An Open Source CFD-DEM Perspective

DI Christoph Goniva, christoph.goniva@jku.at **DI Christoph Kloss**, christoph.kloss@jku.at **Alice Hager**, alihag@tele2.at **Dr Stefan Pirker**, stefan.pirker@jku.at	C.D. Laboratory on Particulate Flow Modelling 4020 Linz Austria

Abstract

We report on the synthesis of the Discrete Element Method (DEM) modelling the behaviour of granular materials and a finite volume model for the continuous interstitial fluid using Computational Fluid Dynamics (CFD). In [1], the authors presented a similar coupling approach for the commercial software packages EDEM and FLUENT along with experimental validation. They could show the feasibility of a modelling approach comprising the effect of (a) volume displacement by the particles, (b) drag of the fluid on the particles as well as (c) Magnus force due to particle rotation. With that approach, a wide range of coupled particle-fluid systems can be handled. Yet, due to shortcomings of the commercial software packages this approach is limited to shared memory machines. As real industrial problems are often characterized by high computational effort, the need for a DEM-CFD solver able to run on large clusters is obvious. To take a first step into this direction, the authors developed a coupled solver (“LammpsFoam”) joining the two software packages LAMMPS and OpenFOAM. While the former covers the DEM modelling, the latter describes the fluid dynamics of the continuous phase. The implementation of the code permits fully parallel simulations on distributed memory machines using MPI functionality.

This paper is organized as follows:

First, key results and validations from the commercially based coupling approach are shown. Secondly, a simple validation test case for the “LammpsFoam” solver is discussed before finally an outlook on future work is given.

Key words: CFD-DEM, LAMMPS, OpenFOAM,

References

[1] KLOSS, C., GONIVA, C., AICHINGER, G. and PIRKER, S. (2009): "Comprehensive DEM-DPM-CFD simulations: Model synthesis, experimental validation and scalability", Proceedings Seventh International Conference on CFD in the Minerals and Process Industries, CSIRO, Melbourne, Australia, December 9-11