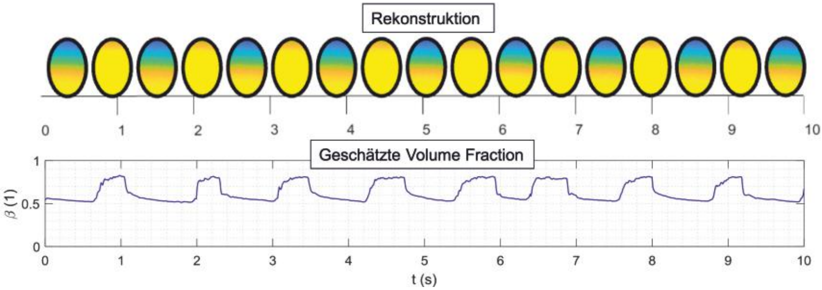


Fig.2: Reconstruction of particle distribution inside the pipe, recorded at 100Hz. The data clearly shows an unsteady transport mode (slug flow).

Fig.3: Visualization of particle cluster formation.

EXPERIMENTS & DATA ANALYSIS | CLUSTER FORMATION IN GAS-SOLID FLOWS

Understanding cluster formation is essential in modelling gas-solid flows. Inhomogeneous particle distribution will influence various aspects like e.g. gas-solid drag, chemical reactions and numerous others. PFM has published important contributions in modelling the sub grid effects of particle heterogeneity over the last years.

However, measuring cluster formation in medium mass loadings is very challenging and for very high mass loadings almost impossible. The main reason is that the optical accessibility in particle laden flows deteriorates very quickly.

To test the limits for our lab equipment, we built a glass box with a pneumatic dispersion unit. A short pulse of pressurized air releases a defined amount of particles into the gas volume of the box. The dispersion and re-sedimentation of the particles is captured with the high-speed camera while a continuous wave laser is used to illuminate a certain plane in the box (Fig. 1).

For particles in the range of 0 to 50 µm we are able to visualize cluster formation in a vey good quality. The amount of dispersed particles in the examples of Fig. 2 and 3 were 10g of spherical glass beads. This corresponds to a volumetric particle density of approximately 0.37 kg/m3.

By filtering the preprocessed images to a certain grid size we can export concentration fields that can be used for comparison with CFD simulations.

Particulate Flow Modelling

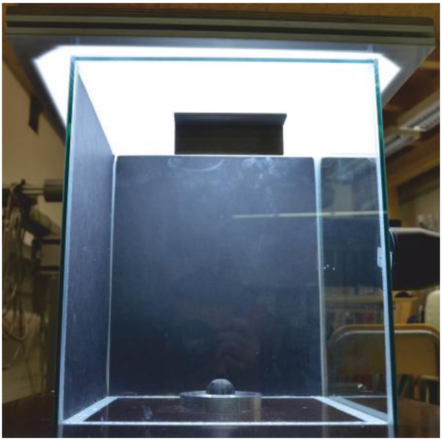


Fig.1: Glass box for particle cluster visualization. In addition to the LED panel we use a CW laser to illuminate a plane close to the front wall of the box.

Fig.2: Snapshot of the particle distribution inside the box recorded with the high-speed camera at 1500fps (image window is approx. 200 x200mm).

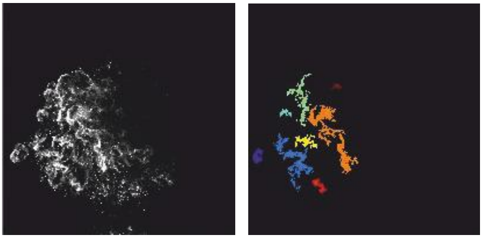


Fig.3: Close-up of the particle clustering captured with a telecentric lens and the identified clusters after image processing (window is approx. 40 x 40mm).

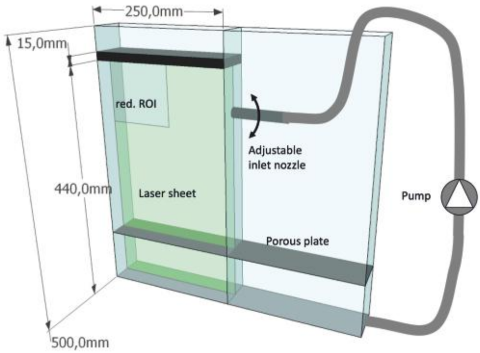
Michael Weiß | Stefan Puttinger

Stefan Puttinger | stefan.puttinger@jku.at

EXPERIMENTS & DATA ANALYSIS | LIQUID-LIQUID INTERFACES

Last years cover photo showed high resolution data from a time resolved PIV experiment on a submerged jet in a basin with two liquids of different densities. This setup corresponds to the typical mold flow situation in continuous casting of steel **(Fig. 1)**. The results demonstrated that we can fully resolve the turbulence spectra from the experimental data obtained with the high-speed camera at 1000Hz. With this data we are bale to validate the LES results of the main flow. However, the deformation of the liquid-liquid interface actually takes place on a much slower time-scale. Thus, image recordings with an ordinary industry camera running at 50Hz are sufficient to fully resolve the interface movements. In addition to the previous results we repeated the shadowgraphy measurements using a top view of the experiment to better visualize the formation of the open eye area (the area where the oil layer is completely pushed aside by the **deflected jet) (Fig. 2).**

Figure 3 shows, that the experimental results so far show much stronger deformations of the liquid-liquid interface than standard LES results (the improved ADM results are not yet available at the time of writing). This once again indicates that Smagorinsky type sub-grid models are not able to correctly capture the momentum exchange in multiphase flows.



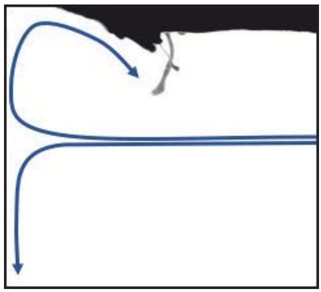


Fig.1: Benchmark experiment for investigating interfacial behavior in stratified two-fluid systems (left). Snapshot of the interface deformation in the side view taken via shadowgraphy.

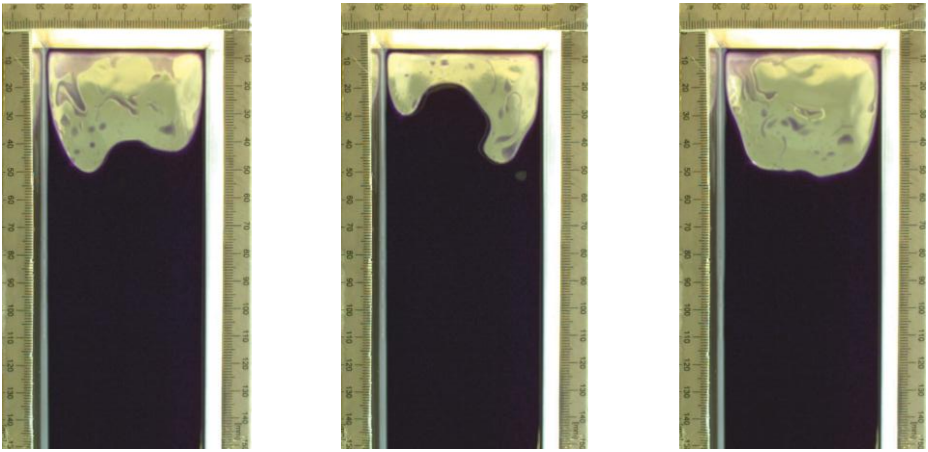


Fig.2: Snapshots of the water/oil experiment recorded from top of the basin. The images show the deformation of the water-oil interface due to the deflected submerged jet. The area of the so-called “open eye“ is then calculated via image processing.

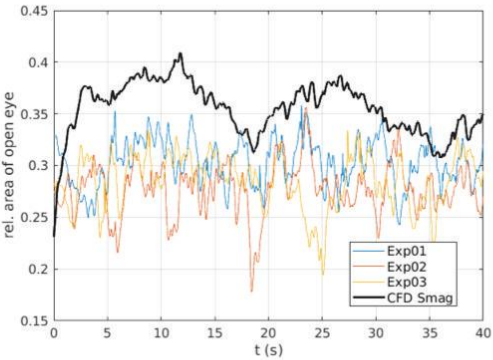


Fig.3: Comparison of the relative open eye area between experimental results and a standard LES model (Smagorinsky). The CFD results show a significantly larger area of the open eye.

Michael Weiß | Stefan Puttinger

Stefan Puttinger | stefan.puttinger@jku.at

EXPERIMENTS & DATA ANALYSIS | PARTICLE SEGREGATION

A common starting point for CFD and DEM simulations of particle flows is to assume a mono-dispers particle phase. However, all real life applications show a more or less broad particle size distribution or mixtures of various particle classes. Thus understanding segregation effects is essential for many applications. Die filling during the production of sintered components is one example from industry.
To visualize segregation of different particle classes we have set up a cascade of simple experiments in our lab to capture the major processing steps in die filling **(Fig. 1)**:

- mixing of two different types of particles from two hoppers into one
- hopper discharge of a particle mixture onto a transport chute
- segregation of particles during chute transport
- filling of a die from the chute

As usual, we want to extract some useful numbers from our experiments to compare the data with CFD/DEM results. Thus, we need to do some image processing to separate and quantify the amount of particles for each particle class. At the first glance separating two particles types of different colour might seem rather easy. In fact it is not trivial to find a robust solution which is independent to the light source or reflexions in the setup. Using the RGB colour space, for example, does not lead to a good recognition rate even if the particles are coloured red, green or blue. Instead transforming the images to the HSV colour space (HSV is based on hue, saturation and value and is better suited to mimic the human perception of colours) provides a perfect separation of the two different particle classes, no matter which colour combination is used **(Fig. 2)**. This allows to calculate a ”volume fraction” (in fact it is only planar data) on a regular grid size which is directly comparable to CFD/DEM results.

Particulate Flow Modelling

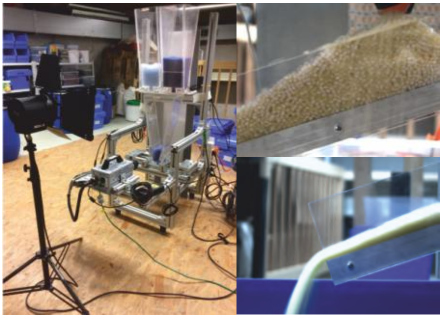


Fig.1: Images of the experimental setup. On the left we have the mixing of two different particle classes in one hopper. The right images show the mixed hopper discharge onto a chute and the segregation of particles during transport along the chute.

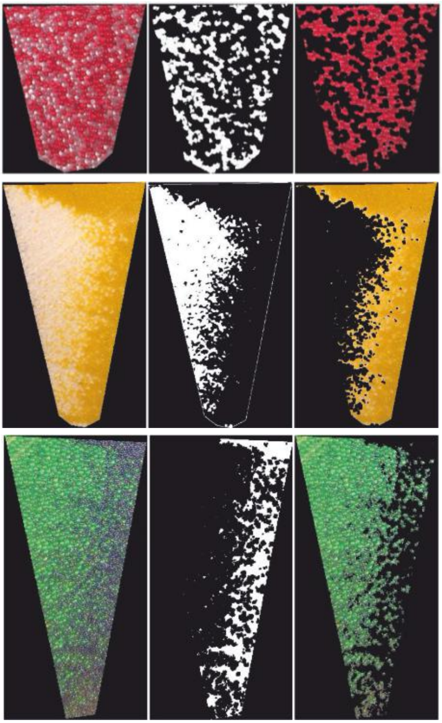


Fig.2: Examples of image processing results to separate particle classes of different size and colour.

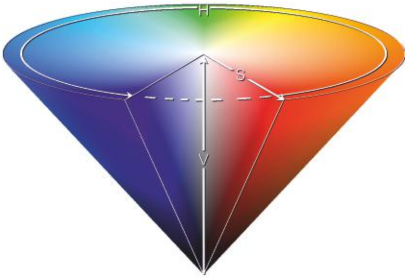


Fig.3: Cone model of the HSV colour space, hue is the angle on the color circle, the saturation is defined by the radius and value corresponds to the axial distance from black to white.

Michael Weiß | Stefan Puttinger


Stefan Puttinger | stefan.puttinger@jku.at

EXPERIMENTS & DATA ANALYSIS | FIELD TESTS OF ECT

Electrical capacitance tomography (ECT) is a well established measurement technique in the academic community since about 25 years. However, it has never seen a major adoption in industrial environments and there are still no commercial products on the market which are able to provide tomographic measurements of pipeline systems in industrial plants. So one could argue that there must be a reason why people don't use it in industry. Despite this negative perspective voestalpine together with the Institute of Electrical Measurement and Measurement Signal Processing (EMT) at Graz Technology University and PFM decided to give it a try and provided the necessary industrial background for a FFG Bridge project which was named "Tomoflow". As an outcome we installed the first field capable ECT sensor in May 2019 into the pipeline system of the pulverized coal injection (PCI) plant at voestalpine Linz (Fig. 1).

The environment for the sensor is quite challenging as the coal transport pipe has an average temperature of approximately 80°C and operates at pressure levels up to 16 bar. Therefore, a lot of issues had to be solved during the construction of the sensor as well as the signal processing electronics. The first months of operation showed a perfect long term stability of the signals, thus, we are quite optimistic that we have found the right solutions for all of these details (Fig. 2).

Pneumatic conveying for high mass throughput should usually be in the dense flow regime. For dense flow conveying the common assumption is that the pipe is almost entirely filled with particles an thus a minimum amount of conveying gas is needed. One of the major findings so far was that the conveying mode actually consists of a dense strand in the lower half of the pipe and travelling dunes at a rate of ~1Hz above the strand (in gas-liquid flows this would be considered as "slug-flow", Fig. 3). During the next months we will test if this transport mode is limited to certain operating conditions and if it is possible to trigger the flow regime to other modes by manipulating the pressure level and conveying gas flow rate. If it would be possible to reduce the needed amount of nitrogen for the coal pipelines this would reduce the operating costs of the PCI plant and thus have an economic impact.

Particulate Flow Modelling

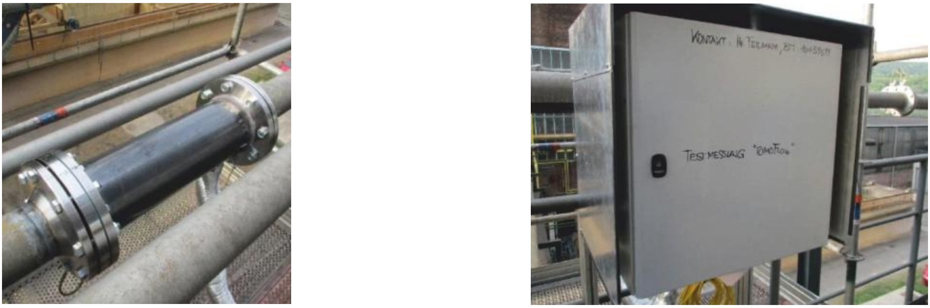


Fig.1: Sensor and signal processing unit installed at the main pipe for pulverized coal transport at voestalpine Linz. The pipe diameter is 150mm and operating temperatures are around 80°C (due to the pre-conditioning of the coal), thus temperature stability of the sensor and electronics is an important issue.

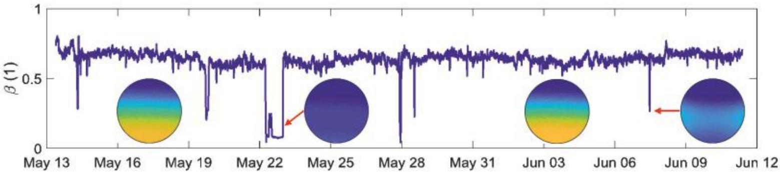


Fig.2: Long term signal of volumetric particle concentration covering one month of operation. The snapshots show examples of the reconstructed particle distribution inside the pipe.

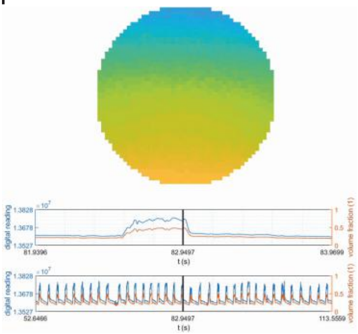


Fig.3: Measurements at 100Hz sampling rate show a slug flow regime with a slug frequency of approx. 1Hz.

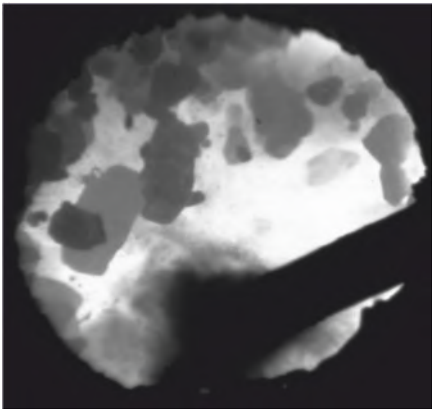
Markus Neumayer | Stefan Puttinger

Stefan Puttinger | stefan.puttinger@jku.at

EXPERIMENTS & DATA ANALYSIS | RACEWAY MONITORING OF BLAST FURNACES

In the final phase of project 2.1 in the first funding period of K1Met voestalpine Donawitz implemented the most promising approach for improved raceway blockage detection **(Fig.1)**. Since the new algorithms are running in the process control system since the beginning of 2019, we now have long term data available for statistical analysis of e.g. the occurrence frequency of blockages.

Figure 2 shows the distribution of blockage events around the two blast furnaces at voestalpine Donawitz. While BF1 has a rather even distribution of blockages around the circumference, BF4 shows some tuyeres which have a significantly higher number of blockages. As a second measure **Fig. 3** shows the strength of the blockage events (which is a non-dimensional integral of the time and amplitude of our detection signal). Here we see that most of the blockages of BF1 are very short events (and thus not relevant for operation) while BF4 has a higher number of very strong events (≥100). During the next project phase we will try to correlate our blockage data with other process data to better understand the causalities of raceway blockages during blast furnace operation.



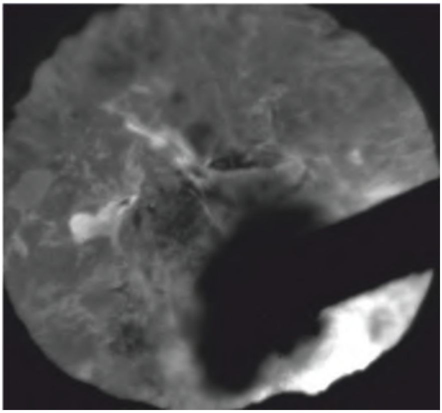
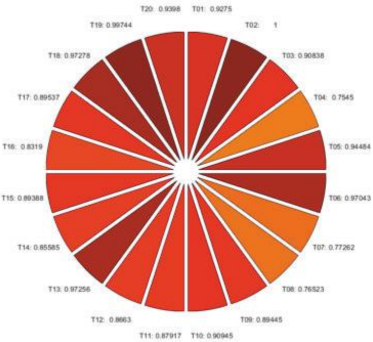


Fig.1: Ordinary operation (left) and entire blockage of the raceway (right) in a blast furnace for ironmaking.

metallurical competence center

Particulate Flow Modelling



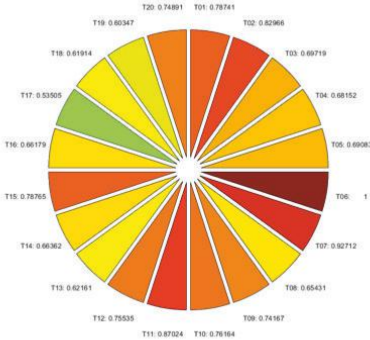
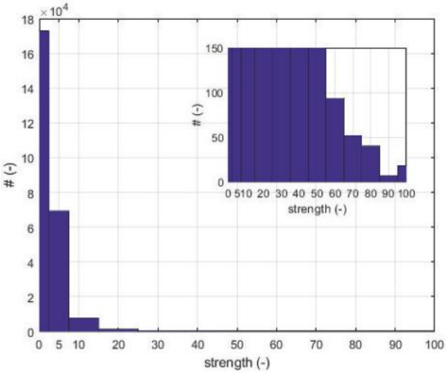


Fig. 2: Long term statistics of BF1 (left) and BF 4 (right) at voestalpine Donawitz for approx. 6500 hours of blast furnace operation. The numbers of detected blockages is given on a relative scale. While BF1 shows a more even distribution of blockages along the different tuyeres, BF4 has some tuyeres with a higher blockage frequency.



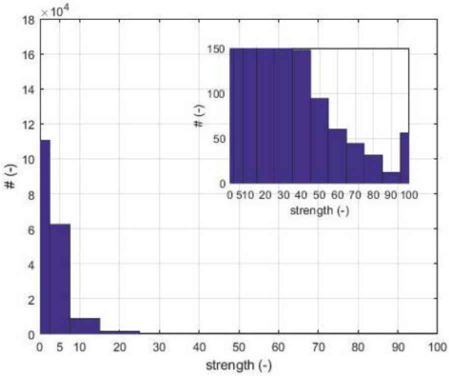


Fig. 3: Occurrence frequency vs. strenght of the blockage events. BF1 (left) shows a hich number of very weak events, while BF4 (right) has a higher number of very strong events (≥100).

Stefan Puttinger | stefan.puttinger@jku.at

SCIENTIFIC FRIENDS |
OUSMANE SAWADOGO & GERRIT BASSON, STELLENBOSCH UNIVERISTIY

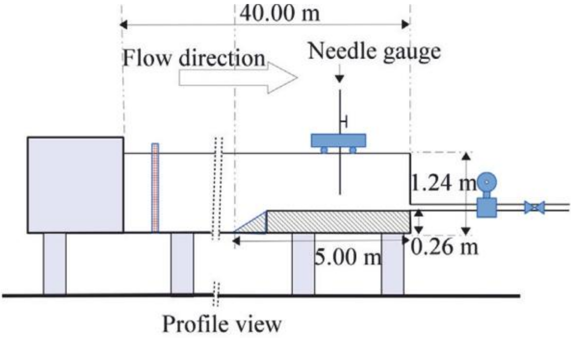
Reservoir sedimentation is not only a serious threat to water storage capacity but it can also block hydraulic structures such as hydropower intakes and bottom outlets. For this reason, reservoir sedimentation has been a topic of high interest for engineers for many years. Prediction of sedimentation in a reservoir, therefore, becomes crucial and could help in the planning and design of reservoirs. However, many reservoirs experience sediment deposition beyond the acceptable level. To deal with reservoir sedimentation some remedial measures exist to prevent incoming sediment from settling in the reservoir or to remove the existing deposited sediment.

Flood flushing with water level drawdown and free outflow conditions, is one of the efficient ways to remove sediment deposits in reservoirs. By increasing the flow velocities the already deposited sediment is re-entrained and transported through the bottom outlet. The width as well as the slope of the river channel will tend to the original regime conditions for periodic flushing events but this depends on the availability of excess water.

Thus, an alternative may be local flushing, where the water level is kept at or above the minimum operating level. Consequently, the flow velocity is much less compared to the drawdown flushing method. These combined conditions lead to the development of a local scour hole just upstream of the bottom outlet. This hole has a conical shape and develops until the slope of the scour hole is equal to the sediment angle of repose.
The current collaboration focuses on the development numerical model for predicting the scour development as well as sediment transport (figures 1 and 2). Thereby, a physical laboratory model of non-cohesive sediment bottom outlet flushing is investigated, and, secondly, coupled fully 3D numerical simulations of the bottom outlet flushing are performed (figure 3).

References:
[1] Sawadogo, O., G.R. Basson, S. Schneiderbauer. Int. J. Sediment Res., 2019, 34 (5), 461–474.oc to actually work on this project (first results are depicted in figure-2).

Particulate Flow Modelling



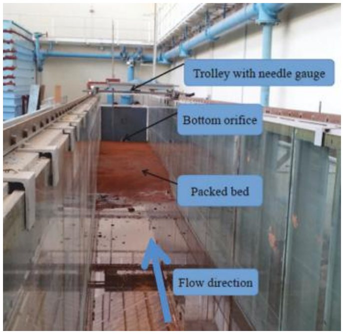


Fig.1: Sketch of experimental facility

Fig.2: Experimental setup of bottom outlet flushing process in reservoirs [1].

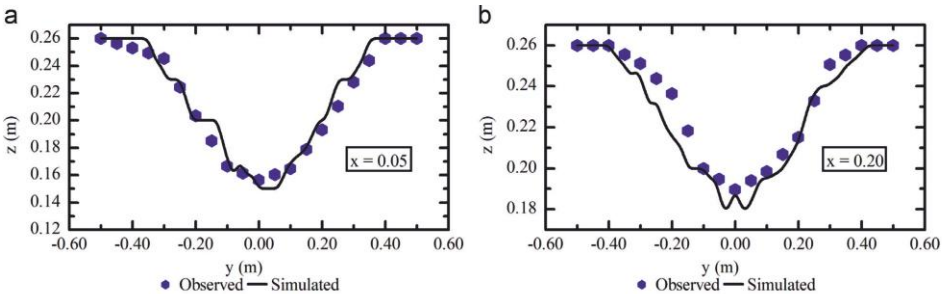
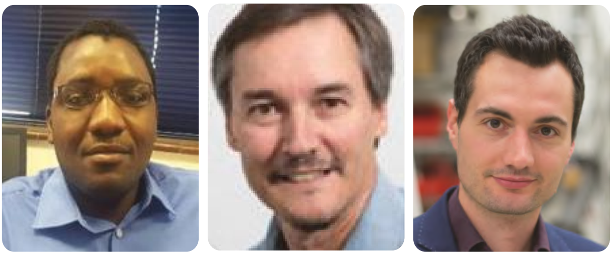


Fig.3: Comparison of simulated bed heights with experiments [1].

O. Sawadogo | G. Basson | S. Schneiderbauer



Simon Schneiderbauer | simon.schneiderbauer@jku.at

SELECTED PUBLICATION

Balachandran Nair A., Pirker S., Umundum T., Saeedipour M.: A reduced-order model for deformable particles with application in bio-microfluidics, in: Computational Particle Mechanics, Page(s) 1-9, 2019.

Kieckhefen P., Lichtenegger T., Pietsch S., Pirker S., Heinrich S.: Simulation of spray coating in a spouted bed using recurrence CFD, in: Particuology, Volume 42, Page(s) 92-103, 2019.

Lichtenegger T., Kieckhefen P., Heinrich S., Pirker S.: Dynamics and long-time behavior of gas–solid flows on recurrent-transient backgrounds, in: Chemical Engineering Journal, Volume 364, Page(s) 562-577, 2019.

Puttinger S., Stocker H.: Improving blast furnace raceway blockage detection. Part I: Classification of blockage events and processing framework, in: ISIJ International, Volume 59, Number 3, Page(s) 466-473, 2019.

Saeedipour M., Schneiderbauer S.: A new approach to include surface tension in the subgrid eddy viscosity for the two- phase LES, in: International Journal of Multiphase Flow, Volume 121, Page(s) 103128, 2019.

Saeedipour M., Vincent S., Pirker S.: Large eddy simulation of turbulent interfacial flows using Approximate Deconvolution Model, in: International Journal of Multiphase Flow, Volume 112, Page(s) 286-299, 2019.

Sawadogo O., Basson G., Schneiderbauer S.: Physical and coupled fully three-dimensional numerical modeling of pressurized bottom outlet flushing processes in reservoirs, in: International Journal of Sediment Research, Volume 34, Number 5, Page(s) 461-474, 2019.

Schneiderbauer S., Saeedipour M.: Numerical simulation of turbulent gas-solid flow using an approximate deconvolution model, in: International Journal of Multiphase Flow, Volume 114, Page(s) 287-302, 2019.

Imprint:

DEPARTMENT OF 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