V International Conference on Computational Methods in Marine Engineering
MARINE 2013
B. Brinkmann and P. Wriggers (Eds)

SIMULATION OF A PENDULUM WITH FLUID INTERACTION AND EXPERIMENTAL VALIDATION

Florian Beck, Florian Fleissner and Peter Eberhard

Institute of Engineering and Computational Mechanics
University of Stuttgart
Pfaffenwaldring 9, 70569 Stuttgart, Germany
[florian.beck, florian.fleissner, peter.eberhard]@itm.uni-stuttgart.de
www.itm.uni-stuttgart.de

Key words: Smoothed Particle Hydrodynamics; Multibody System; Coupled Simulation; Experimental Comparison; Pendulum in Water

Abstract. The coupling of different simulation approaches allows the simulation of complex systems. One interesting combination in this context is to simulate rigid bodies moving in a fluid. In this work, the coupled simulation of a rigid pendulum in a water tank is presented. The simulation results for different immersion depths of the pendulum in the fluid are compared with experimental data.

When using a coupled simulation it is possible to merge several advantages of different simulation techniques in one common simulation. This way, several effects that influence the dynamic behavior of complex systems can be investigated. The deceleration of a solid body while moving in a fluid and the free surface of a fluid can be analyzed in one joint simulation. The setup used in this work allows the pendulum to plunge fully or partly into the fluid. The pendulum in a water tank is a simple example for a complex system for which several effects have to be taken into account in order to reproduce the dynamic behavior of the whole system precisely.

In a first step, the mechanical model is set up and simulated and, afterwards, experiments were performed. For the experiments the simulation model is transferred with a scale 1:1. Besides the simulation and its results, the experimental setup for this pendulum and a comparison are presented. The simulation of the pendulum in the water tank shows nice agreement with the experimental data.

1 INTRODUCTION

Many systems in engineering applications are getting more and more complex. The modeling and simulation of such complex systems can be a very challenging task. One typical example for such a complex system is a floating wind turbine. This kind of system consists of several components, which require different simulation techniques when the dynamic behavior of the entire system is investigated. A floating wind turbine is subjected to different load types, e.g. wind and wave loads [1]. Therefore, both aerodynamic and hydrodynamic forces have to be taken into account.

One possible approach when analyzing the structural response of complex systems subjected to different load types are coupled simulations. In a coupled simulation, two different simulation techniques can be combined in order to be able to simulate such complex systems. In this work, one example of such a complex system is presented. It consists of a pendulum with a rigid body, which immerses into a fluid. The setup allows the pendulum to plunge fully or partly into the fluid. There are different phases during one swing of the pendulum. In the initial phase the pendulum is totally outside of the water. In the beginning the pendulum is moving in the air with little resistance. Then, the impact of the pendulum into the water follows and it is decelerated by the damping from the fluid. In this example, there are several points that have to be considered when studying the dynamic behavior. The motion of the rigid body due to water resistance, the description of the free surface and the fluid body interface are investigated. To handle the challenging points when simulating the pendulum, the coupled simulation is divided into two submodels. The Multibody System (MBS) approach is chosen for the submodel for the rigid body pendulum, whereas the Smoothed Particle Hydrodynamics (SPH) method is employed for the submodel of the fluid. The SPH method is a mesh-less method. Some advantages of this method are the description of the free surface and the interface between the two submodels. When using classical grid-based methods the computational effort to simulate the free surface and the interface is much higher, thus, we have chosen a mesh-less method.

The aim of this work is, on the one hand, to introduce a simulation tool for simulating these complex systems and to show the advantages when applying the mesh-less SPH method. On the other hand, it is to present the experimental framework for a pendulum in a water tank. The experimental setup of the pendulum in the tank allows comparing the performed simulations with experimental data. The work is therefore divided into three parts. The theoretical background and simulation part, the description of the experimental setup and, finally, the comparison of the simulation with the experiment. In the theoretical background and simulation part some theory of the applied simulation methods is presented. Then, the coupling technique is introduced before discussing some details of the results of the simulation. Then the description of the experimental setup and the results of the experiments follow. In the last part the comparison of the simulation and the experiment is made.

2 Theoretical Background and Simulation Model

2.1 Smoothed Particle Hydrodynamics

For each submodel of the coupled simulation a different simulation method is used. In Figure 1, a sketch of the pendulum with the water tank is shown. The SPH method is used for the simulation of the fluid. Due to its mesh-less character, the SPH method can handle the free surface naturally. Another advantage is that no special treatment for the fluid solid interfaces is needed. Because of these two advantages, the dynamic behavior of the pendulum due to water resistance can be analyzed in the simulation.

Originally, the SPH method was designed to investigate astrophysical phenomena. But nowadays, there are other applications of the SPH method like fluid dynamics, e.g. for free surfaces [2], [3] or fluid-structure interaction [4], [5] or multiphase flows [6], [7]. An overview about other field of applications can be found in [8].

The SPH method has been implemented in our particle simulation software Pasimodo [9], which is designed for mesh-less particle methods. Due to its modular structure, it is possible to couple Pasimodo with different other simulation software packages [10].

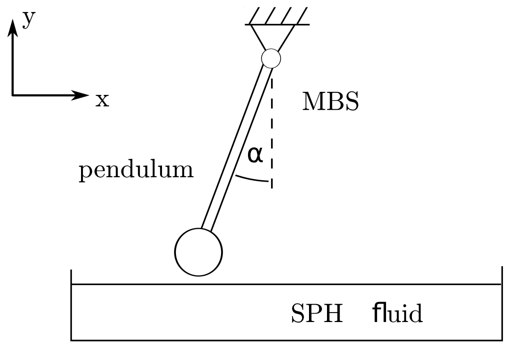


Figure 1: Sketch of the pendulum.

Using the SPH method, the Navier-Stokes (N-S) equations for describing a fluid can be discretized. For a fluid with density ρ, pressure p, viscosity ν, and velocity $\mathbf{v}$ the equation used to describe the conservation of momentum and the equation for the conservation of mass are

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t}+\mathbf{v}\cdot\nabla\mathbf{v}\right)=-\nabla p+\mu\nabla^2\mathbf{v}+\mathbf{f}\text{ and} \qquad (1)$$

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho\mathbf{v})=0\ . \qquad (2)$$

In (1), $\mathbf{f}$ is the vector of the body forces acting on the fluid. To obtain the SPH formulation of the N-S equations basically two steps are required. The first step is a kernel approximation and the second one is referred to as particle approximation [8]. The continuum, which is described using the N-S equations, is discretized into so called particles. At these particles any quantity is described with the function

$$A(\mathbf{r}) = \sum_j m_j \frac{A_j}{\rho_j} W(|\mathbf{r} - \mathbf{r}_j h_j|) \ . \tag{3}$$

Here, the subscript j refers to the particle number, m_j to the mass, $\mathbf{r}$ to the position, ρ_j to the density of a particle, and W is the kernel function. In this work, the classical Gaussian kernel is used for all simulations. For a more detailed description of the method see [8]. The solid body of the pendulum is modeled using the MBS method which is briefly outlined in the next section.

2.2 Multibody System Method

The MBS method can be applied for the analysis of many engineering systems which consists of rigid bodies. It is also possible to take elastic deformations into account. To achieve this, classical Multibody Systems are extended by elastic bodies [11], [12]. Several rigid bodies connected with coupling elements, e.g. springs or dampers, form a classical Multibody System. A detailed description can be found in [13]. The equation of motion can be derived with classical principles of mechanics and yield for the complete MBS

$$\mathbf{M}(\mathbf{q}) \cdot \ddot{\mathbf{q}} + \mathbf{k}(\dot{\mathbf{q}}, \mathbf{q}, t) = \mathbf{g}(\dot{\mathbf{q}}, \mathbf{q}, t) \tag{4}$$

with the mass matrix $\mathbf{M}$, the vector $\mathbf{k}$ for the generalized gyroscopic forces, the vector $\mathbf{g}$ with generalized applied forces, and the vector $\mathbf{q}$ of generalized coordinates.

The MBS simulation is performed with Neweul-M^2 [14]. For the coupling of the two simulation software packages Pasimodo and Neweul-M^2, the possibility of C-export is used. In this way, it is possible to write the complete symbolic description of an MBS model to a shared library. In the next section, the coupling of the MBS and the SPH method is discussed.

2.3 Coupled Simulation

The simulation of complex systems with different dynamic behavior can be realized by dividing the system in several subsystems. This separation into subsystems leads to several advantages, e.g. it is possible to apply for each subsystem another integration scheme. There are various approaches to couple these subsystems [15].

The approach which is applied in this work, is separate modeling and separate simulation. The simulations of the subsystems are coupled, therefore, this is called coupled simulation or modular simulation. The time step size of each subsystem can be different, so, the simulation of the entire system is similar to a multi rate method [16]. In each time step of the simulation the necessary system information, like forces and motions, is transferred between the subsystems. The flow of information can be seen in Figure 2.

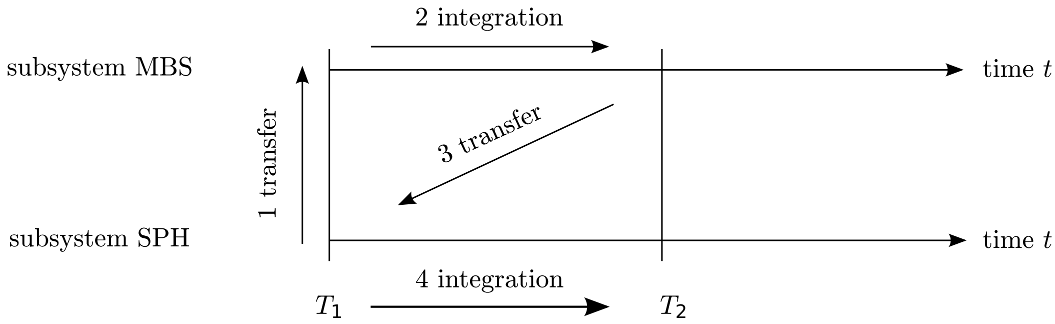


Figure 2: Scheme of the data exchange between the two submodels.

An important point is the stability of the simulation. The information is exchanged at discrete time steps. In [17] is shown for coupled MBS that instability of the modular simulation can occur, if a non-iterative simulator coupling is applied. Therefore, to maintain the stability of the simulation, the exchange scheme is applied in an iterative process.

Another point is the computational effort of the simulation. For the coupling in this work an adaptive exchange time interval is applied. The exchange time interval is then increased or decreased depending on the interaction of the two subsystems. In each simulation step a neighborhood search of the particle simulation is performed. During the neighborhood search the contact detection of the fluid and the solid body takes place. The positions and the velocities are evaluated and the exchange time interval is adapted. The adaptive exchange time interval can be seen in Figure 3, showing also the x-position of the pendulum. When the pendulum immerses into the fluid the exchange time interval is decreased. During the phases when the pendulum is outside the fluid it is increased.

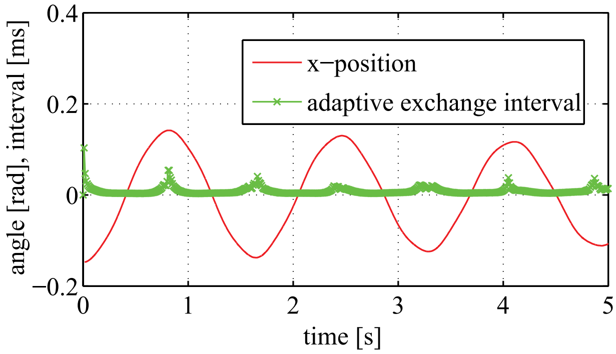


Figure 3: Adaptive exchange time interval.

The calculation of the forces acting between the two subsystems is based on the repulsive force model from [2]. The distance and the difference in the velocities of the solid body and the fluid are determined and the force is calculated. The solid body structure is represented by triangular meshes in Pasimodo. In case that an interaction between the mesh and the particles occurs, the calculated force $\mathbf{f}_c$ is applied to the two subsystems. The equations of motion for the MBS (4) and the momentum equation (1) for the fluid yield

$$\mathbf{M}(\mathbf{q}) \cdot \ddot{\mathbf{q}} + \mathbf{k}(\dot{\mathbf{q}}, \mathbf{q}, t) = \mathbf{g}(\dot{\mathbf{q}}, \mathbf{q}, t) + \mathbf{f}_c \text{ and} \tag{5}$$

$$\rho \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \mu \nabla^2 \mathbf{v} + \mathbf{f} + \mathbf{f}_c \quad . \tag{6}$$

For a fixed immersion depth of the pendulum the parameters are analyzed that take the distance and the difference in velocity into account. Exemplary the x-position for different simulations is shown in Figure 4. On the left side, the influence of the distance parameter is shown and, on the right side, the parameter for the velocity difference. As it can be seen the influence is not significant.

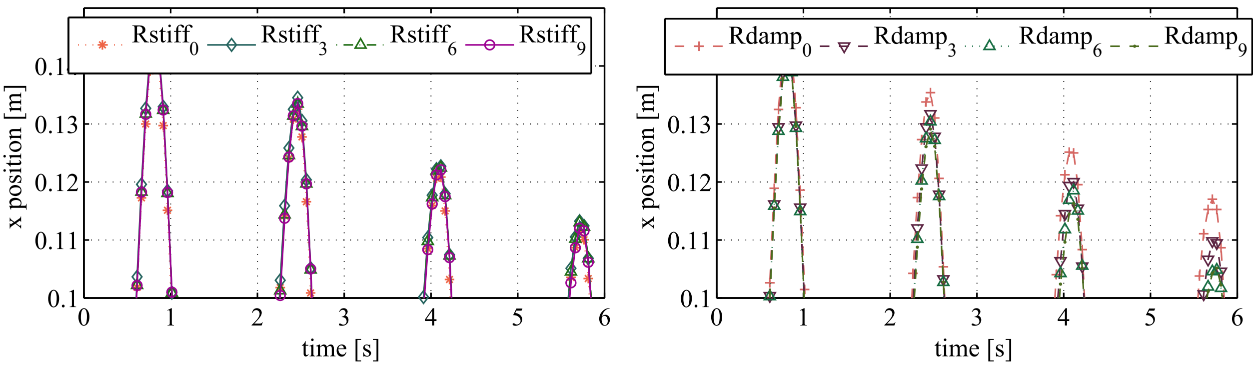


Figure 4: Simulation results when varying the stiffness (left) and the damping (right) parameters.

In the next section, the description of the employed experimental setup as well as the evaluation of the measurement data in order to recreate the trajectory of the pendulum is presented. Later on, the trajectory of the experiment and the simulation are compared.

3 Experiment

3.1 Experimental setup

For the comparison of the simulation and the experiment, measurements are performed in our laboratory. Therefore, the simulation model is transferred in scale 1:1 to the reality. The experimental setup of this work can be seen in Figure 5. It consists of a frame to which the pendulum is attached to using low-friction bearings. The size of the frame is width 0.78 m × height 1.08 m × length 1.18 m. This framework allows the pendulum to plunge fully or partly into the fluid in the tank. The dimensions of the tank are width 0.3 m × height 0.13 m × length 0.4 m. The pendulum itself consists of a rigid metal rod, and a sphere at its tip. The dimension of the rod made of Carbon steel 1.1274 are width 50.0 mm× height 5.0 mm× length 500.0 mm. The diameter of the sphere made of V2A steel is 30 mm. The rigid body, the sphere and the tank are exchangeable, e.g. an elastic pendulum or a larger tank is possible.

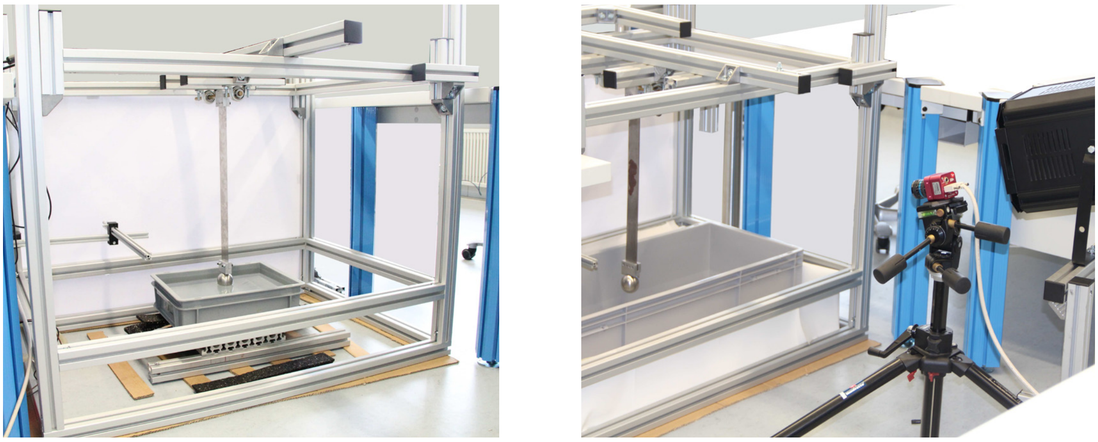


Figure 5: Experimental setup showing a small tank on the left and a larger one on the right side.

3.2 Measurements

In the case of a rigid pendulum, the rotation angle of the pendulum can be determined using an incremental encoder. The encoder is connected to the pendulum with a clutch to compensate a possible shift of the axis. The trajectory can be calculated from this angle. Another possibility for obtaining the trajectory would be to measure the velocity. The used incremental encoder has a maximum resolution of 10000 pulses/round. The maximum sample frequency is 750 kHz. In addition to this highspeed video recordings are made. With these recordings it is possible to track the free surface of the water and compare the experiment with the simulation.

Here, a monochrome camera is used for the highspeed video recordings. Up to 2000 frames per second are possible with this kind of camera, depending on the resolution of the highspeed video recordings. For the lighting of the scene three extremely bright metal-halide spots are used. The highspeed video recordings of the experiments are used for the comparison of the free surface. An example of the highspeed video recording is shown in Figure 6.

4 Results

The results of the coupled simulation of the MBS model and the SPH model are discussed in this section. The simulation is divided into different phases. In one phase, the pendulum is not immersed into the water and there is very little resistance to its motion. After the impact of the pendulum into the fluid, the pendulum is decelerated by the fluid.

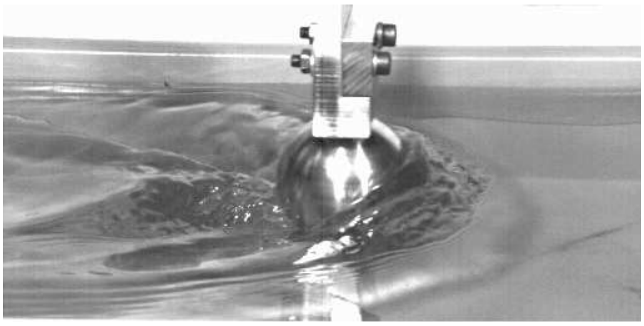


Figure 6: Highspeed video made during the immersion phase.

The whole simulation model is shown for two different times in Figure 7. In Figure 8, the trajectory of three different simulations can be seen. In these simulations the immersion depth of the pendulum is varied, from 1 cm in the lowest point up to 3 cm. The surface of the simulation is shown in Figure 9.

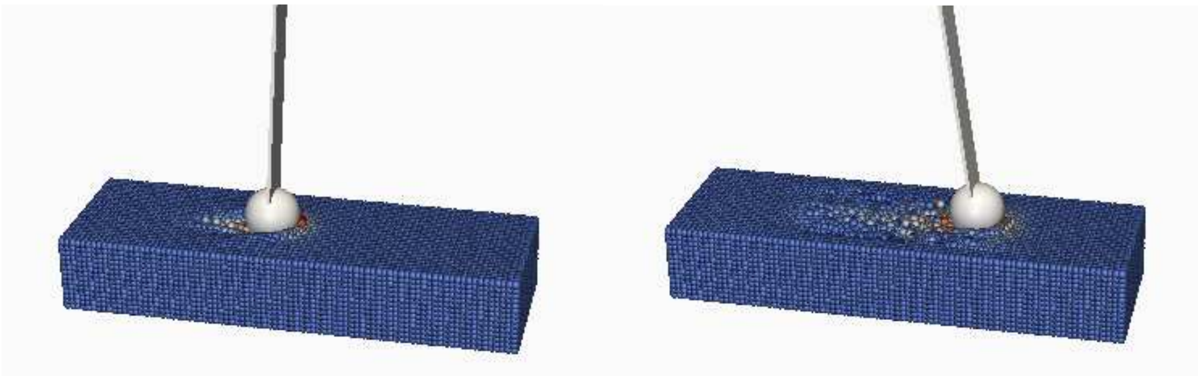
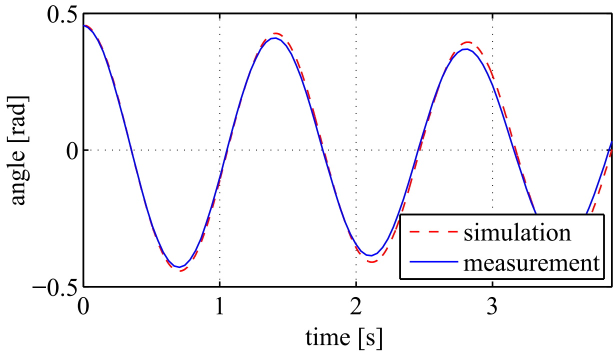


Figure 7: Two different states of the simulation.

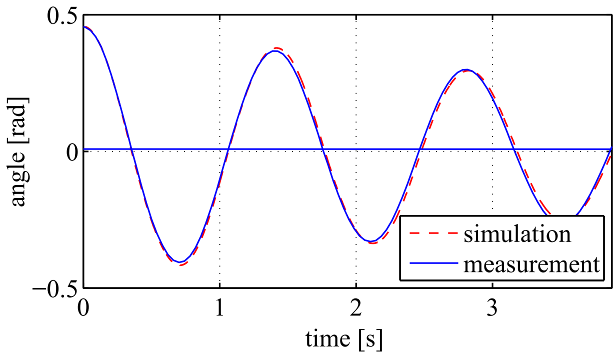
The shape of the free surface is varying during the phase when the pendulum is immersed into the fluid. Using only the pure MBS model it is not possible to get precise results due to the influence of the different shapes of the free surface. But, as it can be seen in the figure, using a coupled simulation gives good results.

5 Conclusion

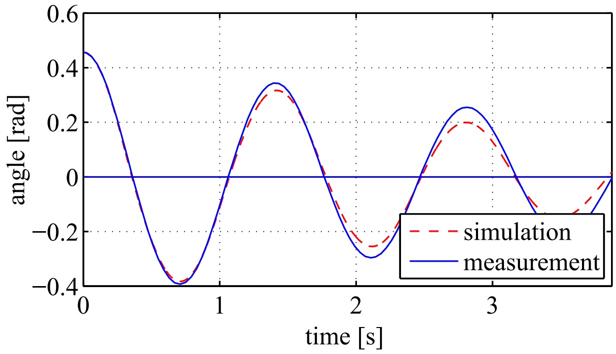
In this work, a simulation framework that allows the coupled simulation of solid bodies and a fluid is presented. The fluid is modeled with the SPH method. The motion of the solid body is modeled using the MBS method. The two subsystems are coupled to an explicit



(a) Immersion depth 1cm

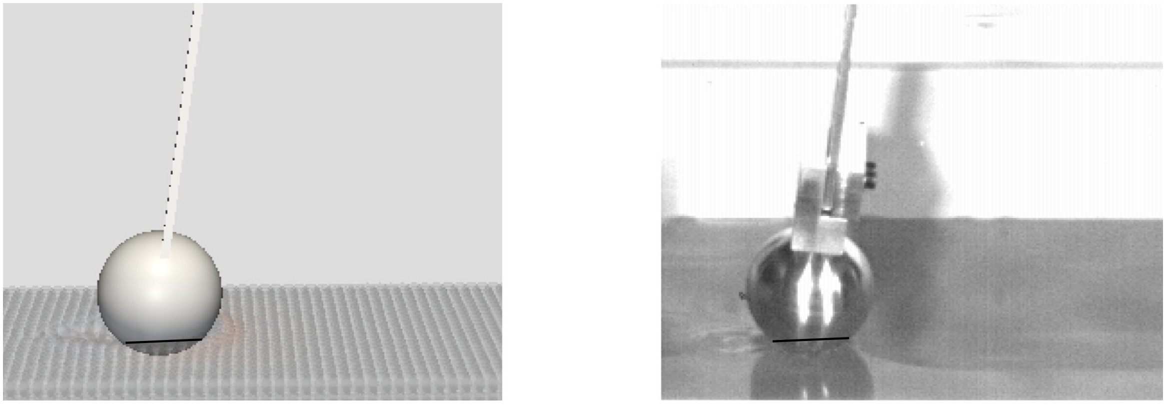


(b) Immersion depth 2 cm

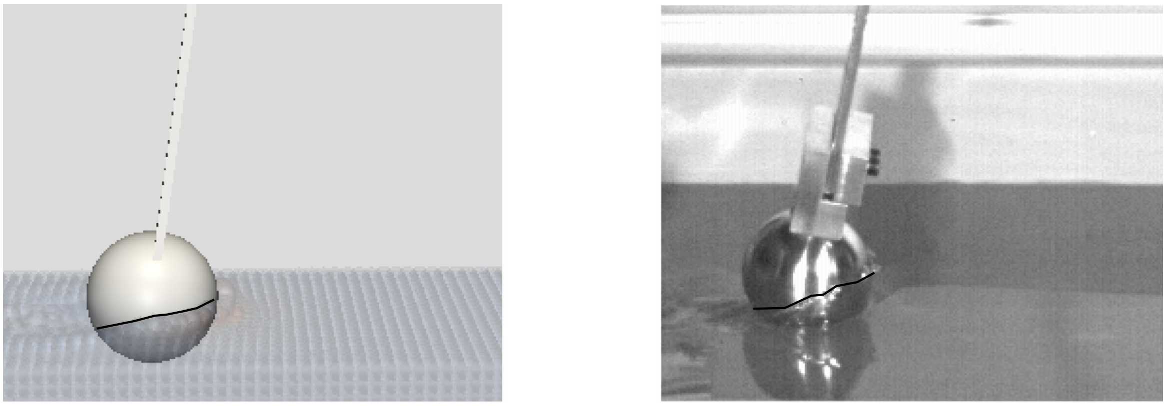


(c) Immersion depth 3 cm

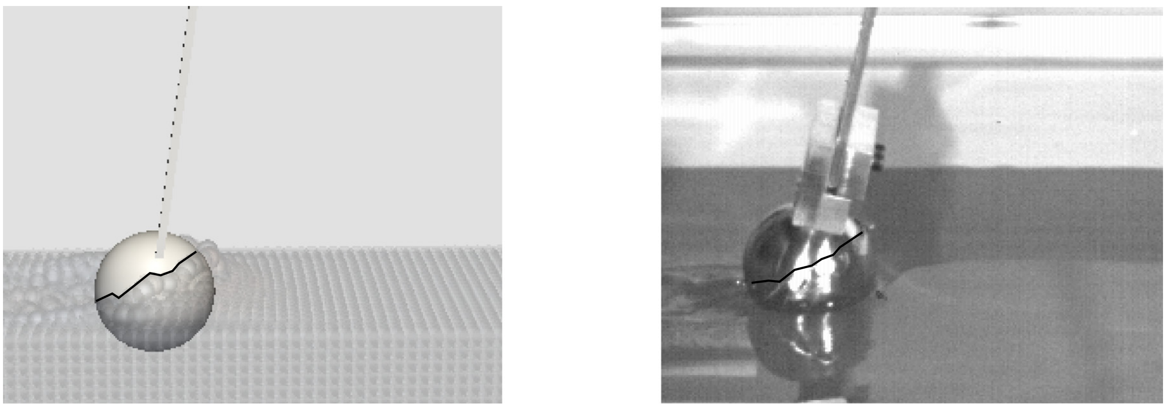
Figure 8: Trajectories of the simulation and the experiment of three different immersion depths.



(a) Immersion depth 1 cm



(b) Immersion depth 2 cm



(c) Immersion depth 3 cm

Figure 9: Simulation and experiment for three different immersion depths. The fluid surface is highlighted with a black line.

modular co-simulation. Furthermore, the simulation model was built in hardware in scale 1:1 and experiments were performed. The motion of the pendulum was recorded with a highspeed video camera. The trajectories and the free surface of the coupled simulation were compared with the experiment and are in good agreement. A most interesting point in the context of the coupled simulation is the possibility to reproduce the free surface of the fluid in a correct way. In this work a rigid body was used for the coupled simulation. It will be a very interesting task to study the dynamic behavior when taking its elastic deformations into account. A further issue will be the use of the so called incompressible SPH method.

REFERENCES

[1] Karimirad M, Meissonnier Q, Gao Z, Moan T. Hydroelastic Code-to-Code Comparison for a Tension Leg Spar-Type Floating Wind Turbine. Marine Structures. 2011;24(4):412–435.

[2] Monaghan JJ. Simulating Free Surface Flows with SPH. Journal of Computational Physics. 1994;110:399–406.

[3] Takeda H, Miyama SM, Sekiya M. Numerical Simulation of Viscous Flow by Smoothed Particle Hydrodynamics. Progress of Theoretical Physics. 1994;92(5):939–960.

[4] Campbell J, Vignejevic R, Libersky L. A Contact Algorithm for Smoothed Particle Hydrodynamics. Computer Methods in Applied Mechanics and Engineering. 2000;184:49–65.

[5] Kulasegaram S, Bonet J, Lewis RW, Profit M. A Variational Formulation Based Contact Algorithm for Rigid Boundaries in Two-Dimensional SPH Applications. Computational Mechanics. 2004;33:316–325.

[6] Colagrossi A, Landrini M. Numerical Simulation of Interfacial Flows by Smoothed Particle Hydrodynamics. Journal of Computational Physics. 2003;191:448–475.

[7] Hu XY, Adams NA. A Multi-Phase SPH Method for Macroscopic and Mesoscopic Flows. Journal of Computational Physics. 2005;213:844–861.

[8] Liu MB, Liu GR. Smoothed Particle Hydrodynamics (SPH): an Overview and Recent Developments. Archives of Computational Methods in Engineering. 2010;17:25–76.

[9] Pasimodo. www.itm.uni-stuttgart.de/research/pasimodo/pasimodo_en.php.

[10] Fleissner F, D'Alessandro V, Schiehlen W, Eberhard P. Sloshing Cargo in Silo Vehicles. Journal of Mechanical Science and Technology. 2009;23(4):968–973.

[11] Schwertassek R, Wallrapp O. Dynamik flexibler Mehrkörpersysteme (in German). Braunschweig: Vieweg; 1999.

[12] Shabana AA. Flexible Multibody Dynamics: Review of Past and Recent Developments. Multibody System Dynamics. 1997;1(2):159–223.

[13] Schiehlen W, Eberhard P. Technische Dynamik (in German). 3rd ed. Wiesbaden: Vieweg+Teubner; 2012.

[14] Kurz T, Eberhard P, Henninger C, Schiehlen W. From Neweul to Neweul-M^2: Symbolical Equations of Motion for Multibody System Analysis and Synthesis. Multibody System Dynamics. 2010;24(1):25–41.

[15] Geimer M, Krüger T, Linsel P. Co-Simulation, gekoppelte Simulation oder Simulatorkopplung? Ein Versuch der Begriffsvereinheitlichung (in German). O + P Zeitschrift für Fluidtechnik. 2006;50(11-12):572–576.

[16] Gear CW, Wells DR. Multirate Linear Multistep Methods. BIT Numerical Mathematics. 1984;24:484–502.

[17] Kübler L, Schiehlen W. Modular Simulation in Multibody System Dynamics. Multibody System Dynamics. 2000;4:107–127.