

COMPARISON OF COUPLED EULER-LAGRANGE AND SMOOTHED PARTICLE HYDRODYNAMICS IN FLUID-STRUCTURE INTERACTION

CHRISTIAN ZEHETNER, MARKUS SCHÖRGENHUMER,
FRANZ HAMMELMÜLLER AND ALEXANDER HUMER

Linz Center of Mechatronics GmbH
Altenbergerstraße 69
4040 Linz, Austria
e-mail: christian.zehetner@lcm.at, web page: www.lcm.at

Key words: Fluid-Structure Interaction, Smoothed Particle Hydrodynamics (SPH), Coupled Euler-Lagrange Finite Element Method (CEL)

Abstract. In this paper a fluid-structure interaction problem is investigated, in which fluid flow and flexible deformations of structures are coupled. Exemplarily, the collision of a moving deformable water-filled container with a rigid wall is considered. Two simulation methods are compared to analyze the impact: the Coupled Euler-Lagrange Finite Element Method (CEL) and Smoothed Particle Hydrodynamics Analysis (SPH). On the other hand, the solutions of two software packages are compared, the commercial Finite Element code Abaqus (CEL, SPH) and the open source package HOTINT/LIGGGHTS (SPH). Goal is to find the various advantages and disadvantages of the two simulation methods and the two software codes.

1 INTRODUCTION

In fluid-structure interaction problems, fluid flow and flexible deformations of structures are coupled. Examples are obstacles in fluid flow or the dynamics of fluid-filled structures. In this paper, two methods are compared for modelling and simulation of such processes.

First, the Coupled Euler Lagrange approach (CEL) is considered [1], where solid bodies and structures are modelled by Lagrangian finite elements in which the material is fixed to the element. On the other hand, the fluid is modelled by Eulerian finite elements which are fixed in space, and the material moves through the elements. The coupling is established by contact interactions, implemented as a penalty contact method.

In contrast, Smoothed Particle Hydrodynamics (SPH) [2] is a mesh-free Lagrangian method where the fluid is divided into discrete elements represented by particles. The interaction with solid structures, modelled by Lagrangian finite elements, is a crucial issue in today's SPH approaches. In the present work, penalty-based formulations are used to model the wall boundaries.

In the following computational study, a fluid-filled, flexible container colliding with an obstacle is considered. The resulting motion of the fluid, the deformation of the container, and the contact forces are investigated.

Three solutions are compared: (i) The multibody code HOTINT is coupled with the particle simulator LIGGGHTS which includes an SPH implementation [3]. Both packages are available as open-source software. The results of this co-simulation are compared with two formulations implemented in the commercial software Abaqus, i.e., the SPH formulation (ii) and the CEL approach (iii).

In section 2 the investigated problem is stated. In section 3 the simulation models are described in detail. Section 4 shows a comparison of the simulation results and a discussion of the advantages and disadvantages of the investigated simulation methods.

2 EXAMPLE PROBLEM

In the following we consider the example problem as shown in Figure 1. A water filled container is horizontally moving towards a rigid wall with constant speed v_0 , which is defined as initial condition. Friction between container and floor is neglected, and the gravitation g directed towards the floor is acting on both the container and the fluid. The impact due to the collision of the container with the wall causes an elastic deformation of the container and flow of the water. The deformation is assumed to be elastic, but large deformations are considered. Goal is to investigate the contact force between container and wall during impact, the deformation of the container, and the distribution and motion of the fluid after the impact.

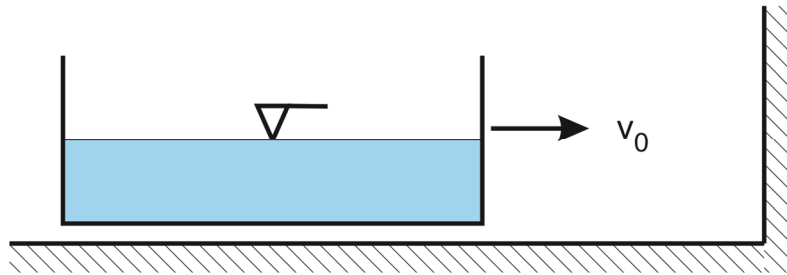


Figure 1: Water filled container colliding with a wall

Concerning the container, a linear elastic behavior is assumed, for which the material properties of aluminium in terms of Young's modulus E and Poisson's ratio ν are given below. The geometric dimensions are length L , width B , height H and wall thickness t . The container is filled with water up to a height of $H/2$. The properties of water are the dynamic viscosity η , speed of sound c_0 and density ρ . For the numerical investigations the parameters are chosen as

$$\begin{array}{llll}
 v_0 = 10e3 \text{ mm/s} & L = 1e3 \text{ mm} & \eta = 1e-9 \text{ Ns/mm}^2 & g = 9.81e3 \text{ mm/s}^2 \\
 E = 70e3 \text{ N/mm}^2 & B = 0.5e3 \text{ mm} & c_0 = 1.45e5 \text{ mm/s} & \\
 \nu = 0.3 & H = 0.5e3 \text{ mm} & \rho = 1e-6 \text{ kg/mm}^3 & \\
 & t = 1 \text{ mm} & &
 \end{array}$$

Note that c_0 is a factor of 10 smaller than the actual physical speed of sound in water, which leads to a higher compressibility of the simulated fluid. However, the density variations in the present example remain in the range of 1%. Therefore, the relevant behavior of water is still closely approximated, while the computational effort is significantly decreased.

In section 4, the simulation models are specified in more detail. The models in HOTINT and Abaqus are differing in several respects because of different implementation of the shell elements of the container, the contact formulations and the SPH formulations. In section 5, the solutions of the two software tools and the two simulation methods are compared. As mentioned above, the goal is to find the advantages and disadvantages of the simulation models and to verify the results.

3 SIMULATION MODELS

In this section three simulation models are compared. Care has been taken to develop coinciding models with respect to physical and numerical parameters. However there are some differences in the implementations in the two software tools Abaqus and HOTINT, as well as in the two simulation methods SPH and CEL in Abaqus. Especially, the contact algorithms are differing in the three investigated formulations. In the following sections the implemented models are described in detail.

3.1 HOTINT/LIGGGHTS – SPH

HOTINT is an open source (<http://www.hotint.org>), multi-purpose simulation software for both research and industrial applications [4]. The core functionality of HOTINT focuses on a detailed and efficient analysis of complex flexible multibody systems. In addition to conventional features of multibody codes, i.e., rigid bodies, simple flexible members and a large variety of kinematic pairs, HOTINT offers several elements for large deformation problems. The latter range from advanced beam, plate and shell elements to classical non-linear solid finite elements. In the present investigations, a four-noded large deformation plate element is utilized for the representation of the solid container. The plate element [5] employs Kirchhoff's kinematic assumption of shear deformation being negligible for thin structures. Concerning the numerical formulation, the element belongs to the family of elements based on the absolute nodal coordinate formulation (ANCF) [6]. The key idea of ANCF is to avoid rotational degrees of freedom in favour of slope vectors, i.e., the two gradient vectors of the mid-surface's position in the present case. Each node consequently has nine degrees of freedom requiring third-order polynomials for the interpolation. Due to the high order of convergence, a relatively small number of elements are required as compared to the linear elements used in Abaqus. Each side of the container is discretized separately first; afterwards, the sidewalls are stitched together by means of kinematic constraints that prohibit the relative translation and rotation at the nodal positions on the edges. A rigid body fixed to the ground represents the obstacle. For the contact between the container and that obstacle, a penalty formulation is utilized in the simulations.

Regarding time integration, a range of implicit high-order Runge-Kutta methods are available in HOTINT. For the present investigations, a third-order Radau IIA scheme with a constant step size is utilized.

The fluid part is not directly included in HOTINT; instead, a co-simulation with the open-source software LIGGGHTS (<http://www.liggghts.com>) is performed for this purpose. LIGGGHTS is a massively parallel solver originally developed for large-scale particle simulations in the field of molecular dynamics, on top of which the SPH functionality and a fluid-structure interface have been implemented [3].

As opposed to the solid structure, the explicit time integration of the SPH formulation typically requires much smaller time-steps. Accounting for that, the two simulators HOTINT and LIGGGHTS are coupled in the sense of a weakly-coupled force-displacement scheme of Jacobi type (see, e.g., [7]). Performing the time-integration, several explicit time-steps are computed for the fluid during a single implicit step on the solid side. After each implicit step, the updated forces of the fluid on the structure and the position and velocity data of the structure's surface (which forms the moving boundary for the fluid) are exchanged mutually by means of a TCP/IP interface. During the fluid substeps, the velocity field of the boundary is assumed constant, and the positions are obtained by explicit integration. Further details on the underlying penalty-based boundary formulation for SPH, the coupling scheme, and the implementation can be found in [3]; for advanced features and applications, the reader is referred to [8].

3.2 Abaqus – SPH and CEL

The container is modelled by linear shell elements of type S4R, the floor and wall by rigid elements of type R3D3. In the SPH-model, the fluid is initially modelled by 3D continuum elements C3D8R and automatically converted to particles by the solver in the first simulation step. This conversion procedure enables a more comfortable way for defining the particles. On the other hand, in the CEL-model the fluid is represented by Eulerian elements of type EC3D8R. For modelling the interactions of the parts, a general contact formulation with soft behavior in normal direction is defined. Thus, contact is considered between fluid and container, as well as container and wall. Due to the differing implementations of SPH and Eulerian elements, there are slight differences in the contact model. Explicit time integration is performed considering large deformations. Further information about the implemented elements and algorithms can be found in Abaqus Manual [9].

4 SIMULATION RESULTS

4.1 Displacements in the Container without Fluid

In the first step, the displacements of the container in the static case are investigated. The container is clamped on the back wall. On the front wall a pressure of $4e-4$ N/mm² is applied. The results of a geometrically nonlinear analysis for the magnitude of the displacements are shown in Figure 2. The maximum displacements in longitudinal (x) and transversal (z) direction obtained by HOTINT and Abaqus are compared in Table 1.

Table 1: Maximum longitudinal and transversal displacements in a static simulation

Model	u_x (mm)	u_z (mm)
HOTINT (8x4x4)	20.51	11.39
HOTINT (16x8x8)	21.15	11.93
Abaqus	21.73	11.38

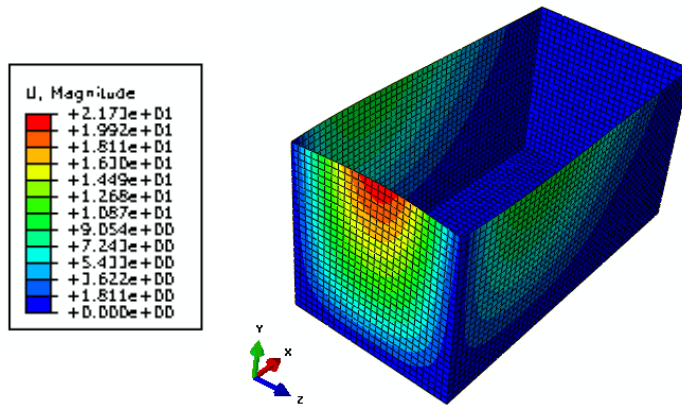


Figure 2: Magnitude of displacements in a static simulation

In the next step, a modal analysis has been performed to obtain the eigenfrequencies of the container. The results obtained by HOTINT (three mesh sizes) and Abaqus (one mesh size) are compared in Table 2, Figure 3 shows the first two symmetrical modes.

Table 2: Eigenfrequencies

Mode Nr.	Frequency (Hz)			Abaqus
	HOTINT (4x2x2)	HOTINT (8x4x4)	HOTINT (16x8x8)	
1	3.370	3.378	3.380	3.373
2	5.388	5.444	5.459	5.454
3	7.680	7.698	7.706	7.696
4	12.233	12.346	12.364	12.356
5	16.075	15.890	15.963	15.962
6	16.599	16.156	16.133	16.129

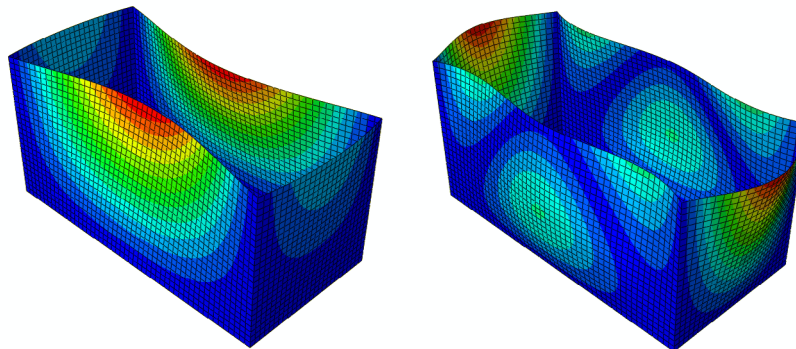


Figure 3: Symmetrical modes (a) mode nr. 2 and (b) mode nr. 5

In general these results show a good coincidence for the two simulation codes. The slight differences are caused by the different element types in the HOTINT and Abaqus model.

4.2 Motion and Pressure of the Fluid

Figures 4 and 5 show the situation 50 ms after the collision of the container and the wall. The figures show the deformed container and the motion of the fluid, by a plane cut through the scene. The contour plot refers to the pressure in the fluid. Good mutual agreement was obtained between the three results for both the displacement and the pressure field in the fluid. The Abaqus SPH-solution for the pressure exhibits much more noise than the SPH-solution of HOTINT/LIGGGHTS.

However, the deformation field in the three solutions is differing significantly at the same time. The mechanical behaviour of the container has been investigated in section 4.1, showing a very good coincidence of the static results and the modal analysis. Therefore, we can assume that the discrepancies of the displacement field of the container in Figures 4 and 5 are caused by the different contact formulations in the simulation models. The specific model of the impact has significant influence on the amplitude and phase of the excited modes of the container.

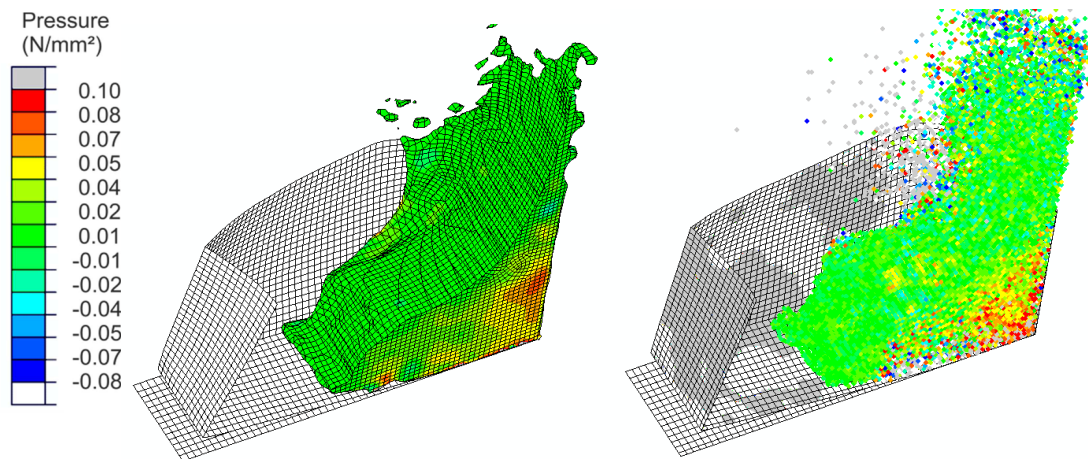


Figure 4: Pressure in the Fluid 50 ms after impact, (a) Abaqus CEL, (b) Abaqus SPH

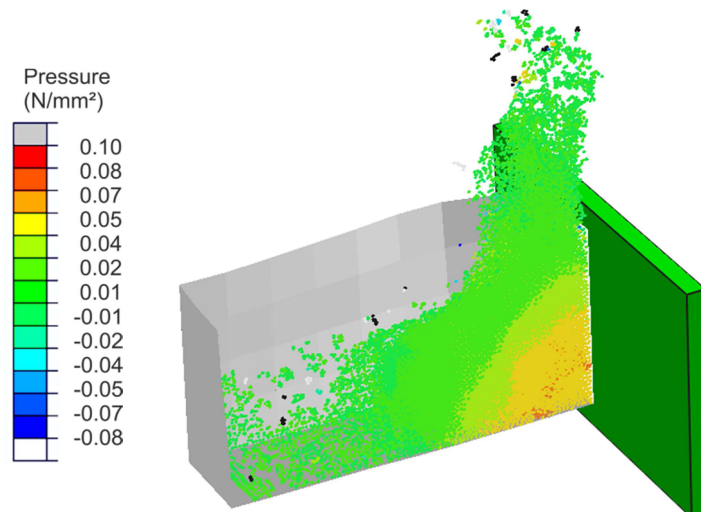


Figure 5: Pressure in the Fluid 50 ms after impact, HOTINT SPH

4.3 Stress in the container

In the next step, the distribution of the von Mises stress in the container is compared. Corresponding to the results in section 4.1, the von Mises stress field is compared for points in time with coinciding deformation of the container. Thus, Figure 6 shows the Abaqus results 38 ms after the impact and Figure 7 the HOTINT results 28 ms after impact. Comparison of the three simulation models shows a good coincidence for the maximum von Mises stress in the container of about 400 N/mm².

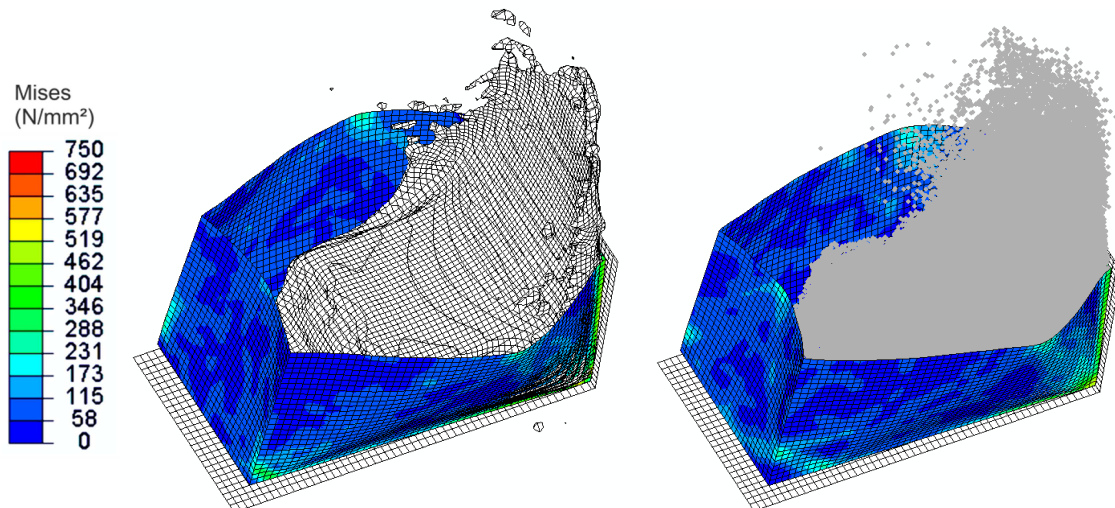


Figure 6: von Mises Stress in the container 38 ms after impact (a) Abaqus CEL (b) Abaqus SPH

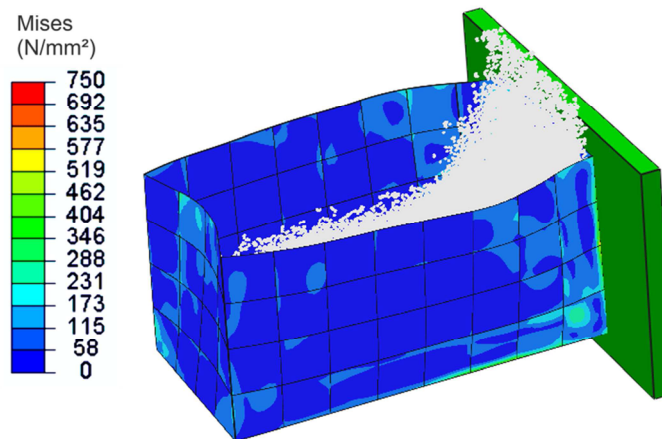


Figure 7: von Mises Stress in the Container 28 ms after impact, HOTINT

4.4 Contact Force

Finally, the resulting contact force between container and wall is compared for the three simulation models in Figure 8, starting 2 ms after the impact. Due to differing contact formulations it is not reasonable to compare the contact force during the impact. This is also the reason for the phase shift a short time after the impact, and thus for the differences in the displacement field in section 4.2. For $t > 40$ ms after impact the contact force shows a very good coincidence.

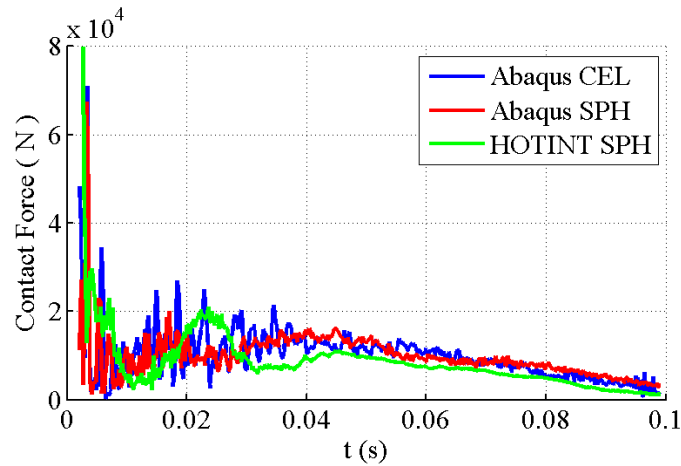


Figure 8: Contact force 2 ms after impact

4.4 Notes on the Computational Efficiency

All three coupled fluid-structure simulations were performed with the same spatial resolution for the fluid domain (in the range of 80000 particles or elements, respectively), and comparable accuracy for the solid parts. As to the latter, the spatial resolution for the plate elements of the container is, as already mention, significantly lower in case of the high-order elements in HOTINT (cf. section 3.1). The different resolutions can be compared Figure 6 and Figure 7.

Regarding time integration, Abaqus exclusively uses an explicit integrator, while the HOTINT/LIGGGHTS approach is based on a coupling between high-order implicit and fast explicit integration (see again section 3.1).

As a reference point, on 4 desktop CPUs, the average wall clock times for the whole impact simulation (i.e., 100 ms simulated physical time) are summarized in Table 3. We observed significantly better performance for the HOTINT/LIGGGHTS approach compared to the Abaqus simulations. Mainly, this is due to the highly efficient SPH implementation within LIGGGHTS. Note that the difference increases with increasing spatial resolution for the fluid domain.

Table 3: Wall clock times T for the three simulations

Simulation	T
Abaqus CEL	~ 3h 30min
Abaqus SPH	~1h 45min
HOTINT/LIGGGHTS – SPH	~40 min

5 CONCLUSIONS

In this paper two simulation methods and tools for modelling fluid-structure interaction have been compared. The results have shown a good coincidence for the displacement and pressure field of the fluid, the maximum von Mises stress in the container and the time response for the contact force. Due to differing contact formulations in the three models, it is not reasonable to compare the results during the impact. These differences also cause a phase shift in the results of the contact force and displacement field of the container. However, comparing the maximum displacements and von Mises stresses in the container for corresponding deformation shapes yields satisfactory agreement.

The presented results show that all three methods are appropriate for simulating fluid-structure interaction. Appropriate results can be expected for the motion of the fluid, the pressure field in the fluid, the contact force between container and wall, as well as the maximum displacements and stresses in the structure.

REFERENCES

- [1] Benson, D.J. Computational Methods in Lagrangian and Eulerian Hydrocodes, *Comput. Methods Appl. Mech. Engrg.* (1992) **99**: 235–394
- [2] Monaghan J.J. Smoothed particle hydrodynamics, *Rep. Prog. Phys.* (2005) **68**: 1703–1759
- [3] Schörgenhumer, M., Gruber, P. and Gerstmayr J. Interaction of flexible multibody systems with fluids analyzed by means of smoothed particle hydrodynamics, *Multibody Syst. Dyn.* (2013) **30**: 53–76.
- [4] Gerstmayr, J. Dorninger, A., Eder, R., Gruber, P., Reischl, D., Saxinger, M., Humer, A., Nachbagauer, K., Pechstein, A.S. and Vetyukov, Y. HOTINT – a script language based framework for the simulation of multibody dynamics systems. *In Proc. ASME 2013 IDETC/CIE*, Portland (OR), USA, 2013.
- [5] Dmitrochenko, O.N., and Pogorelov, D.Yu. Generalization of plate finite elements for absolute nodal coordinate formulation, *Multibody Syst. Dyn.* (2003) **10**: 17–43.
- [6] Gerstmayr, J., Sugiyama, H., Mikkola, A. Review on the Absolute Nodal Coordinate Formulation for Large Deformation Analysis of Multibody Systems. *J. Comput. Nonlinear. Dynam.* (2013) **8** (3): 031016 (12 pages).
- [7] Busch, M. Zur Effizienten Kopplung von Simulationsprogrammen, PhD Thesis, University Kassel, Germany, 2012
- [8] Schörgenhumer, M., Humer, A., and Gerstmayr, J. Efficient fluid-structure interaction based on modally reduced multibody systems and smoothed particle hydrodynamics. *In Proc. WCCM IX*, Barcelona, Spain, 2014.
- [9] Abaqus 6.12 Online Documentation, Dassault Systèmes, 2012