Applied Ocean Research 76 (2018) 88–97

Contents lists available at ScienceDirect

Applied Ocean Research

journal homepage: www.elsevier.com/locate/apor

Extending DualSPHysics with a Differential Variational Inequality: modeling fluid-mechanism interaction

R.B. Canelas[a,*], M. Brito[a], O.G. Feal[b], J.M. Domínguez[b], A.J.C. Crespo[b]

[a] CERIS, Instituto Superior Técnico, ULisboa, Lisbon, Portugal
[b] EPHYSLAB, Universidade de Vigo, Ourense, Spain

ARTICLE INFO

Keywords:
SPH
DVI
Non-smooth dynamics
Multibody dynamics
Fluid–structure–structure interaction
Meshless

ABSTRACT

This work details the coupling of a Smoothed Particle Hydrodynamics (SPH) fluid solver with a general-purpose Differential Variational Inequality (DVI) based non-smooth multibody dynamics solver, allowing for efficient and accurate modeling of fluid-mechanism interactions, an ubiquitous scenario in natural and industrial settings. The SPH fluid model (DualSPHysics) can deal with flow non-linearities, free-surface and intense topological changes, while the non-smooth dynamics model (Project Chrono) deals with discontinuous frictional contacts and kinematic restrictions. An open-source integrated framework to model fluid–structure–structure coupled systems is presented by implementing Project Chrono under DualSPHysics.

The model is validated with fluid–structure–structure interaction cases. Both frictional and multi-restriction based behaviors are tested and simple convergence analysis are presented, showing that the model is capable of reproducing complex interactions. Several hypothetical cases are then presented, in order to demonstrate possible applications, showcasing a wide set of options useful for practitioners requiring the use of advanced fluid-mechanism models.

1. Introduction

Devices composed of rigid bodies interacting through frictional contacts and several nonlinear constraints are extensively used in many engineering fields, either featuring a small number of unilateral contacts or including thousands of contacts between a large number of parts. Mechanisms involving contacts and impacts between parts can be modeled in terms of multi-body systems with unilateral constraints. The simulation of rigid contacts entails the solution of non-smooth equations of motion: the dynamics are non-smooth since the non-interpenetration, collision, and adhesion constraints are discontinuous [1]. The interaction of these types of mechanisms with fluid flow is widely seen in fields such as offshore engineering, fabrication processes, coastal protection and renewable energy production.

Smoothed Particle Hydrodynamics (SPH) is becoming a mature tool regarding environmental free-surface flows. It treats unsteady and nonlinear features, extreme deformations and complex topological evolutions, such as a breaking free-surface, implicitly and with sufficient accuracy to provide meaningful solutions to engineering problems. Considerable advantages when computing interactions between objects or structures and a flow [2] are also met. High-performance computing advances have allowed the method to cover applications once reserved to specialized models, opening new possibilities in modeling even further complex phenomena. Using the same developments in computing and the introduction of accessible parallel computing solutions, very efficient solutions are found for non-smooth multi-body systems. Considering the success of SPH for fluid descriptions and non-smooth multibody solvers for mechanical systems, attempting to couple both under a generalized framework should provide new simulation possibilities, by leveraging the strengths in both methods.

In this work the DualSPHysics code [3] is augmented with the Project Chrono library [4], developed as a general-purpose simulation framework for multi-body problems with support for very large systems. The library is implemented under the DualSPHysics code, providing an integrated interface to define and run arbitrarily defined fluid–structure–structure coupled systems. Our implementation allows for the straightforward definition of constraints such as joints (spherical, hinged and full restriction) and sliders (along an axis), combinations of these (hinged slider for example) with arbitrary degrees of freedom, i.e., such restrictions can be set between two bodies that are otherwise unrestricted. The main contribution however is the efficient treatment of such kinematic restrictions with user defined dynamic

* Corresponding author.
E-mail addresses: ricardo.canelas@tecnico.ulisboa.pt (R.B. Canelas), moises.brito@tecnico.ulisboa.pt (M. Brito), orlando@uvigo.es (O.G. Feal), jmdominguez@uvigo.es (J.M. Domínguez), alexbexe@uvigo.es (A.J.C. Crespo).

https://doi.org/10.1016/j.apor.2018.04.015
Received 21 December 2017; Received in revised form 27 March 2018; Accepted 27 April 2018

properties such as friction and restitution coefficients, restitution forces from spring and damper systems and user-imposed forces and trajectories.

The aim of this paper is to explore the DualSPHysics implementation of Project Chrono for modeling of interactions between fluid and rigid bodies systems, with arbitrary mechanical restrictions applied. The fluid implementation on DualSPHysics represents the current state-of-the-art in balancing computational efficiency and numerical accuracy, while maintaining the necessary degree of generality for users and researchers. More accurate particle approximation schemes have been introduced such as Incompressible SPH [5] and CRKSPH [6] among various others. Applicability to large and complex problems is limited however, hence they are not considered currently. The work presented is agnostic to the fluid discretization method, as well as the particulars of the fluid–solid coupling.

In Section 2 the conceptual and numerical models used for the fluid description are reviewed, mapping the equation systems underlying the DualSPHysics implementation of SPH. Section 3 introduces the concepts for the non-smooth multi-body dynamics model and the Differential Variational Inequality (DVI) equation system. Section 4 details the validation cases of the fluid-mechanism solver, using three reference experimental results. Following the validation cases, Section 5 showcases the potential of the model via a selection of cases were non-linear flows drive and interact with complex mechanisms. Conclusions are drawn in Section 6, by discussing the validation results, the usability and attractiveness of the model from a practitioner standpoint and the future developments.

2. Smooth-Particle-Hydrodynamics (SPH)

In SPH, the fluid domain is represented by a set of nodal points where physical quantities such as position, velocity, density and pressure are approximated at. These points move with the fluid in a Lagrangian manner and their properties change with time due to the interactions with neighboring nodes. The term Smoothed Particle Hydrodynamics arises from the fact that the nodes, for all intended means, carry the mass of a portion of the medium, hence being easily labeled as “particles”, and their individual angular velocity is disregarded, hence “smooth”. The method relies heavily on integral interpolant theory [7]. An approximation to discrete Lagrangian points can be made, by a proper discretisation of the continuous integral by

$$A_i \approx \sum_j A_j W(\boldsymbol{r}_{\mathrm{ij}}, h) V_j, \tag{1}$$

called the summation interpolant, extended to all particles j, $|\boldsymbol{r}_{ij}| = |\boldsymbol{r}_i - \boldsymbol{r}_j| \le \epsilon h$, where V_j is the volume of particle j, A_i is the approximated variable at particle i and W is the weight, or kernel, function. The summation approximation implies that particle first order consistency, i.e., the ability of the kernel approximation to reproduce exactly a first order polynomial function, may not be assured, since the approximation error is inherent to the discrete form

$$\sum_j V_j W(\boldsymbol{r}_{\mathrm{ij}}, h) \approx 1 \tag{2}$$

may be large. This typically occurs near open boundaries or other discontinuities, where the kernel W does not satisfy compact support. Mitigation may be considered, as the Shepard and MLS corrections. In the work of [8] spatial gradients are computed using the gradient of the kernel function.

A Quintic [9] kernel is employed in this work:

$$W(\boldsymbol{r}_{\mathrm{ij}}, h) = \alpha_D \left(1 - \tfrac{q}{2}\right)^4 (2q + 1), \quad 0 \le q \le 2, \tag{3}$$

where $q = |\boldsymbol{r}_{ij}/h|$ and $\alpha_D = 21/16\pi h^3$, for a 3D case. The choice of kernel function weights on the quality of the solutions [10], with the Quintic kernel being recognized as a good choice for general free-surface problems [11].

2.1. Equations of motion in SPH

The proposed SPH formulation relies on the discretisation of the Navier–Stokes and continuity equations. Written for a variable density and neglecting the divergence of the velocity field, these are

$$\frac{d\boldsymbol{v}}{\mathrm{dt}} = -\frac{\nabla p}{\rho} + \frac{\mu}{\rho}\nabla^2 \boldsymbol{v} + \boldsymbol{g} \tag{4}$$

$$\frac{d\rho}{\mathrm{dt}} = -\rho \nabla \boldsymbol{v}, \tag{5}$$

where $\boldsymbol{v}$ is the velocity field, p is the pressure, ρ is the density and μ and g are the kinematic viscosity and body forces per unit mass, respectively. The system is written in such a way as to avoid solving a Poisson equation, using $p = f(\rho)$ [12], using a weakly compressible formulation. The continuity equation is discretised as

$$\frac{d\rho_i}{\mathrm{dt}} = \sum_j m_j (\boldsymbol{v}_i - \boldsymbol{v}_j) \cdot \nabla W(\boldsymbol{r}_{\mathrm{ij}}, h) + \Phi_i, \tag{6}$$

where m_j is the mass of particle j and Φ_i is a diffusive term [13], designed to stabilize the density field from high-frequency oscillations, written as

$$\Phi_i = 2\delta h c_0 \sum_j (\rho_j - \rho_i) \frac{\boldsymbol{r}_{\mathrm{ij}} \cdot \nabla W(\boldsymbol{r}_{\mathrm{ij}}, h)}{|r_{\mathrm{ij}}|^2} \frac{m_j}{\rho_j}, \tag{7}$$

where δ is a free parameter and c_0 is the numerical sound velocity. The discretised version of Eq. (4) [14] can be written as

$$\frac{d\boldsymbol{v}_i}{\mathrm{dt}} = -\sum_j m_j \left(\frac{p_i + p_j}{\rho_i \rho_j}\right) \nabla W(\boldsymbol{r}_{\mathrm{ij}}, h) + \boldsymbol{\Pi}_{\mathrm{ij}} + \boldsymbol{g} \tag{8}$$

The first term of the right side is a symmetrical, balanced form of the pressure term [7]. The second term represents viscous stresses, given by either an artificial viscosity formulation [7], or a laminar [15] and a sub-particle-scale (SPS) stress [16].

Following [7], the commonly used relationship estimate between pressure and density is Tait's equation

$$p_i = \frac{\rho_0 c_0^2}{\gamma}\left[\left(\frac{\rho_i}{\rho_0}\right)^{\gamma} - 1\right] \tag{9}$$

where ρ_0 is a reference density and $\gamma = 7$ for a fluid like water. According to Eq. (9), the compressibility of the fluid depends on c_0, in such a way that for a high enough sound celerity the fluid is virtually incompressible. However the value of c_0 in the model should not be the actual speed of sound, as the stability region is defined by

$$\Delta t = C \min\left[\min_i\left(\sqrt{\frac{h}{|f_i|}}\right);\ \min_i\left(\frac{h}{c_0 + \max_j \left|\frac{v_{\mathrm{ij}} r_{\mathrm{ij}}}{r_{\mathrm{ij}}^2}\right|}\right)\right], \tag{10}$$

where C is the Courant number, a constant of the order of 10^{-1} [10]. The first term results from the consideration of force magnitudes and the second is a version of the classical *CFL* condition. This expression takes into account numerical information celebrities and a restriction arising from the viscous terms [10]. If the sound celerity in the simulation is too high, it will render Δt very small and the computation more expensive. c_0 is kept to an artificial value of around 10 times the maximum flow speed, restricting the relative density fluctuations at less than 1% [7]. As a consequence, the estimated pressure field given by Eq. (9) usually shows some instabilities and may be subject to erroneous distributions. The δ-SPH diffusive terms contribute to the density field and smooth most of the high frequency oscillations.