CHAM

Modelling a Gravity Dust Catcher in a Steelmaking Plant

By Tom James and Michael Malin

October 2018

During steelmaking, blast furnaces release a dust-laden gas flow which enters a dust collection system for gas cleaning and recycling of particulate matter. The dust contains fine particles that are formed from the reactions taking place in the blast furnace. The main component of the collection system is a either a gravity or a cyclone dust catcher which separates the mixture of dusts from the spent gas flow. Recently, CHAM helped a major steelmaking company build a PHOENICS CFD model of a gravity dust catcher to calculate the particle separation efficiency of an existing design with a view to investigating geometrical modifications to improve the efficiency.

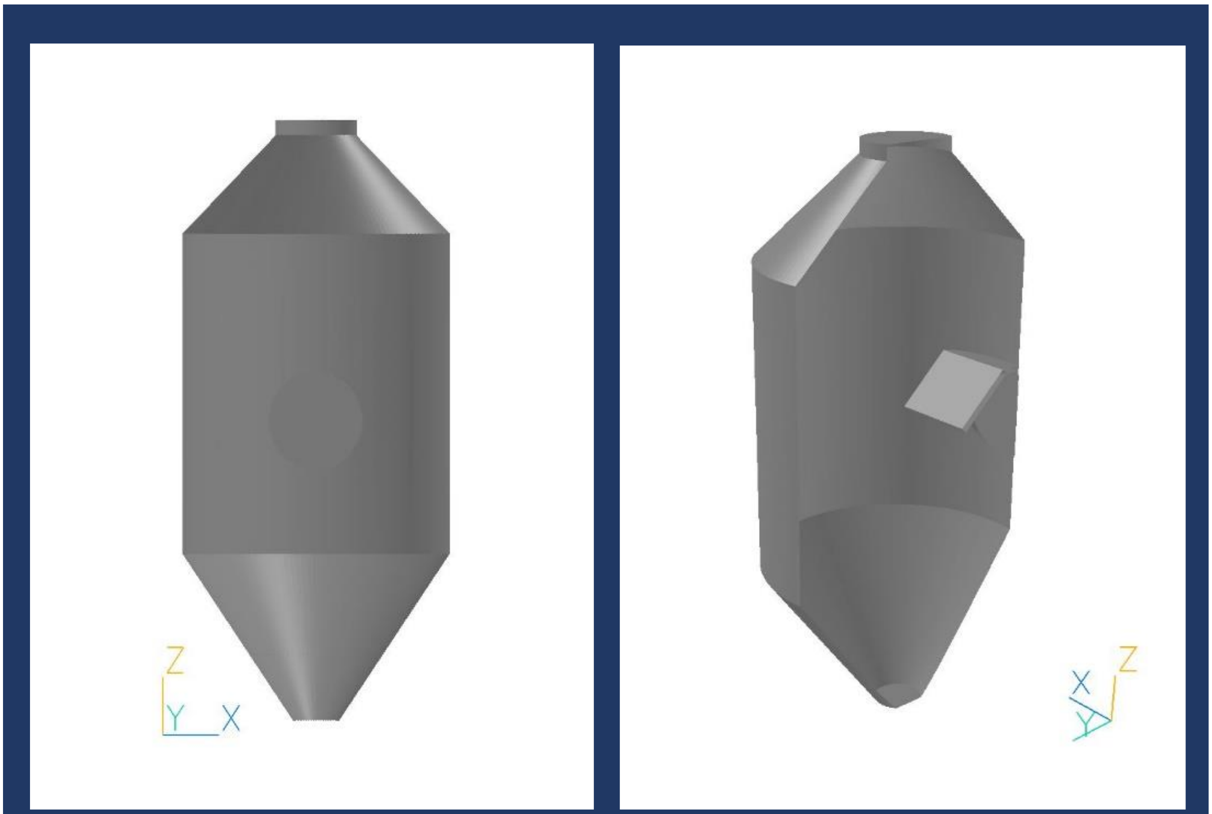

Figure 1: Gravity Dust Catcher –the geometry, including a cut-away perspective showing the internal guide positioned over the inlet.

As shown in Figure. 1, the gravity dust catcher comprises a side-entry inlet section, a main cylindrical separation chamber incorporating a dust-deflector plate, a lower funnel-shaped dust hopper, and an upper conical-shaped transition section leading to the gas outlet pipe. The dust catcher relies solely on gravity to separate out dust particles. Flow passes beneath the deflector plate into the main body of the unit before eventually turning upwards in the bottom third of the dust-catcher causing the heavier particles to deposit themselves in the bottom hopper.

The PHOENICS Cartesian cut-cell solver was sued to simulate the gas flow through the dust catcher. The two-equation Chen-Kim k-ε model was used to model turbulence. The volume fraction of the particulate material is typically less than 2% by mass, so an Eulerian-Lagrangian approach (GENTRA) was used to track this sample of dispersed particles through the flow field. Turbulent fluctuations of the gas phase can be expected to have a significant effect on the trajectory of dust particles, so this was accounted for by using a stochastic eddy-interaction model. The separation efficiency was obtained by releasing an even spread of mono-dispersed particles of 10 µm diameter across the inlet boundary, and then monitoring the number escaping through the top outlet boundary.

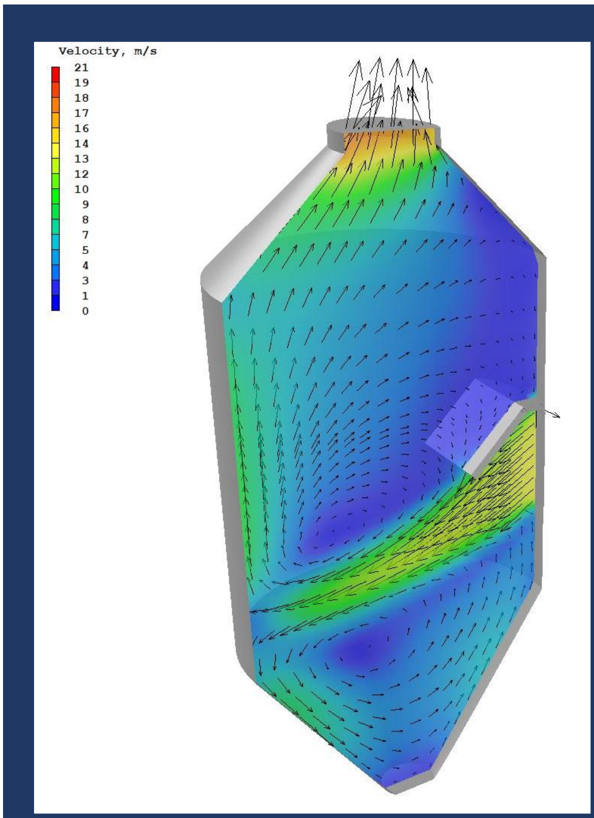

Figure 2: Gravity Dust Catcher – absolute gas velocity contours

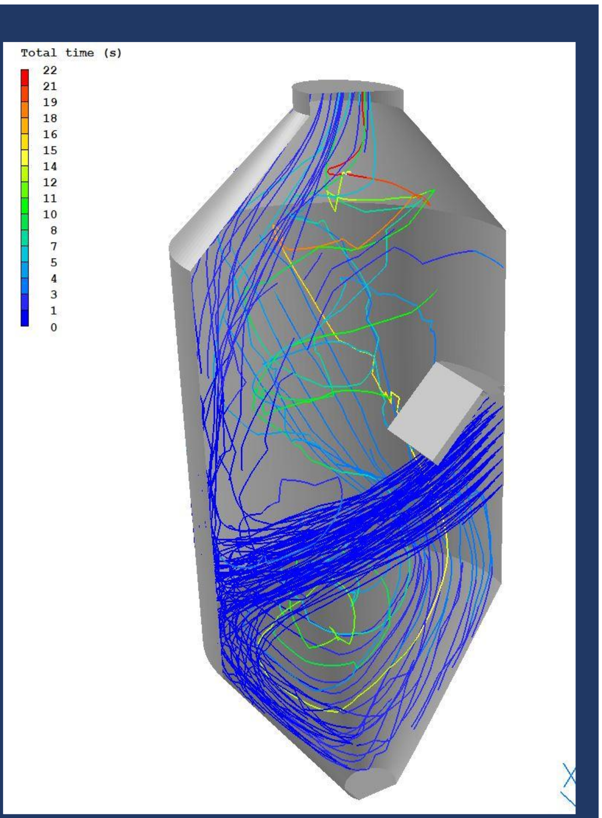

Figure 3: Gravity Dust Catcher – sample of dust particle trajectories coloured by total time

The predicted gas flow in shown in Figure 2 where one can see that the inflow is guided by the deflector plate into the bottom third of the dust catcher, where it then slips into an up-flowing and down-flowing stream after impingement on the far wall.

A sample of the particle trajectories emanating from the inlet is plotted in Figure.3, and for this basic design the CFD model predicts a collection efficiency of 45%.

In Figure.3 the trajectories are coloured with time, whereas in Figure 4. (below) they are coloured with the absolute gas velocity.

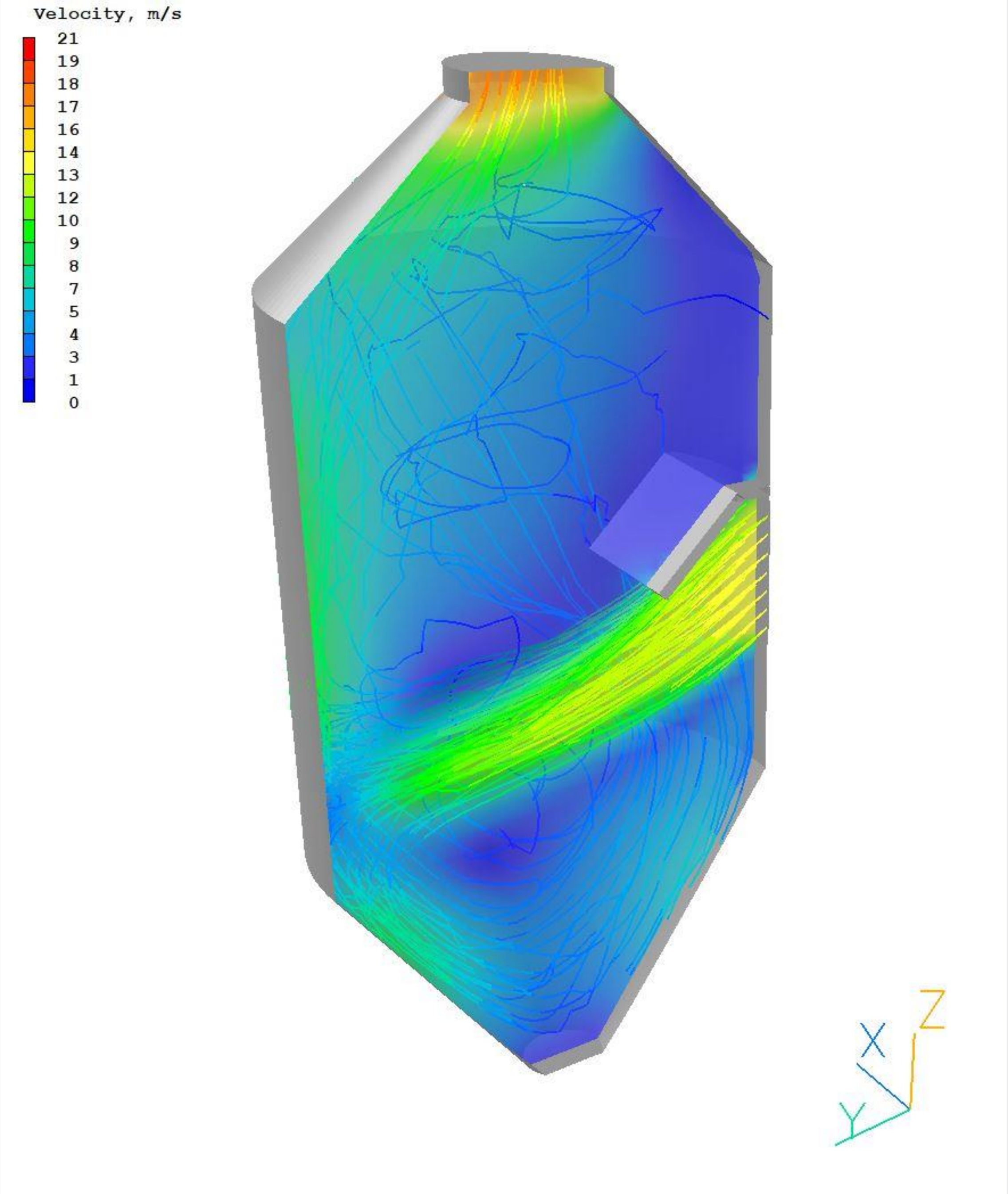

Figure 4: Gravity Dust Catcher – sample of dust particle trajectories coloured by absolute gas velocity

The next phase of the work is to assess the sensitivity of the model predictions to grid size, particle size distribution, increases in the particle population number and spread across the inlet boundary. Once the simulation of the basic design has been verified for numerical accuracy, the model will then be used to explore design alternatives aimed at improving collection efficiency.