On the Coupling of Computational Fluid Dynamics with Discrete Element Modelling

Berend van Wachem

Chair of Mechanical Process Engineering,
University of Magdeburg,
Universitaetsplatz 2, 39106 Magdeburg, Germany

Using a coupling of computational fluid dynamics (CFD) and discrete element modelling (DEM) is increasingly popular in industrial and academic communities, to simulate flows laden with particles. There are many academic codes, open-source codes and commercial codes that support this framework. However, not much is known about the accuracy of CFD/DEM. A key parameter to analyse the accuracy is the ratio of the particle diameter, d_p, and the mesh spacing, h. When the mesh spacing is very large compared to the particle diameter, the homogenization of the particle and fluid flow parameters assumed at the scale of the mesh removes the presence of the “sub-grid” scale details of the flow and the particle packing, and an accurate prediction of the gas-particle flow cannot be obtained. However, when the ratio of the particle diameter over the mesh spacing increases, it is also known that the accuracy of the CFD/DEM framework decreases. Although the community typically recommends particle diameters to be at least an order of magnitude smaller than the mesh spacing for particles to be accurately tracked, the errors corresponding to a given d_p/h ratio and/or flow regime have not been systematically studied and this makes estimating the error of a CFD/DEM simulation currently very difficult.

This presentation will contain of two parts. In the first part, we provide an expression to estimate the error in the fluid velocity at each particle which governs the transport of the particles in the CFD/DEM framework, based on the $\hat{d}_p = d_p/h$ ratio and the particle Reynolds number, Re_p. This enables us to determine the error made in Euler-Lagrange simulations and the result is shown in Figure 1. In the second part, we propose a correction scheme, based on the recovery to the local undisturbed flow velocity from the velocity field available on the Eulerian mesh. It relies upon the solution to the Stokes flow through a regularized momentum source and is extended to finite Reynolds numbers based on the Oseen flow solution. This correction is also extended to the case of a wall-bounded flow domains. We show that, in the Stokes regime, our method for the recovery of the locally averaged undisturbed velocity converges when the ratio between the particle size and the mesh spacing both decreases and when it increases. Applying our scheme to various cases, such as a settling particle, oscillating particles, and many particles in a packing, we show that the correction allows to nearly eliminate any impact of the mesh resolution on the accuracy of the particle's velocity prediction, and that this correction holds for a wide range of Reynolds numbers. We believe that such an improved CFD/DEM coupling strategies will significantly increase the accuracy of particle-laden flow predictions and will bridge the gap between fully-resolved particle simulations and CFD/DEM frameworks.

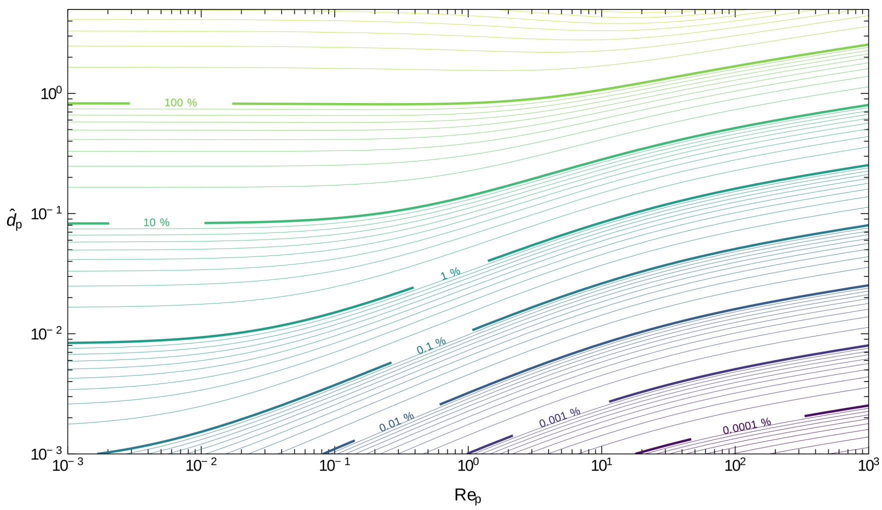

Figure 1: Amplitude of the normalized velocity disturbance (corresponding to the error in fluid velocity at the particle) as a function of the ratio between the particle diameter and the mesh spacing, d_p/h, and the particle Reynolds number, Re_p.