Coupling of Finite-Volume-Method and incompressible Smoothed Particle Hydrodynamics Method for Multiphase flow

Christian A. Walter*, Manuel Hirschler and Ulrich Nieken

* Institute of Chemical Process Engineering (ICVT)
University of Stuttgart
Böblinger Straße 78,70199 Stuttgart, Germany
e-mail: christian.walter@icvt.uni-stuttgart.de, web page: http://www.icvt.uni-stuttgart.de

ABSTRACT

A drawback of the Smoothed Particle Hydrodynamics (SPH) method is its computational effort. An intuitive way to reduce the calculation time, is to use SPH only in parts of the problem domain where it is necessary, e.g. in a region where interfaces are present, because of the advantages of SPH for multiphase flow. In other parts of the problem domain, e.g. in a region of simple flow, a cheap method like the Finite-Volume method (FVM) may be used.

Coupling of grid-based and mesh-free simulation methods was done for solid-fluid systems, e.g. [1], in the past. Especially for SPH, only a few works on coupling of SPH and FVM are available in literature. For the incompressible SPH (ISPH) variant, which shows some similarities to FVM, only one approach to link fluid-fluid systems is available [2].

In this work, a general coupling strategy for combining ISPH with various grid-based methods is proposed. It is based on our previous work on open boundary conditions [3]. The basic idea is to divide the problem domain into different parts, using a different method in each part. At the interface open boundary conditions are applied. The different parts are then coupled by applying a two-way mapping of the boundary conditions. We demonstrate this strategy by using the FVM and highlight implementation details to enable a coupling to other grid-based methods.

For investigating the accuracy of the coupling strategy a simple Poiseuille flow in a channel where the simulation domain is divided into 3 parts (2 parts FVM and 1 part SPH) is used. In addition, the stability is discussed by moving the SPH part over time inside the channel. Finally, its applicability to multiphase applications is demonstrated by using a capillary rise that includes dynamic wetting. The results are then compared to analytic and numerical references.

REFERENCES

[1] G. R Johnson., "Linking of Lagrangian particle methods to standard finite element methods for high velocity impact computations", *Nuclear Engineering and Design*, **150(2)**, 265 – 274 (1994).

[2] E. Napoli, M. De Marchis, C. Gianguzzi, B. Milice and A.Monteleone, "A coupled Finite Volume-Smoothed Particle Hydrodynamics method for incompressible flows", *Computer Methods in Applied Mechanics and Engineering*, **310**, 674 – 693 (2016).

[3]] P. Kunz, M. Hirschler, M. Huber, U. Nieken, "Inflow/outflow with Dirichlet boundary conditions for pressure in ISPH", *Journal of Computational Physics*, **326**, 171 -187 (2016).