XXII FLUID MECHANICS CONFERENCE
Słok near Bełchatów city, Poland, 11–14 September 2016

IID-3

Particle-fluid interaction inside a beater mill

M.J. Marijnissen, J. Rojek

Institute of Fundamental Technological Research, Polish Academy of Sciences,
Pawińskiego 5b, 02-106 Warsaw, Poland. E-mail: mmarijn@ippt.pan.pl

In this work a trajectory study of copper ore particles through a fan mill was performed with the use of a commercial CFD code, ANSYS Fluent, coupled with DEM (Discrete Element Method). Particles of different sizes were analysed. Results highlight ore behaviour, fluid flow conditions and mark places requiring geometrical improvements.

1. Introduction

Particle-fluid interaction plays a grand role in high speed comminution of ore. Any improvements made into this process have an immense impact on cost reduction. Standard ball mills have been known to take up to 50% of the ore production process's energy requirement [1]. Fan mills on the other hand seem to compete in energy efficiency whilst achieving high internal fluid velocities due to a spinning flywheel with its axis of revolutions normal to the flow. After hitting the flywheel the particles are shed upward into a filter, where particles small enough are passed further on towards the next process. Particles considered too big are recirculated back onto the flywheel. The analysis of particle trajectories allows for insight in the current design of fan mills and provides ground for further improvements. With high internal flow velocities a particle-fluid coupling is required to simulate the appropriate behaviour of the ore. This was achieved with a CFD-DEM coupling which has been used in simulations and validations of particulate flow in a channel [2], fluidized beds [3] and slurry flow [4]. Additionally it has successfully been tested against analytic and experimental data [5].

2. Methodology

The fluid was considered compressible and the flow turbulent. Particle volume fraction were taken into consideration during calculations. Additional required equations of state, turbulence and energy are not presented here.

$$\frac{d\alpha_q \rho_q}{dt} + \nabla \cdot (\alpha_q \rho_q \vec{u}_q) = 0 \tag{1}$$

$$\frac{d\alpha_q \rho_q \vec{u}_q}{dt} + \nabla \cdot (\alpha_q \rho_q \vec{u}_q \vec{u}_q) = \alpha_q \rho_q \rho F_i - \alpha_q \nabla p_r + \nabla \cdot \bar{\tau}_q + \vec{F}_{add,q} \tag{2}$$

$$\frac{d\vec{u}_p}{dt} = \vec{F}_{drag}(\vec{u} - \vec{u}_p) + g\frac{\rho - \rho_p}{\rho_p} + \frac{\vec{F}_{add}}{\rho_p} \tag{3}$$

The equation of continuity and momentum conservation for multiphase flow are shown in (1), (2) while the governing equation for the particle's motion is presented in (3). In these equations α is the phasic volume fractions, $\vec{u}$ is the velocity of the fluid, g gravity, ρ density, p_r pressure and subscript p denotes the particle's equivalent while q denotes the equivalent for a particular phase. Moreover $\bar{\tau}$ is the stress tensor, $\vec{F}_{drag}$ is the approximated drag force and $\vec{F}_{add}$ represents additional forces (ex. contact, coupling, pressure gradient). A two-way CFD-DEM coupling was used.

3. Simulation

The flow calculations were conducted for the full mill. Afterwards boundary condition profiles were passes over to each individual subdomain and particles were added. This was made possible due to a modular geometry connected with interface surfaces. For any particular domain approximately 2000 particles of different size were studied, which were injected after the flow conditions stabilized. Figure 1 shows this for the flywheel domain during operation.

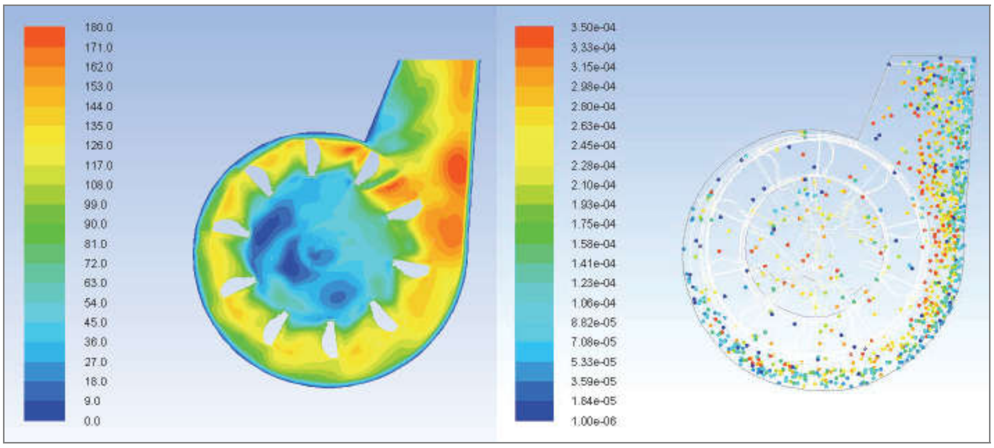

Fig. 1. Result obtained for flywheel domain, fluid velocity (left) [m/s], particles coloured by diameter (right) [m]

4. Results and major conclusions

- Particles tend to leave the flywheel domain with an outer side dominance.
- Bigger particles show a tendency to remain near the inner-middle side of the outlet.

Acknowledgements

The authors would like to acknowledge the financial support from the Polish NCBiR and KGHM granted upon the contract number CuBR/I/3/NCBR/2014.

References

[1] CLEARY P.W., *Predicting charge motion, power draw, segregation and wear in ball mills using discrete element methods*, Minerals Engineering, 1998, 11 (11).

[2] AZIMIAN M., LICHTI M., BART H.J., *Investigation of particulate flow in a channel by application of CFD, DEM and LDAPDA*, The Open Chemical Engineering Journal, 2014, 8, 1–11.

[3] YANG C., DUAN Y., *CFD-DEM model for simulating solid exchange in a dual-leg fluidized bed*, Chemical Engineering and Technology, 2013, 36 (11), 1907–1914.

[4] KUBICKI D., LO S., *Slurry transport in a pipeline Comparison of CFD and DEM models*, 9th International Conference on CFD in the Minerals and Process Industries, 10–12 December, 2012.

[5] KLOSS C., GONIVA C., HAGER A., AMBERGER S., PIRKER S., *Models, algorithms and validation for opensource DEM and CFD-DEM*, Progress in Computational Fluid Dynamics, 2013, 12 (2/3), 140–152.