DEM-CFD modeling of fluidized beds

Manuel Bernard[1] – Eric Climent[2] – Anthony Wachs[1]

[1] *Fluid Mechanics Department, IFP Energies Nouvelles, 69360 Solaize*
[2] *Institut de Mécanique des Fluides de Toulouse, University of Toulouse - UMR CNRS 5502, 31400 Toulouse*

Fluid-particle flows are frequently encountered in industrial facilities and especially in chemical engineering processes. In this work, we focus on fluidized beds, which involve a fluid flow passing upward through a pack of particles with such a velocity that the fluid force acting on particles is larger than their weight. This technology is commonly used for its high heat and mass transfer rates in Circulating Fluidized Beds or Fluid Catalytic Cracking processes which are composed of millions of particles from a few micrometers to some millimeters, fluidized by a liquid or a gas. In order to optimize performances of those engineering processes, numerical simulations of multiphase flows became indispensable, especially over the last 20 years with the considerable rise of computational power and the progress in multiphase computational fluid dynamics. Depending on the length scale, there are three main approaches to simulate the dense particulate flows in fluidized beds (see fig 1).

At the micro-scale, the fluid motion equations are solved directly on a small mesh compared to the particle diameter. Direct numerical simulation (DNS) provides precise solutions but the number of fluid cells does not permit to simulate systems containing more than a few thousands particles in a reasonable computing time.

At the macro-scale, by using Eulerian methods, the fluid and solid phases are considered as two inter-penetrating media. The mesh is then coarser than a particle diameter and simulation of large domains, up the real size reactor, are doable. However, this length scale requires the introduction of numerous assumptions in the model to describe the evolution of the solid phase and its coupling with the surrounding fluid. Moreover, since the solid phase is considered as a continuous media, particle trajectories are not individually treated, which is a crucial lack of information for engineering processes.

At an intermediate scale between the DNS and Euler-Euler methods, the fluid is solved on a larger grid than the particle diameter, as it is in the Euler-Euler methods, but as in DNS methods, the particle trajectories, including collisions, are tracked with a discrete element method (DEM). This approach, commonly called discrete element method / computationnal fluid dynamics (DEM-CFD) or Euler-Lagrange method, was first introduced by Tsuji *et al.* [Tsuji et al., 1993] and Hoomans *et al.* [Hoomans et al., 1996]. This method has been widely developped and used since the beginning of the XXIst century (see *e.g.* [Kafui et al., 2002; Xu et al., 2000]).

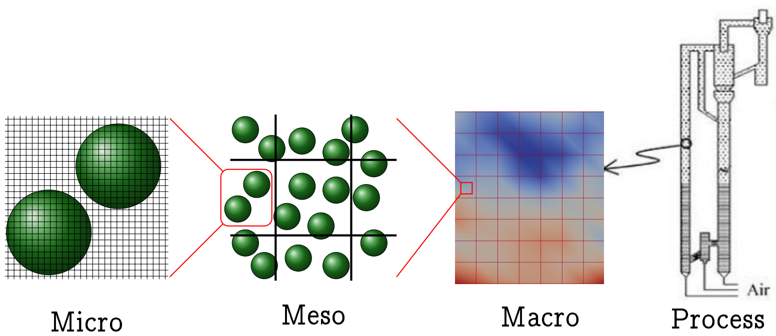

Figure 1: Different length scales used for simulating fluid-particle flows