ROCKY DEM and ANSYS CFD COUPLING: a powerful tool for simulating granular-fluid systems

The coupled DEM (Discrete Element Method) – CFD (Computational Fluid Dynamics) approach is a promising alternative for modeling granular-fluid systems, enlarging the range of coupled particle-fluid processes that can be managed with numerical simulations. Complex phenomena such as pneumatic conveying, granular drying, slurry flow inside grinding mills, or even chemical reactions between particles and fluids can be handled with these powerful tools.

A few successful coupling cases using DEM and CFD packages were published, along with experimental validation, which indicated the feasibility of this modeling approach to take into account both the effects of fluid volume displacement by particles as well as the drag force of the . Currently, there are two ways to couple DEM and CFD technologies: one-way coupling, in which only the fluid flow affects the particle movement, and two-way coupling, in which the particle flow also influences the continuous phase behavior.

However, considering the industry necessities, there is a strong desire to have an adequate results-guaranteed coupling. This is exactly why ESSS, ANSYS channel partner in Brazil, GDI, creator of the original ROCKY software, and ANSYS itself decided to combine their efforts and develop 1-way and 2-way coupling between ROCKY and FLUENT.

ROCKY is a powerful DEM software package, capable of performing 3D simulations of granular flow through a handling system. It uses real particle geometries (not a combination of spherical particles, for example), and simulates and analyzes aspects like 3D surface wear modification, particle breakage, sticky particles, and rotating and vibrating boundaries. For the past several releases, ROCKY has been coupled with Finite Element Analysis (FEA) ANSYS software, allowing engineers to evaluate the tensions stresses and forces generated by these particles as they interact with the handling equipment, such as chutes and conveyor belts.

ROCKY can run in parallel, under a CPU or, even better, a GPU cluster system. ANSYS FLUENT package is one of the world leaders for CFD applications, and its HPC CPU capability is very well documented. Both ROCKY and FLUENT have been extensively validated and tested.

Considering all these aspects, the work that ESSS, GDI and ANSYS accomplished now allows FLUENT to work with ROCKY in parallel CPU and GPU capabilities, which leads to a very trustworthy, robust, and fast DEM-CFD simulator. In addition, this parallel processing methodology opens up the possibility of simulating large multiphase flow devices, with many millions of different sized and shaped particles, which is historically a weak spot with DEM technology.

This integration is deeply rooted in both software packages. The interface inside FLUENT makes it easy to start this multiphase coupling with ROCKY (Figure 1). After a few simple steps, both solvers are set to work in parallel.

To show the coupling capabilities of ROCKY and FLUENT, some examples are presented below. A windshifter, equipment typically used in industrial waste processes to separate light from heavy particles, was modeled using both a pure DEM approach and a one-way coupling approach

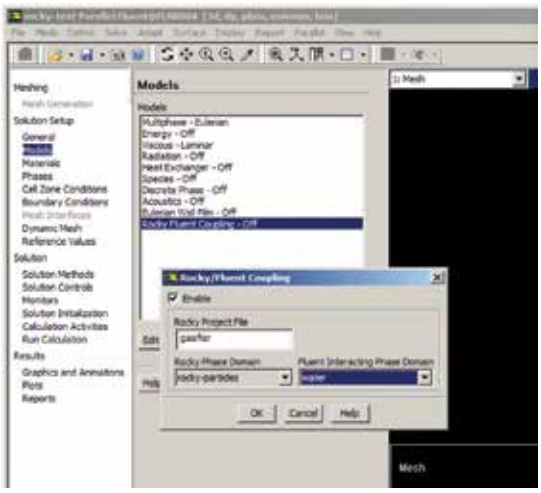


Figure 1. ROCKY-FLUENT setup panel inside of the Models panel of FLUENT. The interface provides an easy setup for two-way coupling simulations

Figure 2. Comparison between pure DEM (above) and one-way CFD-DEM approaches (below) for a windshifter case

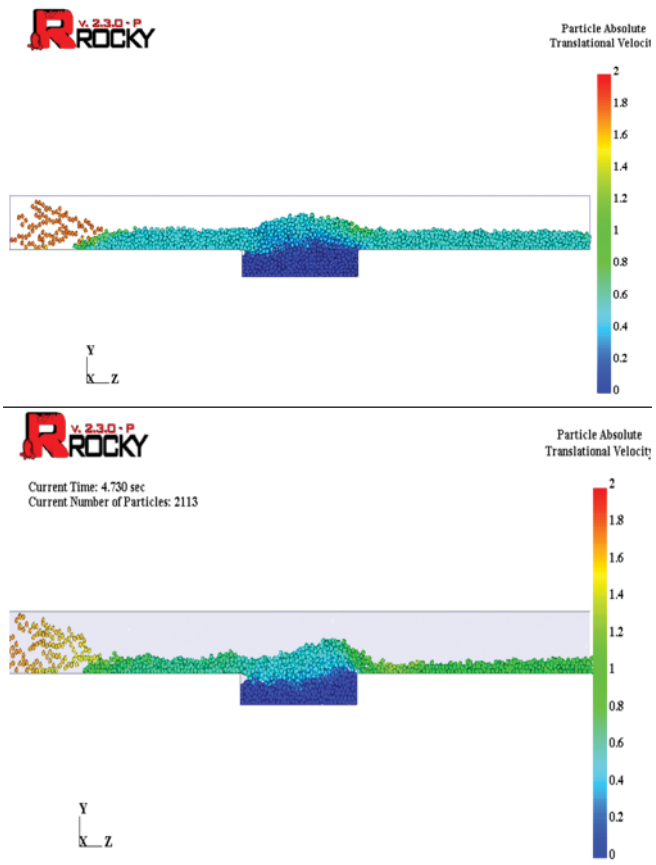
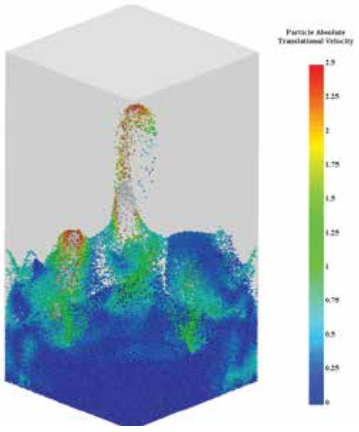


Figure 3. Comparison between one-way (above) and two-way (below) CFD-DEM approaches for a cavity filling case

between ROCKY and FLUENT. Particles were dropped into the shaft and, depending on their sizes, shapes, and densities, they went up (light fraction) or down (heavy fraction). The comparison in Figure 2 shows the differences between the two results and the possibility of investigating the influence of wind speed and feed rate on separation efficiency.

For comparing one-way and two-way DEM-CFD approaches, a simple cavity filling case was simulated and the simulation results for both approaches were compared (Figure 3). In the two-way case, the formation of a wave pattern of solids above the cavity can be observed.

A more complex case is shown in Figure 4. This case is the simulation of water injection (in three injection points) below a bed of initially still particles. The image at the top shows the ROCKY results for ~2s of physical simulation time. At this point of the simulation, the particle bed is entirely agitated and some rapid streams of particles are starting to be dragged as the fluid finds shortcuts and accelerates among the particles. The images at the bottom of Figure 4 show the results of water velocity from the FLUENT solution, and also shows the iso-surfaces of the particles volume fraction that have been imported from the ROCKY solution.



As can be seen in these examples, ROCKY- FLUENT coupling is a powerful tool for designing and troubleshooting particulate processes using simulation technology, enabling engineers to analyze a large range of processes in many different industries such as oil and gas (drilling, for example), agroindustry (grain transportation and drying, for example), pharmaceutic (transport), mining (mineral washing, for example), and many others.

João Américo Aguirre Oliveira Jr., Lucilla Coelho de Almeida, Clarissa Bergman Fonte

ESSS – Engineering Simulation and Scientific Software

Alexander Potapov

GDI – Granular Dynamics International

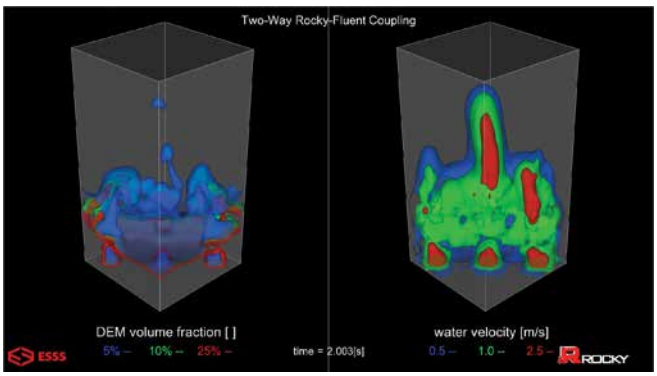


Figure 4. Two-way coupling simulation of injection of water below a bed of particles, ~400,000 particles. Images show ROCKY results (top) and FLUENT results (bottom) after ~2s of physical time