A PARTITIONED LAGRANGIAN-LAGRANGIAN APPROACH FOR FLUID-SOLID INTERACTION PROBLEMS

Milad Rakhsha\textsuperscript{1}, Arman Pazouki\textsuperscript{2*}, Radu Serban\textsuperscript{1}, Dan Negrut\textsuperscript{1}
\textsuperscript{1}Department of Mechanical Engineering, University of Wisconsin-Madison
Madison, Wisconsin 53706
Emails: \{rakhsha,serban,negrut\}@wisc.edu

\textsuperscript{2}Mechanical Engineering Department, California State University, Los Angeles
Los Angeles, California 90032
Email: apazouk@calstatela.edu

ABSTRACT
We present a Lagrangian-Lagrangian method for solving Fluid-Solid Interaction (FSI) problems in which the solid phase is deformable/compliant. The fluid phase is modeled using Smoothed Particles Hydrodynamics (SPH); the deformable bodies are modeled with the Absolute Nodal Coordinate Formulation (ANCF). Each phase is integrated implicitly in time and the solutions are coupled explicitly by a force-displacement coupling in which the fluid-on-solid effect is modeled via forces applied to the solid phase; and, the solid-on-fluid effect is modeled via fluid boundary conditions. We validate the formulation against two experimental tests: dam brake and elastic gate analysis.

1 INTRODUCTION
Many engineering applications, e.g., fluid sloshing, fording operations, particles in suspension, etc., require the solution of a Fluid-Solid Interaction (FSI) problem. Some of these problems involve large deformations, in which case the fluid-solid coupling remains challenging to solve. In this context, there are two general approaches to tackle the FSI problem: monolithic and partitioned approaches. In the former, the governing equations of the fluid and solid phases are solved simultaneously, whereas in the latter approach the two phases are solved separately and the solutions are coupled together. Each method has its own advantages and disadvantages. What makes the partitioned approach appealing is the fact that it solves each phase with efficient techniques that make the most out of the state of the art in modeling, numerical solvers, and hardware architectures. Among partitioned methods, the Arbitrary Lagrangian-Eulerian (ALE) approach for the fluid and Lagrangian one for the solid are commonly used to treat strongly coupled FSI problems. In the ALE approach, the fluid mesh is deformed to adapt to the solid domain deformation. Although this approach has proven capable of handling large-deformation FSI problems, it is demanding owing to continuous mesh adaptation, a task that is not needed in meshless Lagrangian methods such as SPH. This is the main reason why we embraced SPH; i.e., a Lagrangian method, to solve the dynamics of the fluid phase. In addition, we rely on the ANCF formulation to characterize the time evolution of the solid phase undergoing large deformations.

The SPH-FEM coupling for FSI problems was discussed in [7] and further improved in [3,8,9]. In scenarios with contact and friction, the contact forces were calculated based on an iterative master-slave scheme which finds the best penalty forces required for no-penetration condition. A similar but non-iterative scheme was used in [9]. In [3,10] a dummy-particles scheme was used to compute the pressure from the fluid side. This work follows in the footsteps of this last approach. Regarding the time
integration, each phase is integrated implicitly in time and the fluid–solid sub-systems are coupled explicitly. Specifically, we embrace the implicit incompressible SPH (IISPH) formulation used in [11] and take advantage of boundary conditions introduced in [12]. On the solid-phase side, for which we focus on 1D (cable) and 2D (shell) flexible elements, the Hilber-HughesTaylor (HHT) method is used for implicit time integration of the Multi-Body Dynamics (MBD) system [13]. The contribution is organized as follows. First, we describe the numerical approach used for spatial discretization in the solution of the fluid and solid phases; i.e., SPH and ANCF, respectively. Next, we outline the two-way coupling approach between the fluid and solid phases. A validation study of the methodology is followed by a “demonstration of technology” component. Lastly, we summarize a series of advantages and disadvantages associated with the current method and outline a series of directions of future research.

2 Numerical Approach

Smoothed Particle Hydrodynamics

In relative terms, Smoothed Particle Hydrodynamics has been introduced recently [14, 15]. In SPH, a kernel function \( W \) is defined over a support domain \( S \) to approximate a field function \( f \). The approximation is second-order accurate provided that \( i \) in the limit, as the size of the support domain goes to zero, the kernel function approaches the Dirac delta function, i.e., \( \lim_{h \to 0} W(r, h) = \delta(r) \); \((ii)\) the kernel function is symmetric i.e., \( W(r, h) = W(-r, h) \); and, \((iii)\) the normalization condition holds over the support domain, i.e., \( \int_S W(r, h) = 1 \), where \( h \) is the kernel function’s characteristic length. An example of a 3D cubic spline interpolation kernel \( W \) that satisfies the above conditions is

\[
W(q, h) = \frac{5}{14 \pi h^3} \begin{cases} 
(2-q)^3 - 4(1-q)^3, & 0 < q < 1 \\
(2-q)^3, & 1 < q < 2 \\
0, & q \geq 2 
\end{cases}
\]

where \( q \equiv |r|/h \).

By making use of the identity \( f(r) = \int_-\infty^\infty f(r') \delta(r - r')dV \), and approximating the delta function with a kernel function which meets the above criteria and has a finite support domain of \( \kappa h \) i.e., \( W(r - r', h) = 0 \) when \( |r - r'| > \kappa h \), the SPH approximation of function \( f \) is expressed as

\[
f(r) = \int_V f(r') W(r - r')dV + O(h^2) .
\]

The radius of the support domain, \( \kappa h \), is proportional to the characteristic length \( h \) through the parameter \( \kappa \), the latter commonly set to 2 for the cubic spline kernel. The above integral is further approximated to evaluate \( f \) at the location of an SPH particle \( i \) as

\[
f(r_i) = \int_V f(r_j) W(r - r_j)\rho(r_j)dV + O(h^2) = \sum_{j \in \mathcal{S}(i)} \frac{m_j}{\rho_j} f(r_j) W_{ij} = \langle f \rangle_i ,
\]

where \( \langle f \rangle_i \) indicates “approximation” of \( f \) at the location of SPH particle \( i \). For simplicity, in what follows, \( \nabla_i W_{ij} \) is replaced by \( \nabla W_{ij} \).

**SPH for Fluid Dynamics.** For the fluid phase, the mass and momentum balance, or continuity and Navier-Stokes (NS), equations assume the form [17]

\[
\begin{align*}
\frac{dp}{dt} &= -\rho \nabla \cdot \mathbf{v} \\
\frac{d\mathbf{v}}{dt} &= -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{v} + \mathbf{f},
\end{align*}
\]

where \( \nu \) is the fluid kinematic viscosity, \( \rho \) is the fluid density, and \( \mathbf{v} \) and \( p \) are the flow velocity and pressure, respectively.

In the Incompressible Implicit SPH (IISPH) method [11], the continuity equation, Eq. 2, is discretized in time using forward Euler method, and in space using the SPH discretization method of Eq. 1. Therefore, the discretized form of the continuity equation is written as:

\[
\frac{\rho_i(t + \Delta t) - \rho_i(t)}{\Delta t} = \sum_{j \in \mathcal{S}(i)} m_j \mathbf{v}_{ij}(t + \Delta t) \cdot \nabla W_{ij} ,
\]

where incompressibility is imposed by setting \( \rho_i(t + \Delta t) = \rho_0 \).

The NS equations, Eq. 3, are discretized by making using a Chorin’s two-step projection method [18]: first, the intermediate velocity, \( \mathbf{u}^* \), is obtained with the known values of the current
velocities and the values of the RHS terms of Eq. 3 except the pressure term $\nabla p/\rho$; then, the new velocity is obtained by correcting the intermediate velocities $u_i^*$ with pressure information at the new time step. In IISPH the pressure at the new time step is obtained by solving a Poisson equation with a source term that is related to the zero-density-invariant condition, $dp/\rho dt = 0$. More specifically, the RHS of the momentum equations is decomposed into the pressure and the non-pressure contribution to the force acting at the location of marker $i$ as:

$$m_i \frac{v_i(t + \Delta t) - v_i(t)}{\Delta t} = \mathbf{f}_i^p(t) + \mathbf{f}_i^{np}(t + \Delta t),$$

(5)

where $\mathbf{f}_i^p$ and $\mathbf{f}_i^{np}$ represent the pressure forces and non-pressure forces acting on marker $i$, respectively. By manipulating the above equation it will emerge that:

$$v_{i,l+1} = \left( v_{i,l} + \frac{\mathbf{f}_i^{np} \Delta t}{m_i} \right) + \frac{\mathbf{f}_i^{p} \Delta t}{m_i} = v_{i,l} + \mathbf{f}_i^{np} + \mathbf{f}_i^{p} \Delta t, \quad l = 1, 2, \ldots,$$

(6)

where $v_{i,l}^{np} = v_{i,l} + \mathbf{f}_i^{np} / m_i \Delta t$ is the intermediate velocity, $v_{i,l+1}^{p} = \mathbf{f}_i^{p} / m_i \Delta t$ is the velocity correction, and subscripts $l$ and $l + 1$ are selected to indicate the discretized values at time $t_l$ and $t_{l+1}$, respectively.

In contrast to the original Chorin’s projection method and other ISPH flavors, which obtain the pressure equation by imposing the divergence-free velocity condition, $\nabla \cdot v = 0$, in IISPH the pressure equation is obtained by imposing $dp/\rho dt = 0$. The choice of density-invariance condition leads to a more uniform particle distribution and lower density error, although better accuracy in pressure distribution is obtained if one uses the divergence-free velocity condition to create the velocity-pressure coupling [19]. Hence, in Eq. 4, density is also divided into the pressure and non-pressure parts. Subsequently, the intermediate density resulting from the intermediate velocity $v_{i,l}^{np}$ is expressed as:

$$\rho_{i,l}^{np} = \rho_{i,l} + \Delta t \sum_{j \in S(i)} m_j v_{i,j,l}^{np} \nabla W_{ij},$$

(7)

By subtracting the above equation from Eq. 4 it can be concluded that:

$$\rho_{i,l+1} - \rho_{i,l}^{np} = \Delta t \sum_{j \in S(i)} m_j (v_{i,j,l+1} - v_{i,j,l}^{np}) \nabla W_{ij},$$

(8)

where $\rho_{i,l} = \sum_j m_j W_{ij}$. In order to preserve incompressibility, $\rho_{i,l+1} = \rho_0$ must be satisfied within the margin of numerical accuracy. Moreover, using Eq. 6 and the definition of $v_{i,j,l+1}^{p}$, the above equation is further simplified to the following form, which implicitly gives the pressure equation:

$$\rho_0 - \rho_{i,l}^{np} = \Delta t \sum_{j \in S(i)} m_j v_{i,j,l+1}^{p} \nabla W_{ij} = \Delta t \sum_{j \in S(i)} m_j \left( \frac{\mathbf{f}_i^{p}}{m_j} \right) \nabla W_{ij} = \Delta t^2 \sum_{j \in S(i)} m_j \left( \frac{\mathbf{f}_i^{p} \Delta t}{m_j} \right) \nabla W_{ij},$$

(9)

where pressure forces may be obtained from

$$\mathbf{f}_i^{p} \Delta t = -m_i \sum_{j \in S(i)} m_j \left( \frac{p_{i,l+1}^{l} + p_{j,l+1}^{l}}{\rho_{i,l}^{2} + \rho_{j,l}^{2}} \right) \nabla W_{ij},$$

(10)

and the non-pressure forces from

$$\mathbf{f}_i^{np} = m_i \sum_{j \in S(i)} m_j \Pi_{ij} + \mathbf{f}_b.$$ (11)

Above, $\mathbf{f}_b$ represents the body forces and $\Pi_{ij} = -((\mu_i + \mu_j) x_{ij} \nabla W_{ij}) v_{i,j,l}$, in which $h_{ij} = (h_i + h_j)/2$ is defined based on kernel function’s characteristic length, and $\overline{\rho}_{ij} = (\rho_i + \rho_j)/2$. Finally, by substituting Eqs. 7 and 10 into 9, a linear system of equations, with pressure as unknown, is obtained. A new set of velocities is obtain by correcting the intermediate velocity, $v_{i,l}^{np}$, using Eq. 6. Position of particles can also be obtained using $\mathbf{x}(t + \Delta t) = \mathbf{x}(t) + v_{i,l+1} \Delta t$. More detailed information about the pressure equation and its solution was explained in [20].

**Flexible Body Dynamics**

Herein, the flexible body dynamics solution draws on the Absolute Nodal Coordinate Formulation (ANCF), a nonlinear finite element method that handles large deformations of moving bodies [21]. The salient feature of ANCF is its use of position vector gradients to quantify deformation. ANCF uses the nodal global position and nodal position vector gradient vectors to describe the dynamics of flexible bodies that undergo large deformations. In general, the position field of $i^{th}$ ANCF element may be defined as:

$$\mathbf{r}^i(\xi, \eta, \zeta, t) = \mathbf{S}(\xi, \eta, \zeta) \times \mathbf{q}^i(t),$$

(12)

Position of an arbitrary point within the element | Space-dependent shape function | Time-dependent vector of nodal degrees of freedom
which at time $t$ decouples the position of any point $(\xi, \eta, \zeta) \in [-1, 1]$ inside the element into space-dependent and time-dependent components, $S(\xi, \eta, \zeta)$ and $\dot{q}(t)$, respectively. The velocity of any point within an element $i$ may be written as

\[
\dot{r}'(\xi, \eta, \zeta, \xi', \eta', \zeta', t) = S(\xi, \eta, \zeta) \times \dot{q}(t),
\]

Velocity of an arbitrary point within the element

Space-dependent shape function

Time-dependent vector of generalized velocities

The kinetic energy of a finite element $i$ can be obtained as

\[
T = \frac{1}{2} \int \rho \dot{r}' \dot{r}' \, dV = \frac{1}{2} \dot{q}'^T M \dot{q}',
\]

where the mass matrix is defined as $M = \int \rho A S^T S \, dV$. Since the unknowns are relative to a global reference fame, the mass matrix is time-independent. The equations of motion assume then form [22]

\[
M \ddot{q} + \dot{q}' = \dot{Q}',
\]

where $\dot{Q}'$ and $\ddot{q}'$ are the generalized element elastic and applied forces, respectively. The description of these elements and calculation of the internal forces are described next.

**1D Elements – ANCF Cable Element.** The gradient-deficient ANCF cable element introduced in [23] is used in the present work to simulate 1D flexible bodies. As shown in Fig. 1, the coordinates of this element at each node are a position vector and a position vector gradient along the beam center axis. The position gradient vectors normal to the cable axis are not defined, hence the element is gradient deficient. For instance, torsion and shear deformation cannot be captured with this set of degrees of freedom. The coordinates (nodal degree of freedom) of the $j^{th}$ node is expressed as the $6 \times 1$ matrix $q^j(t) = [r^j^T \ r^j^T \ r^j^T]^T$. The position of any point inside the $i^{th}$ element may be interpolated from the nodal degrees of freedom of its nodes as follows

\[
r^i = [s_1 I \ s_2 I \ s_3 I \ s_4 I] [q^1 \ q^2]^T = S(\xi) \dot{q}',
\]

where $I$ is the $3 \times 3$ identity matrix, $S(\xi)$ is a $3 \times 12$ matrix, $q^i$, are $q^1$ are the nodal coordinates of the two nodes forming element $i$, as defined before, and finally $\dot{q}'$ is a $12 \times 1$ matrix combining the nodal coordinates of the element $i$. The interpolation functions are defined as

\[
\begin{align*}
s_1 &= 1 - 2\xi^2 + 2\xi^3, \\
s_2 &= l (\xi - 2\xi^2 + \xi^3), \\
s_3 &= 3\xi^2 - 2\xi^3, \\
s_4 &= l (-\xi^2 + \xi^3),
\end{align*}
\]

where $0 < \xi < 1$ is the non-dimensional parameter defined over the natural coordinates of the element locates a point along the cable centerline ($\xi = 0$ at the first node, and $\xi = 1$ at the second node), and $l$ is the length of the element.

Knowing the axial and bending strains, one can define the internal loads of this element. The generalized element elastic forces are calculated as follows:

\[
Q^a = \int_L \left[ EA \dot{\epsilon}_x \left( \frac{\partial \dot{\epsilon}_x}{\partial q} \right)^T + EI \kappa \left( \frac{\partial \kappa}{\partial q} \right)^T \right] \, dx,
\]

where $E$, $A$, and $I$ are the modulus of elasticity, the cross section area, and the area moment of inertia, respectively. The axial strain and curvature are defined as follows

\[
\epsilon_x = \frac{1}{2} (r^i_x \ r^i_x - 1) \quad \text{and} \quad \kappa = \frac{|r^i_x \times r^i_{xx}|}{|r^i_x|^3},
\]

where $r_{xx} = \partial^2 r / \partial x^2$.  

\[
FIGURE 1. \ ANCF cable element’s schematic. Each node features a global position vector and a position vector gradient along the axis of the element (6DOF). Using shape functions and knowing $\xi$ one can interpolate the degrees of freedom to any point $P$ within the element.
\]
2D Elements – ANCF Shell Elements  The gradient-deficient ANCF shell element studied in [24] was used in the present work to simulate 2D flexible bodies. The nodal global position vector ($\mathbf{r}^i$) and global position vector transverse gradient ($\mathbf{r}^i_z = \frac{\partial \mathbf{r}^i}{\partial \zeta^i} (\xi^i, \eta^i)$) are chosen as the nodal degrees of freedom, as shown in Fig. 2.

![ANCF shell element's schematic](image)

**FIGURE 2.** ANCF shell element’s schematic. Global position vector $\mathbf{r}^i$ and fiber’s direction $\mathbf{r}^i_z = \frac{\partial \mathbf{r}^i}{\partial \zeta^i} (\xi^i, \eta^i)$ are the nodal coordinates of the $i^{th}$ node (6DOF). Using shape functions and knowing $\xi$ and $\eta$ one can interpolate the degrees of freedom to any point within the element.

The positions and gradients on the mid-plane for any point inside the $i^{th}$ element can be interpolated from the positions and gradients of its nodes as follows

$$\mathbf{r}^i_m(\xi^i, \eta^i) = S^i_m(\xi^i, \eta^i)\mathbf{e}^i_p,$$

$$\frac{\partial \mathbf{r}^i}{\partial \zeta^i}(\xi^i, \eta^i) = S^i_m(\xi^i, \eta^i)\mathbf{e}^i_e,$$  \hspace{1cm} (16)

where $\xi^i$ and $\eta^i$ refer to $i^{th}$ element’s natural coordinates in the parametric space, $S^i_m = [S^i_1 \ I \ S^i_2 \ I \ S^i_3 \ I \ S^i_4 \ I]$ is a bilinear shape function matrix, $\mathbf{e}^i_p = \mathbf{r}^i$ is the position vector of $i^{th}$ node of the $i^{th}$ element $i$, and $\mathbf{e}^i_e = \frac{\partial \mathbf{r}^i}{\partial \zeta^i}$ is the position vector gradient of node $j$ of element $i$, and $\mathbf{I}$ is the $3 \times 3$ identity matrix. The bilinear shape functions of the ANCF shell element are given by the following expressions

$$S^i_1 = \frac{1}{4} (1 - \xi^i)(1 - \eta^i), \quad S^i_2 = \frac{1}{4} (1 + \xi^i)(1 - \eta^i),$$

$$S^i_3 = \frac{1}{4} (1 + \xi^i)(1 + \eta^i), \quad S^i_4 = \frac{1}{4} (1 - \xi^i)(1 + \eta^i).$$ \hspace{1cm} (17)

The position of an arbitrary point in the $i^{th}$ element may be described as

$$\mathbf{r}^i(\xi^i, \eta^i, \zeta^i) = S^i(\xi^i, \eta^i, \zeta^i)\mathbf{e}^i,$$ \hspace{1cm} (18)

where $S^i = [S^i_m \ z^i S^i_m]_{3 \times 24}$ is the combined shape function matrix, and $\mathbf{e}^i = [(\mathbf{e}^i_p)^T (\mathbf{e}^i_e)^T]_{1 \times 24}$ is the coordinates of the $i^{th}$ element grouped together. Eq. 18 allows for interpolating points along the element thickness by incorporating the element natural coordinate $\zeta^i$.

The Green-Lagrange strain tensor, which is expressed as follows,

$$\mathbf{E}^i = \frac{1}{2} \left( (\mathbf{F})^T \mathbf{F} - I \right),$$ \hspace{1cm} (19)

is used to obtain the strains, where $\mathbf{F}^i$ is the deformation gradient matrix defined as the Jacobian of the current configuration over the reference configuration, which is expressed as

$$\mathbf{F}^i = \frac{\partial \mathbf{r}^i}{\partial \mathbf{X}^i} = \frac{\partial \mathbf{r}^i}{\partial \mathbf{x}^i} (\frac{\partial \mathbf{X}^i}{\partial \mathbf{x}^i})^{-1}. \hspace{1cm} (20)$$

The strain tensor can then be expressed in vector form as follows

$$\mathbf{e}^i = [e^i_{xx} \ e^i_{xy} \ e^i_{xz} \ e^i_{yy} \ e^i_{yz} \ e^i_{zz}]^T \hspace{1cm} (21)$$

where $\mathbf{e}^i$ is the engineering strain vector in the deformed configuration. The elastic internal forces are obtained by integration over the element volume using Gaussian quadrature as follows

$$\mathbf{Q}^i = \int_{V_0} \left( \frac{\partial \mathbf{e}^i}{\partial \mathbf{e}^i} \right)^T \mathbf{\sigma}^i dV_0^i \hspace{1cm} (22)$$

where $\mathbf{\sigma}^i$ is the vector of the second Piola-Kirchhoff stresses and $dV_0^i$ is the infinitesimal volume at the reference configuration of the element $i$.

The bilinear quadrilateral ANCF shell elements suffer from the the in-plane shear/normal and transverse shear lockings, which can be eliminated using the ANS approach. Moreover, the use of position vector transverse gradient in this element causes thickness locking, which can be alleviated using the EAS approach. More detailed information about comparison of this element with non-ANCF elements, as well as ANS and EAS corrections have been discussed in [24].

The Fluid-Solid Coupling

The fluid-solid coupling is obtained by making use of ghost Boundary Condition Enforcing (BCE) SPH particles that are placed on and close to the flexible bodies. This idea is similar
Experimental Results

Copyright © 2017 ASME

...to what was discussed in [3, 10, 25]. More specifically, the position, velocity and acceleration of each BCE marker is dictated by the flexible body to which it belongs. In other words, these BCE particles are attached to the solid. The shape functions of Eqs. 14 and 17 are used to interpolate the position, velocity, and acceleration of each BCE particle based on nodal values of the corresponding ANCF element. The velocity and the acceleration of the BCE markers factor in the solution of the fluid phase equations of motion when imposing the no-slip and no-penetration boundary conditions. Herein, the velocity \( \mathbf{v}_a \) of an SPH BCE marker \( a \) is calculated based on [12]

\[
\mathbf{v}_a = 2\mathbf{v}_a^p - \bar{\mathbf{v}}_a, \tag{23}
\]

where \( \mathbf{v}_a^p \) is the prescribed wall velocity obtained from the interpolation of nodal velocities for each FE, and \( \bar{\mathbf{v}}_a \) is an extrapolation of the smoothed velocity field of the fluid phase to the BCE markers

\[
\bar{\mathbf{v}}_a = \frac{\sum_{b \in \mathbf{F}} \mathbf{v}_b W_{ab}}{\sum_{b \in \mathbf{F}} W_{ab}}.
\]

Above, \( \mathbf{F} \) denotes a set of “fluid” markers that are within the compact support of the BCE marker \( a \). The pressure of a BCE marker is calculated via a force balance condition at the wall interface, which leads to [12]

\[
p_a = \frac{\sum_{b \in \mathbf{F}} p_b W_{ab} + (\mathbf{g} - \mathbf{a}_w) \cdot \sum_{b \in \mathbf{F}} p_b \mathbf{r}_{ab} W_{ab}}{\sum_{b \in \mathbf{F}} W_{ab}}, \tag{24}
\]

where \( \mathbf{g} \) is the gravity and \( \mathbf{a}_w \) is the prescribed acceleration of the solid phase at the location of SPH BCE particle \( a \), location obtained by interpolating from the nodal accelerations.

The overall algorithm can be summarized as follows: (i) the fluid phase is integrated in time; (ii) the pressure and viscous forces exerted by the fluid markers onto the BCE markers are obtained via Eqs. 10 and 11; (iii) the viscous and pressure generalized forces are calculated from BCE markers and applied to the nodal coordinates; (iv) knowing the forces impressed by the fluid phase onto the solid phase, the solid phase is integrated in time; (v) the new positions, velocities, and acceleration of the BCE markers are interpolated from the new nodal values; and, (vi) the velocities and pressure of the BCE markers required for the no-slip and no-penetration conditions are obtained respectively from Eq. 23 and 24.

---

3 Numerical Experiments

We discuss below two validation studies. The dam break simulation is a widely used benchmark test for validating the SPH solution. The second test, an elastic gate simulation, is used to validate the fluid-solid coupling. Both tests are run using an open source simulation engine called Chrono [26, 27].

**Dam Break Validation**

This test represents a 2D simulation of a dam break. The dam’s height is 1 m and the width is 1 m. The fluid is modeled as inviscid using 16000 fluid particles. The reference density is \( \rho = 1 \text{ kg/m}^3 \), and the SPH particles are placed on a uniform Cartesian lattice at \( t = 0 \) s. The gravity \( g = 1 \text{ m/s}^2 \) is applied in the negative y direction as discussed in [12]. The results for the water-front propagation are shown in Fig. 3.

At early stages the numerical result agrees well with the experiment, yet it gradually over-predicts the experimental results after \( t(Hg)_{1/2} \approx 0.5 \). We attribute this discrepancy to the inviscid flow assumption, which becomes inaccurate as the flow speed (\( \propto \) tangent line in Fig. 3) increases and the effect of the viscosity term, \( \nabla \cdot \mathbf{v} \), in the Navier-Stokes equations becomes significant. Moreover, smaller front propagation speeds are expected for viscous fluids. Thus, it is expected that the numerical results over-estimate the experimental results.

**Elastic Gate in Flow Validation**

Elastic gate simulation is a test used to validate the coupling between the fluid and solid phases. In this experiment, water is encapsulated in a cubic container. The container consists of...
Water properties:
\[ \rho = 1000 \text{ kg/m}^3, \mu = 0.0001 \text{ Pas} \]
Elastic gate properties:
\[ \rho_s = 1100 \text{ kg/m}^3, \nu = 0.4, \text{ thickness} = 0.005 \text{ m} \]

The discrepancy between the numerical results and the experimental results may be attributed to the linear constitutive model used in the present work and previous numerical studies [4, 30]. The rubber gate used in the experiment shows a non-linear behavior when loaded and its stiffness decreases when the strain increases. Hence, by applying a constant stiffness value of the unloaded plate, the numerical result will under-predicts the actual deformation of the gate as shown in Fig. 5.

4 Conclusion
We presented a partitioned method for solving the fluid-solid interaction problem involving flexible bodies. Implicit time integration is used to advance the state of each sub-system. The use of implicit time integrators allowed for one order of magnitude larger time steps, which came at the cost of solving a linear system of equations for each phase at each time step. The method works well in capturing the fluid-solid interaction even when the coupling between them is strong. However, the disadvantage of this method is that the use of ghost markers on “thin” shells and cables requires a very small kernel length, which will ultimately increase the computational costs of the fluid subsystem. One possible approach to addressing this limitation is the use of variable resolution SPH methods, a task that remains to be addressed in the future.

REFERENCES


Copyright © 2017 ASME