A FEM-SPH coupling approach for the simulation of FSI problems with free surfaces

Jessica Stasch*, Bircan Avci and Peter Wriggers

Institute of Continuum Mechanics (IKM)
Gottfried Wilhelm Leibniz Universität Hannover (LUH)
Appelstraße 11, 30459 Hannover, Germany
e-mail: stasch@ikm.uni-hannover.de, web page: http://www.ikm.uni-hannover.de/

ABSTRACT

In technical applications, like airbag deployment and biomechanical systems, elastic structures are used to control fluid flow. Depending on the characteristics of the structure and the fluid different effects have to be taken into account. For an accurate simulation of the fluid flow in those systems the used numerical method has to deal with fluid structure interactions (FSI). We propose a method to solve such coupled problems by the use of a combination of Smoothed Particle Hydrodynamics (SPH) for the fluid part and Finite Elements (FE) for the solids.

The SPH-method is solving the Navier-Stokes equations on the basis of a disordered set of freely moving interpolation points in space. Since a constant mass is assigned to every point, they are considered as particles, described in a Lagrangian framework. Every field variable such as velocity and pressure is evaluated using a kernel function or its derivatives. Using this approximation method short range forces are automatically computed with respect to the underlying interpolation space, the so called smoothing length. Moreover, as SPH is a mesh-free method, it easily allows the computation of large deformations and self-intersections when compared to mesh-based methods, where very sophisticated re-meshing strategies are mandatory.

For the calculation of the fluid state the GPU-based open-source framework 'DualSPHysics' is used [1]. The behavior of the structure is calculated using the Finite Element solver FEAP. In order to determine the exchanged energy between the solvers a FSI-coupling module is implemented which is tracking the interaction zone. Since the fluid as well as the structure are described in a Lagrangian framework, the interface tracking does not need any complex treatment. The major task in FSI-simulations is the correct transfer of the interaction forces and the fulfillment of the dynamic and kinematic interface conditions. In our SPH-FEM algorithm the energy of the domains is used as a criterion of the interface exchange. The reaction forces resulting from the structure onto the SPH fluid particles and vice verse are estimated using the semi-analytical wall boundary condition [2]. Thus a single layer of SPH particles is placed on the structures surface. These particles are fixed and build a barrier for the fluid particles.

While using two different solvers a partitioned coupling scheme is practicable. In our approach the coupled problem is solved by an iterative scheme developed by Farhat [3]. Both, the fluid and the solid equation, are solved consecutively and individually using the latest information supplied by the counterpart of each problem. Since the time step of the fluid and the solid calculation differs, an adaptive time scheme is used to improve the calculation efficiency.

REFERENCES

[1] Crespo, A. J. et al. "DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH)." Computer Physics Communications 187 (2015): 204-216.

[2] Ferrand, Martin et al. "Unified semi-analytical wall boundary conditions for inviscid, laminar or turbulent flows in the meshless SPH method." International Journal for Numerical Methods in Fluids 71.4 (2013): 446-472.

[3] Farhat, C. et al. "Mixed explicit/implicit time integration of coupled aeroelastic problems: Three-field formulation, geometric conservation and distributed solution." International Journal for Numerical Methods in Fluids 21.10 (1995): 807-835.