2476

FILTECH 2011
CONFERENCE PROCEEDINGS
VOLUME I

CONTENT VOLUME I

CONTENT VOLUME II

Conference Dates:
March 22 – 24, 2011

Venue:
Rhein-Main-Hallen · Rheinstr. 20 · 65028 Wiesbaden · Germany

Organizer:
Filtech Exhibitions Germany
PO Box 1225 · 40637 Meerbusch – Germany
phone: +49 (0) 2132 93 57 60
fax: +49 (0) 2132 93 57 62
e-mail: Info@Filtech.de
web: www.Filtech.de

ISBN 978-3-941655-011-0

USING NUMERICAL FLOW AND PARTICLE SIMULATION TO PREDICT THE SEPARATION PERFORMANCE OF A CENTRIFUGE

Dipl.-Ing. X. Romaní Fernández*; Prof. Dr.-Ing. H. Nirschl
Karlsruhe Institute of Technology (KIT) Campus South
Institute of Mechanical Process Engineering and Mechanics

ABSTRACT

The prediction of the separation efficiency of a centrifuge is often limited to the cut size and to the assumption of a certain flow inside the centrifuge. With the knowledge of the precise flow and the sedimentation behaviour inside a solid bowl centrifuge it is possible to predict the separation efficiency accurately. Computational Fluid Dynamics (CFD) was used to simulate the multiphase flow of liquid, gas and particles in some centrifuges. To simulate the particles the Discrete Element Method (DEM) was also applied. The flow patterns in tangential, axial and radial direction, as well as the particle paths, are analysed and compared for the different centrifuges. To validate the simulated flow field, flow measurements via Laser Doppler Anemometry (LDA) were performed in a downscaled transparent replica of an industrial centrifuge. The separation efficiency obtained in the experiments with the real centrifuge serves to confirm the particle simulation model.

KEYWORDS

Solid bowl centrifuge, CFD-Simulation, Multiphase Flow, LDA.

1. Introduction

The prediction of the separation efficiency of a centrifuge is often limited to the cut size, the smallest particle size that can settle on the bowl wall for certain operation conditions. Some analytical models allow the calculation of the cut size assuming a certain internal flow in the centrifuge. In a rotating bowl the main flow occurs in tangential direction, but the flow in axial direction is the one that determines the residence time of the particles inside the centrifuge and therefore the settling condition. The two most common flow models assumed in centrifuges are the plug flow model, where the whole liquid in the rotating pool has the same axial velocity directed to the outlet, and the boundary layer model [1] [2] [3], where just a thin layer of liquid at the interface between air core and rotating pool has axial velocity directed to the outlet. But in industrial solid bowl centrifuges, with complicated shapes and internal structures, it is difficult to determine the flow patterns. Thus, the prediction of the cut size becomes difficult and inaccurate. Moreover the analytical models use the Stokes law to calculate the settling velocity of the particles, which is only applicable for laminar flow regime and single particle sedimentation. Frequently, the predicted cut size of a centrifuge diverges from the real separation efficiency due to the fact that these assumptions are not strictly true in the real centrifugal conditions. For example, some turbulence commonly appears near the inlet and not all the length of the centrifuge can be effectively used for settling.

Due to the increasing computational resources and variety of simulation models, Computational Fluid Dynamics (CFD) becomes a relevant tool to analyse the flow inside industrial devices, and thus, also in centrifuges. Some researchers have recently attempted to simulate the flow in centrifuges [4] [5]. However, because of the high velocity gradients and strong interaction between phases, the simulation of the multiphase flow, and particularly, the sedimentation process, in centrifugal field is still a challenge. Some results have been already achieved and can be read in [6].

The aim of this study is to investigate the multiphase flow in an industrial solid bowl centrifuge. The strategy followed is to solve numerically the governing flow equations with the software FLUENT and the particle movement equations with the software EDEM. Some modifications in the geometry and its effect on the flow patterns are also investigated.

2. Model description

2.1 Mathematical formulation of the fluid

The Volume of Fluid (VOF) method, designed to track the interface of two phases that are not interpenetrating, simulates here the gas-liquid multiphase flow. The presence of solid particles was ignored for flow simulation purposes, which is an acceptable assumption for low solid concentration. A continuity conservation equation (Eq.1) is solved for each phase, and the volume fraction of each phase α_q in any cell has to obey Eq.2,

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) = 0, \tag{1}$$

$$\sum_{q=1}^{n} \alpha_q = 1. \tag{2}$$

As the flow inside the bowl is non-laminar, the Reynolds Averaged Navier-Stokes equation (Eq.3) was chosen to solve the velocity field. With the VOF method a single momentum equation is solved throughout the domain and the resulting velocity field is shared among the phases. The dependency on the volume fraction is implemented by using volume averaged values for the density ρ and the viscosity μ,

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \rho(\vec{v} \cdot \nabla)\vec{v} = -\nabla p + \nabla \cdot \tau + \nabla \cdot \tau_t + \vec{F}. \tag{3}$$

In Eq.1 and 3, v represents the velocity, p the pressure, τ the shear stress tensor, τ_t the tensor of turbulences and F an external volumetric force. All terms are discretized and calculated for each volume cell with the exception of $\tau_{t,}$ the tensor of turbulences, which is modelled with a k-ε model.

2.2 Mathematical formulation of the particles

There are different multiphase models available in order to simulate the particulate phase. The Euler-Lagrangian formulation was applied here, firstly without particle-particle interaction, with the Discrete Phase Model (DPM) existing in Fluent; and then using a particle-particle soft sphere contact model implemented in a commercial Discrete Element Method (DEM) software.

2.2.1. Discrete Phase Model

Once the flow has reached a quasi-steady state, which means that the interface between gas and liquid is stable and the outflow coincides the inflow, particle trajectories can be calculated with the Discrete Phase Model (DPM) available in Fluent. In this model spherical particles are considered as discrete phase dispersed in the continuous phase in a Lagrangian frame of reference. Adequate for low loaded flows, the Discrete Phase Model computes the current positions of particles integrating the force balance represented with Eq. 4. This equation takes into account the discrete phase inertia, the hydrodynamic drag, the force of gravity and the rotational forces.

$$\frac{d\vec{v}_p}{dt} = F_D\left(\vec{v} - \vec{v}_p\right) + \frac{\vec{g}\left(\rho_p - \rho\right)}{\rho_p} + \vec{F}_r . \tag{4}$$

In Eq.4 v, v_p, ρ and ρ_p stand for the fluid velocity, particle velocity, the fluid density and particle density respectively.

One limitation of the DPM is the restriction to particle phase with low concentration because no particle-particle interaction is taken into account. The trajectory of the particles will be tracked until they reach the bowl wall (settled particles) or leave the domain with the outlet flow (not settled particles).

2.2.2. Discrete Element Method

In order to consider the sediment formation, the particle-particle interactions during settling and in the sediment must be considered. The Discrete Element Method (DEM) [7] considers the translation (Eq. 5) and rotation (Eq. 6) of a particle based on particle-particle and particle-wall collisions, electrostatic forces and van-der-Waals forces.

$$\frac{d\vec{v}_p}{dt} = \frac{1}{m_p}\sum F_i , \tag{5}$$

$$\frac{d\vec{\varpi}_p}{dt} = \frac{1}{I_p}\sum M_i . \tag{6}$$

Where v_p and ϖ_p are the linear and angular velocities of the particles, m_p and I_p are the mass and the moment of inertia of the particle respectively. M_i represents the momentum and F_i the different forces applied on each particle. The Hertz Mindlin contact model [8] was used for the particle-particle and particle-wall contact force. Electrostatic and van der Waals forces were not considered here because no submicron particles were simulated with this method.

But this methodology alone does not account for hydrodynamic forces. Through coupling of CFD and DEM both, hydrodynamic and contact forces on the particles, can be determined correctly. The fluid flow and particles movement have to be computed alternatively. First the flow will be determined with CFD, then the particle motion is calculated with the DEM but adding the hydrodynamic force thanks to the information about the flow velocities. Afterwards, the fluid flow will be computed again considering the influence of the particles as an additional source term in the momentum equations. For the flow computations all particles are reduced to mass points without volumetric extend.

3. Experimental set up

3.1. Industrial centrifuge

Experimental work was carried out on the industrial solid bowl centrifuge (Fig. 1) to confirm the simulation results. A suspension of water and quartz powder particles (with a mean size value of 2,07 µm and a geometric standard deviation of 2,15 µm) with a concentration of 0,4 vol.% was fed to the industrial centrifuge. Samples of the suspension and of the centrate were taken respectively at the inlet and at the outlet of the centrifuge. Its particle size distribution was analyzed with a static laser scattering method to calculate the grade efficiency. The grade efficiency characterizes the quality of the solid-liquid separation as a function of the particle size. The comparison of the simulated and the experimental grade efficiency is used for the validation of the particle simulation model.

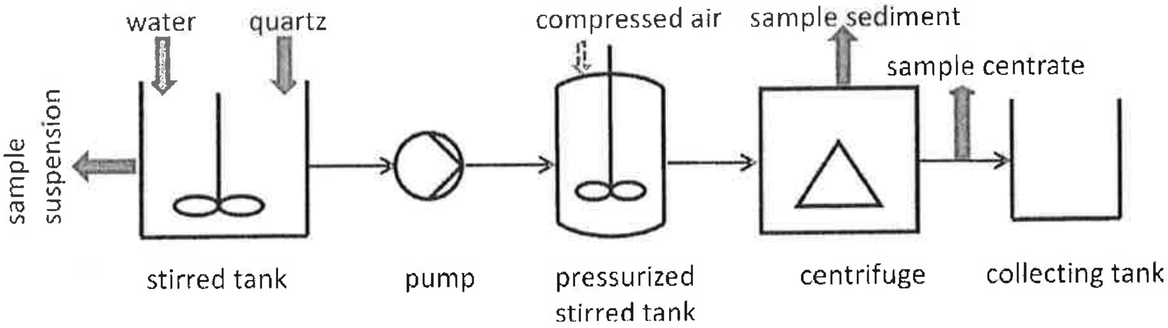


Fig. 1. Flow diagram of the experimental set up with the industrial centrifuge.

3.2. Plexiglas centrifuge

To confirm the simulated flow field, a downscaled transparent replica of the centrifuge was built, permitting the optical entrance necessary for the flow measurements via Laser Doppler Anemometry (LDA) (Fig. 2).

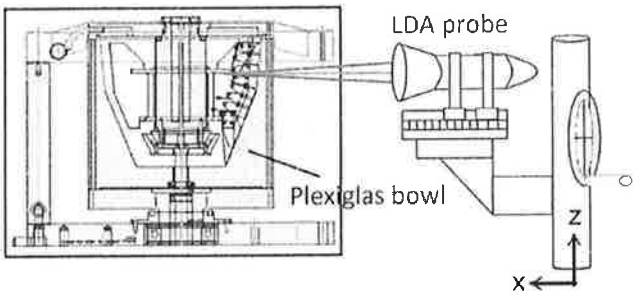


Fig. 2. Flow diagram of the experimental set up with the Plexiglas centrifuge.

For the measurements with the LDA, the flow must contain tracer particles with no drag but no particles are added for settling purposes. These tracer particles scatter the light as they move through the probe volume formed in the intersection of the laser beams. The detected scattered light contains information about the particle velocity, and hence about the flow velocity. The advantages of this method are that it is non-intrusive, it is direction sensitive, it has a high spatial and temporal resolution and there is no need for calibration. Nevertheless, because of the curved geometry and the intensity lost when the beam crosses the centrifuge drum, the data rate was not as good as expected.

Moreover the laser had to be triggered with the frequency of the centrifuge to be able to distinguish between the different angular positions. The triggering adds another uncertainty in the mess volume position of about ± 2°. The measurements can be performed through the Plexiglas bowl, to determine the tangential and axial velocities, or through two glass windows on the top of the centrifuge to measure the radial and tangential velocities.

4. Simulation Results and Validation

4.1. Flow patterns

For the numerical simulation, different centrifuge geometries were built and meshed (Fig. 3). The start point was the real geometry of an industrial solid bowl centrifuge with a conical bowl and radial baffles (a). Then, some modifications were done: the same geometry without baffles (b), the same geometry with a cylindrical bowl (c) and finally a replica of the centrifuge, a factor 2,4 smaller and with just two boreholes as outlets instead of eight due to the two glass windows on the top (d). Because of the periodicity of the system, it is possible to simulate just one fourth or one half of the centrifuge depending on the geometry.

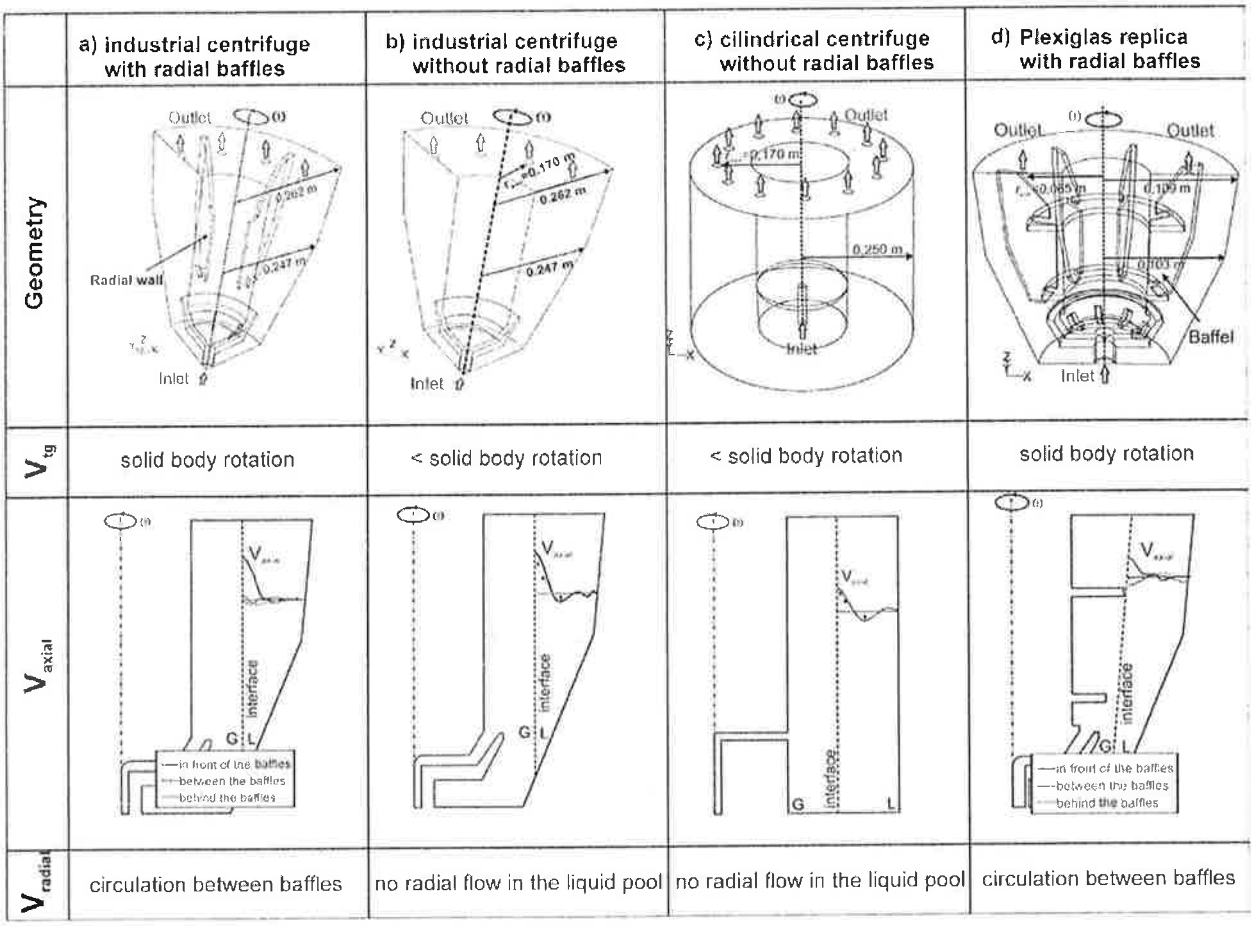


Fig. 3. Geometry and velocity profiles for the different centrifuges simulated.

The main flow occurs in the direction of the rotation speed of the bowl; both the liquid pool and the air core rotate. The tangential velocity of the rotating water achieves the velocity corresponding to a solid body rotation (SBR) only in the centrifuges with radial baffles (a, d). The liquid jet coming from the inlet accelerator has a lower tangential velocity than the rotating liquid pool when entering it. This underaccelerated inlet flow causes a slow down on the rotating pool of about 10%. But the radial baffles, which rotate with the same angular speed as the bowl, drive the water in the rotating pool to reach the solid body rotation velocity.

Regarding the axial flow, a boundary layer of fast moving fluid at the gas-liquid interface was observed in the simulations. In the centrifuges without radial baffles (b, c) this layer appears in a regular manner along the circumferential position. Instead, in the centrifuges with baffles (a, d), an irregular layer at the interface is formed and some regions, such as in front of the walls, acquire high positive axial velocity, whereas in others the axial velocity is almost zero or even negative. This happens because the radial baffles origin a secondary flow between them. The water describes a spiral path between the walls whereas it rotates around the axis, which disturbs the boundary layer at the gas-liquid interface

The radial flow in the centrifuges without baffles (b, c) was only observed at the inlet accelerator and on the jet leaving it. The radial component of velocity was negligible in the remaining domain. In the centrifuges with baffles (a, d) a radial circulation between the baffles was obtained in the simulations. In front of the baffle the water flows to the bowl wall and behind the baffles flows back to the gas-liquid interface. But the radial velocity values are a factor 100 smaller than the tangential velocity values.

For the validation, the values of the simulation of the Plexiglas centrifuge were compared with the LDA results for the three velocity components. The measured tangential velocity agrees with the values of the simulation (Fig. 4). Both, experimental and simulated results show that the velocity reaches the velocity of a solid body rotation. The results of the axial velocity agree with the simulations despite some noise in the measurements (Fig. 5). Near the gas-liquid interface (radius= 70-75mm) the axial velocity could be measured with high data rate (approx. 100 Hz). For higher radius, out of the boundary layer, the axial velocity decreases and takes values around zero, which causes a lower data rate. The radial velocity could not be measured because of the low values, out of the measurement device range.

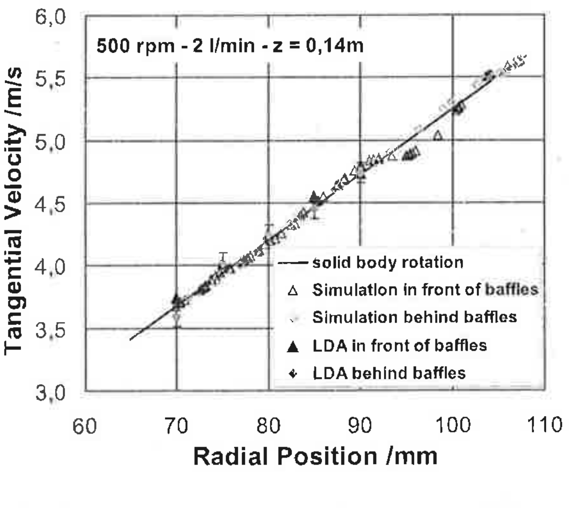


Fig. 4. Simulated tangential velocity values and values measured with the LDA over the radial position at 500 rpm and 2 l/min.

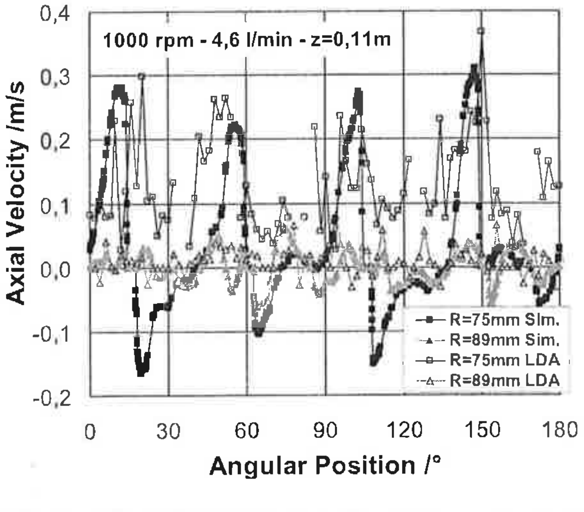
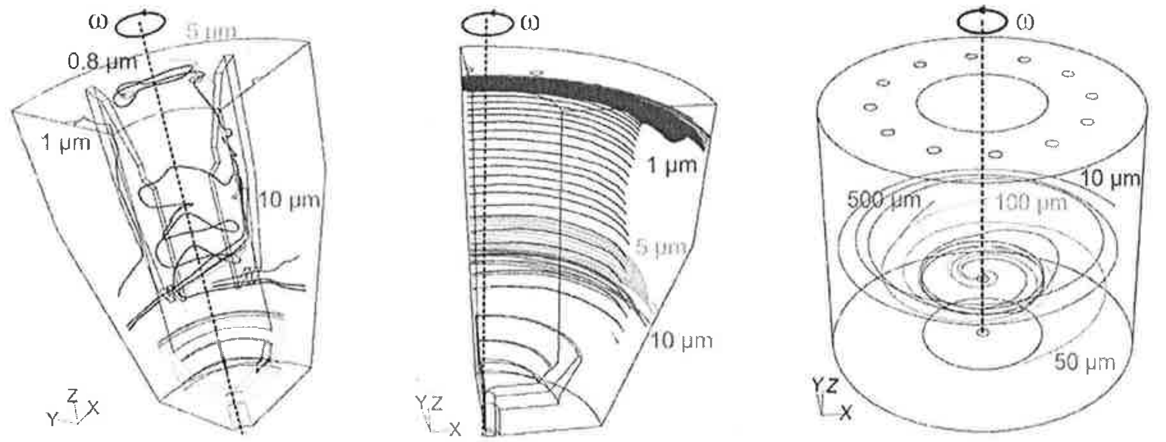


Fig. 5. Simulated axial velocity values and values measured with the LDA over the angular position at 1000 rpm and 4,6 l/min.

4.2. Particles

Once the flow simulation has converged, particles with different diameters (from 500 µm to 0,8 µm) were introduced in the domain and their trajectories were traced using the DPM. The results show different traces depending on the flow existing in the centrifuge (Fig. 6). In the geometry with radial baffles (a), the small particles get captured by the axial flow in front of the walls and the radial circulation between them, which hinders its sedimentation. For the geometries without baffles (b and c) particles do a spiral path, where the tangential movement is superposed to axial and radial movements. Particles follow the axial layers, while they accelerate towards the wall to settle. The grade efficiency calculated after the particle tracking agrees with the experimental results obtained in the industrial centrifuge.

As soon as the particles reach the wall, they disappear from the domain. Thus, the DEM was applied to consider the sediment formation. Because of the high computational time of these coupled CFD-DEM simulations, just particles with 500 µm were simulated until now. The results of this first simulation can be seen in Fig. 7, which represents the particle traces in the whole centrifuge (on the left), agreeing qualitatively with the DPM, and the particles settled at the bowl wall (on the right). Some efforts have to be done to reduce the particle size without increasing so much the computational time.



a) Industrial centrifuge with radial baffles

b) Industrial centrifuge without radial baffles

c) Cylindrical centrifuge without radial baffles

Fig. 6. Particle traces obtained with the DPM for the different centrifuges.

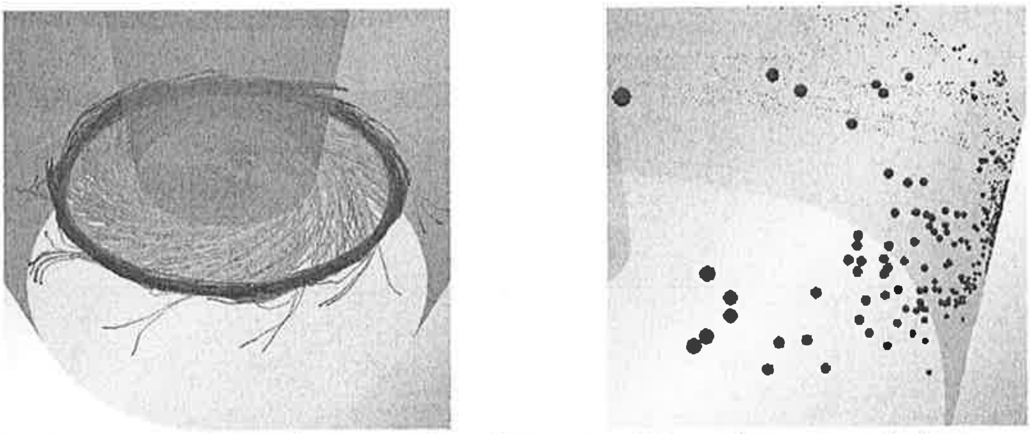


Fig. 7. Particle traces with DEM in the centrifuge (left) and particles settled at the wall (right).

5. Conclusions

Simulations of the multiphase flow in various solid bowl centrifuges were successfully performed with the software FLUENT and EDEM. The main flow takes place in tangential direction and, for the centrifuges with radial baffles; the tangential velocity reaches the solid body rotation. In the centrifuges without baffles the tangential velocity lies underneath the solid body rotation. A boundary layer of axial flow was observed at the gas-liquid interface in the simulations. In the centrifuges with radial baffles this layer appears irregularly along the circumferential position. In these centrifuges also a radial circulation between the baffles appears. The tangential and axial flow patterns were validated with Laser Doppler Anemometer measurements in a Plexiglas replica of one of the simulated centrifuges.

Particles with diameters from 500 µm to 0,8 µm were tracked using the Discrete Phase Model. The Discrete Element Method was coupled with the flow simulation to simulate particles with a diameter of 500 µm size. The results of both methods agree qualitatively, showing both the same particle traces.

After validation, the employed simulation models can be considered as generally valid for the multiphase flow in centrifugal field. With this approach, a tool able to predict the separation performance of a centrifuge with relatively high computational costs but high accuracy regarding the phenomena occurring inside the centrifuge bowl, is obtained.

6. References

[1] W. Gösele, *Schichtströmung in Röhrenzentrifugen*, Chemie-Ing.-Techn. 40. Jahrg., Heft 13 (1968) 657-659.

[2] H. Reuter, *Strömungen und Sedimentation in der Überlaufzentrifuge*, Chemie-Ing.-Techn. 39. Jahrg., Heft 5/6 (1967) 311-318.

[3] U. Glinka, Die Strömung in Überlaufzentrifugen – Neue Ergebnisse mit einem elektrolytischen Markierungsverfahren, Verfahrenstechnik, 5, (1983) 315-319.

[4] M. Jain, M. Paranandi, D. Roush, K. Göklen, W.J. Kelly, *Using CFD to Understand How Flow Patterns Affect Retention of Cell-Sized Particles in a Tubular Bowl Centrifuge*, Ind. Eng. Chem. Res., 44 (2005) 7876-7884.

[5] K.E. Wardle, T.R. Allen, *Computational Fluid Dynamics (CFD) Study of the flow in an Annular Centrifugal Contactor*, Separation Science and Technology, 41 (2006) 2225-2244.

[6] X. Romaní Fernández, H. Nirschl, *A Numerical Study of the Impact of Radial Baffles in solid Bowl Centrifuges Using computational Fluid Dynamics*, Physical Separation in Science and Engineering, vol. 2010, Article ID 510570 (2010) 10 pages.

[7] P. A. Cundall, O. D. L. Strack, *A discrete numerical model for granular assemblies*, Géotechnique, 29, 1 (1979) 47-65.

[8] R. D. Mindlin, *Compliance of elastic bodies in contact*, Journal of Applied Mechanis, Volume 71 (1949) 259-268.