CFD Based Optimisation of a Laboratory Scale Silo

A. Egedy[1], A. Rado[1]

[1]University of Pannonia, Faculty of Engineering, 10th Egyetem Str. Veszprém, Hungary

Abstract

The flow simulation of solid, particulate materials is still challenging, even though estimations show that three-quarters of products and raw materials are present as granules form in the industry. We can think of grain in the agriculture, raw materials of the glass industry, granulated products of plastic industry, etc. The large-scale storage of these substances is frequently carried out in silos. Design and optimisation of the silos is a challenging task. One of the main obstacles is to measure the parameters of these systems since the measurements change the flow conditions, so we do not get real data. Another task to be solved is to influence the flow conditions. During the discharge of the silos dead spots are caused by funnel flow or segregation. Inserts can be built into the system, and are used to influence the flow. The inserts are objects inside the silos, which can be occurred in several shapes and forms to achieve the desired effects.

There are two main methods for modelling these systems. The discrete element method (DEM) is a numerical method in which we define each particle of the material, and we calculate the parameters separately for each particle in each moment by specifying boundary conditions. The other method uses computational fluid dynamics (CFD) simulators. In this case, the solid-gas system is considered as a pseudo-plastic fluid. Thus the flow in the system can be simulated.

In this study, a laboratory scale silo was examined. The silo is a quasi 2D device, which can be used for video recording based validation. These types of devices can be used to understand the inner behaviour of the system, with less computational time, when we think of simulation. Two different coloured plastic particles were used for the experiments (white as bulk, and black as tracer layer). Residence time distribution was used for the validation of the model.

The system was implemented in COMSOL Multiphysics®, using a non-newtonian fluid to model the solid system as a continuum. Figure 1 shows results with one insert.

After the mesh independence study, the proper model parameters were identified. Laminar momentum balance and the component balance was used to model the residence time distribution. LiveLink™ for MATLAB® was used to calculate the optimal model parameters. Based on the results an optimal inner construction of the reactor was also proposed with optimal insert and reactor geometry.

Acknowledgement: Attila Egedy's research was supported by EFOP-3.6.1-16-2016-00015 Smart Specialization Strategy (S3) -Comprehensive Institutional Development Program at the University of Pannonia to Promote Sensible Individual Education and Career Choices project.

Figures used in the abstract

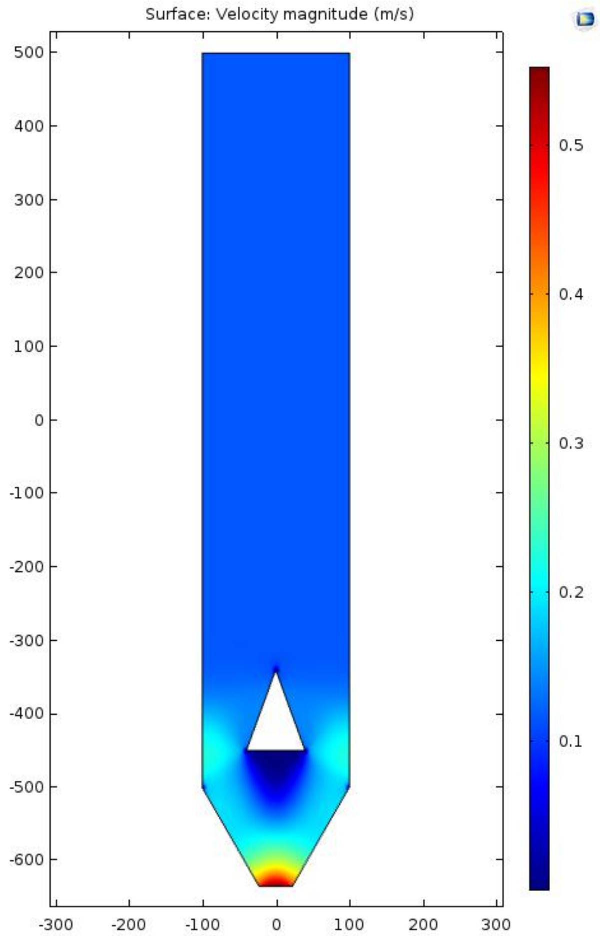

Figure 1: Figure 1 Velocity magnitudes within the silo.