CFD Simulations to investigate Flow and Particle Behaviour in a Solid Bowl Centrifuge

Prof. Dr.-Ing H. Nirschl, Dipl.-Ing. X. Romaní Fernández

Karlsruhe University, Institute for Mechanical Process Engineering,
Karlsruhe, Germany

Summary

This paper presents the results from the numerical simulation of the flow in an industrial centrifuge using the software FLUENT. The Volume of Fluid Model (VOF) is used to simulate the multiphase flow gas-liquid in the centrifuge. The liquid flow is analysed and compared with theoretical models. Once the flow has converged, solid particles are tracked with the Discrete Phase Model (DPM). Particle tracking is validated with the experimental grade efficiency of the centrifuge.

Keywords

CFD Simulation, Centrifuge, Volume of Fluid Model, Discrete Phase Model

1. Introduction

Centrifugation is a common operation to separate fine solid particles from industrial fluids, e.g. in glass and ceramics processing, in metal cutting. Because of the growing stringent environmental requirements, strict quality specifications and cost pressure, high purity of the fluids is mandatory. Among other centrifugal devices as decanters and disk stack separators, that are more complex, solid bowl centrifuges can be used to reach a satisfactory purity.

Flow and sedimentation are well described in tubular centrifuges [1, 2] and in overflow centrifuges [3], but the complex flow fields of industrial solid bowl centrifuges have not been amenable to rigorous mathematical analysis. The knowledge of the flow inside a solid bowl centrifuge is decisive for an optimization in terms of geometry and process conditions. The aim of the present investigation is to solve the governing flow equations in the centrifuge using the CFD software Fluent. Models of flow in some industrial process equipment have been developed using CFD, but only recently researchers have attempted to simulate the flow in centrifuges [4, 5, 6].

2. Model description

Fluent 6.3.26 was used to simulate the multiphase flow in the centrifuge. The presence of solid particles was ignored for flow simulation purposes, which is an acceptable assumption for low solid concentrations. The Volume of Fluid method (VOF) simulates the gas-liquid multiphase flow. This method is a simple and efficient formulation, designed to track the interface of two phases that are not interpenetrating. A continuity conservation equation is solved for each phase (Eq. 1) and, furthermore, the volume fraction of each phase α_q in any cell has to obey Eq. 2,

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

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

With the VOF method a single momentum equation, the Reynolds Averaged Navier-Stokes equation (Eq. 3), is solved throughout the domain, and the resulting velocity field is shared among the phases. The dependency on the volume fraction is provided through the density ρ and the viscosity μ, with volume averaged-values,

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

In Eq.1 and 3, v is the velocity, p the pressure, τ the shear stress tensor, τ_t the tensor of turbulences and F an external volumetric force. All the terms are discretized and calculated for each volume cell with the exception of $\tau_{t,}$ the tensor of turbulences, which will be modelled. The turbulence model considered was the k-ε renormalization group (RNG).

Once the flow has converged, a finite number of particles are injected into the domain and their trajectories are tracked with the Discrete Phase Model (DPM). This model, adequate for low loaded flows, calculates the positions of particles integrating the force balance (Eq. 4) until they reach the bowl wall (settled particles) or leave the domain with the outlet flow (not settled particles).

$$\frac{\mathrm{d}u_p}{\mathrm{d}t} = F_D\left(u - u_p\right) + \frac{g\left(\rho_p - \rho\right)}{\rho_p} + F_r \quad . \tag{4}$$

In Eq.4, u and ρ are the fluid velocity and fluid density, respectively, and u_p and ρ_p are the particle velocity and particle density. The first term on the right side is the drag force (F_D) per unit of mass and the second is the gravitational force, which is negligible compared to the rotational force (F_r).

2.1 Geometry and Mesh

Considering the real centrifuge geometry, a simplified three dimensional geometry, avoiding the inside mechanical parts, was built and meshed. Because of the periodicity of the system only one fourth of the centrifuge was simulated. The obtained grid has approximately 200000 tetrahedral cells.

The liquid enters through the inlet of the centrifuge into an accelerator, which consists of two rotating plates. Side and top walls of the bowl, defined with no-slip condition, rotate with the same angular velocity as the plates. The fluid leaves over a weir at the outlet; there, atmospheric pressure was imposed as boundary condition.

2.2 General Solution Method

Standard wall functions [7], most widely used for industrial flows, were applied at the walls. The spatial discretization scheme used was first order upwind [7]; except for the pressure, with a pressure staggered option scheme [7], recommended for high-speed rotating flows, and the volume fraction, with the Geo-Reconstruct scheme [7] to obtain a sharp interface between both phases. Pressure and velocity were coupled with the PISO schema [7] because of the transient calculation.

3. Results

The obtained gas-liquid interface is relatively sharp and it shows a profile with nearly constant radius along the height (Fig. 1). The jet of fluid coming from the accelerator describes a spiral path and impinges on the rotating liquid.

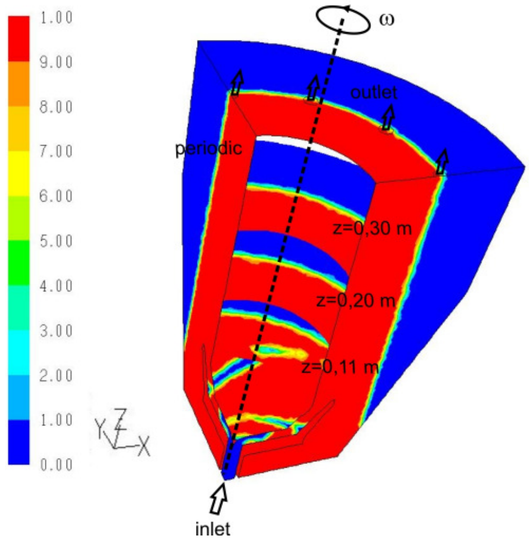

Fig. 1. Volume fraction of each phase (blue: liquid, red: gas).

The axial flow pattern from the simulations shows a boundary layer of fast moving fluid at the gas-liquid interface (Fig. 2). The thickness of this boundary layer agrees with the analytical value for tubular bowl centrifuges [2]. Alongside this layer a thin recirculation occurs. The rest of the liquid, which has nearly zero axial velocity, remains as a stagnant rotating liquid pool. At the outer wall a low recirculation appears.

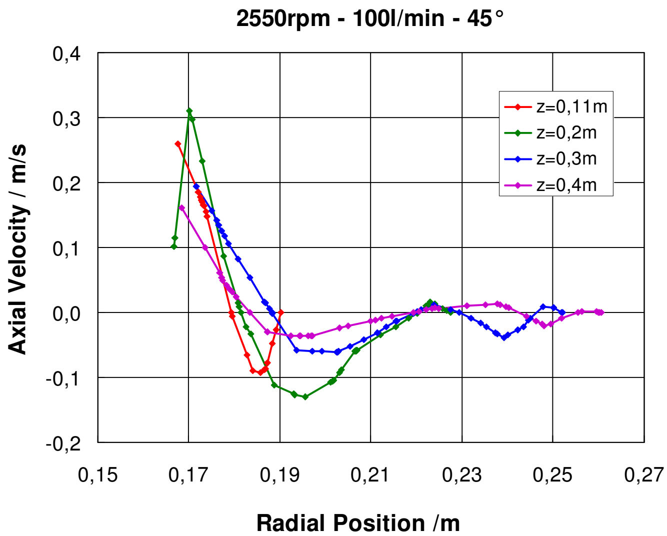

Fig. 2. Axial velocity of the liquid phase. Angular midplane (45°) at 100 l/min inlet flow and 2550 rpm.

Notable radial velocity was only observed at the inlet accelerator and on the jet leaving it. The liquid near the outlet has a slightly radial movement too. There, the axial boundary layer splits up into the outgoing stream and the stream that recirculates in the centrifuge.

A rigid-body motion profile was expected for the tangential velocities. The simulation shows values with the same trend but lying underneath the linear profile (Fig.3). This deceleration is caused by the jet flow coming from the inlet accelerator, which has almost no tangential movement when impacting the liquid pool and needs to be accelerated. Indeed, most fluctuations in the tangential velocity appear on the plane of the impinge point.

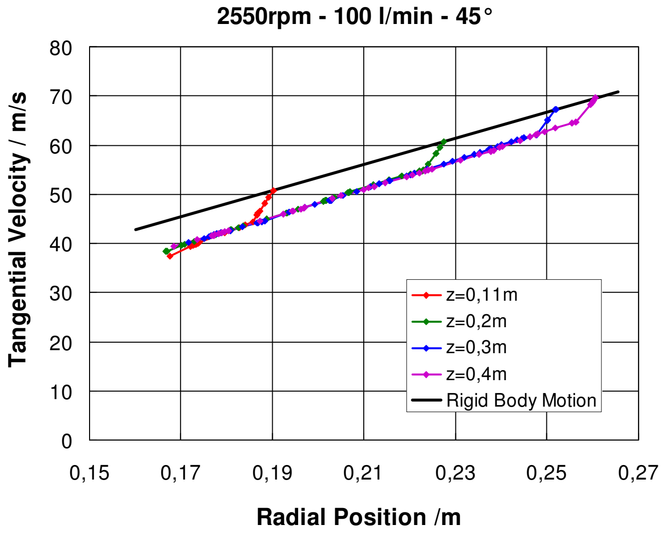

Fig. 3. Tangential velocity of the liquid phase. Angular midplane (45°) at 100 l/min inlet flow and 2550 rpm.

Once the flow simulation has converged, particles with different diameters were introduced in the domain at the point where the liquid impinges in the rotating pool, with the same velocity as the liquid. Their trajectories were traced using Eq. 4. The results show particles doing a spiral path, where the tangential movement is superposed to axial and radial movements (Fig. 4). Particles follow the axial layers, moving first upwards and then backwards. Finally they accelerate towards the wall to settle there. The smallest particles escape through the weir following the flow. The grade efficiency calculated for each particle agrees, in trend, with the grade efficiency reached during the experiments.

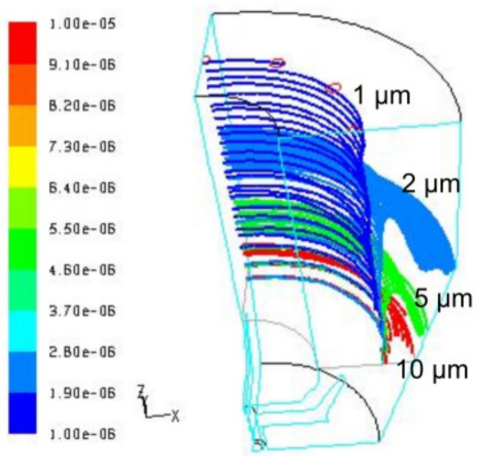

Fig. 4. Particle trajectories, coloured by particle diameter, $\rho_p = 2650$ kg/m^3.

4. Conclusions

The software program Fluent appeared to be an efficient tool for the analysis of the multiphase flow in a solid bowl centrifuge. The Volume of Fluid Model (VOF) was used to simulate the gas-liquid interface and the Discrete Phase Model (DPM) to track the particles.

Axial velocity profiles show a thin boundary layer at the gas-liquid interface moving fast towards the weir and a recirculation layer alongside. The thickness of this layer agrees with the analytics. Tangential velocities are smaller than the velocities corresponding to rigid body motion, because of the impacting liquid jet coming from the inlet, which is underaccelerated. Particle tracks were succesfully analysed.

5. References

[1] Bass E.: "Strömungen im Fliehkraftfeld I", Periodica polytechn., No. 3, 1959, pp. 321-339

[2] Gösele W.: "Schichtströmung in Röhrenzentrifugen", Chemie-Ing.-Techn., No. 13, 1968 pp. 657-659

[3] Reuter H.: "Strömungen und Sedimentation in der Überlaufzentrifuge", Chemie-Ing.-Techn., No. 5/6, 1967, pp. 311-318

[4] Boychyn M., Yim S. S. S., Shamlou P. A., Bulmer M., More J., Hoare M.: "Characterization of flow intensity in continuous centrifuges for the development of laboratory mimics" Chem. Eng. Science, No. 56, 2001, pp. 4759-4770

[5] Jain M. , Paranandi M., Roush D., Göklen K., Kelly W. J.: "Understand How Flow Patterns Affect Retention of Cell-Sized Particles in a Tubular Bowl Centrifuge", Ind. Eng. Chem. Res., No. 44, 2005, pp. 7876-7884

[6] Wardle K. E., Allen T. R.: "Computational Fluid Dynamics (CFD) Study of the flow in an Annular Centrifugal Contactor", Separation Science and Technology, No. 41, 2006, pp. 2225-2244

[7] Fluent 6.3 User's guide 2006.