Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 52(11) (2021) 779-782
DOI: 10.22060/mej.2019.15998.6246

Development of A Fully Lagrangian Smoothed Particle Hydrodynamics -Based Coupled Method for Simulation of Fluid–Structure Interaction

A. M. Salehizadeh, A. R. Shafiei*

Department of Mechanical Engineering, Yazd University, Yazd, Iran

ABSTRACT: In this research, an enhanced computational coupling method is proposed for the transient problems of incompressible fluid-elastic structure interaction based on the smoothed particle hydrodynamics method. The coupling process is conducted between an incompressible smoothed particle hydrodynamics fluid model and a totally Lagrangian smoothed particle hydrodynamics structural model. In the incompressible smoothed particle hydrodynamics method, due to the importance of smoothing particle distribution for accurate and stable simulations with noise-free pressure field, a new scheme for particle shifting has been proposed to regulate particle distribution. In contrast to numerical errors at the free surface in traditional particle shifting algorithm, this proposed algorithm as a suitable treatment for discontinuous boundaries such as the free surface presents an optimized particle shifting scheme without need to adjust the new parameters. The proposed numerical coupling method was examined by simulating several benchmarks in fluid-structure interaction and the results were compared with experimental and numerical results. The considered problems of fluid-structure interaction in this paper include the dam-breaking with an elastic gate and the deflection of an elastic obstacle due to fluid sloshing. The agreement between the presented results with the literature data shows the ability of the proposed model to simulate the phenomenon of fluid-structure interaction.

Review History:

Received: 2019/03/14
Revised: 2019/02/25
Accepted: 2019/05/05
Available Online: 2019/05/08

Keywords:

Incompressible smoothed particle hydrodynamics
Fluid-structure interaction
Totally Lagrangian SPH
Optimized particle shifting scheme

1- Introduction

In this study, a fully Lagrangian coupled particle method based on the Smoothed Particle Hydrodynamics (SPH) method is developed for simulation of Fluid Structure Interaction (FSI) problems that include transient dynamic loads with finite elastic structural displacements. The fluid and structure models are founded on the incompressible fluid and elastic structure. The various refined methods to decrease non-physical pressure oscillations have been proposed. For improvement of stability and accuracy of the SPH method for fluid model, a divergence free source term of Poisson Pressure Equation (PPE) and the proposed optimized particle regularization scheme is used.

The consistency of the fluid-structure coupling scheme in providing a rigorous imposition of the interface boundary conditions (continuity conditions) is a matter of significant importance that has to be carefully treated. Antoci et al. [1] proposed an FSI solver based on an approximate SPH evaluation of a surface integral of fluid pressure. Hence, the solver could become quite sensitive to fluctuations in the pressure field. In the context of projection-based methods, Rafiee et al. [2] developed a coupled SPH-based FSI solver for simulation of incompressible fluid-elastic structure. In their model, the poisson pressure equation was solved explicitly by a simple scheme and through an assumption of negligible pressure time variations in between two computational time steps which are not rigorously valid. Khayyer et al. [3] proposed an enhanced Incompressible Smoothed Particle Hydrodynamics (ISPH)-SPH coupled method which the stability of calculations is guaranteed through incorporation of numerical stabilizer that often requires tuning and brings computational challenges due to calculation of parallel and normal vectors of predicted relative distances. The present study provides a generalized treatment for coupling fluid and structure media with proposed particle shifting algorithm to stabilize the results.

2- Methodology

2- 1- ISPH methodology

In the SPH method, Navier-Stokes equations are solved in the form of a two-step method using the Poisson equation of pressure [4]. A stabilized method for the incompressible fluid has been used to obtain a free-divergence velocity field, which is expressed as follows [5]:

$$\vec{r}_a^{\,*} = \vec{r}_a^{\,n} + \vec{u}_a^{\,n}\Delta t \tag{1}$$

$$\vec{u}_a^{\,*} = \vec{u}_a^{\,n} + \Delta t\left[\nu^e \nabla^2 \vec{u}^{\,n} + \vec{g}\right] \tag{2}$$

*Corresponding author's email: arshafiei@yazd.ac.ir

Where ν^e denotes kinematic viscosity, $\vec{r}$ and $\vec{u}$ are particle position and velocity vectors, respectively. The PPE is obtained based on the concept of the projection method and expressed as:

$$\nabla.\left\langle \frac{1}{\rho}\nabla P^{n+1}\right\rangle_a = \frac{1}{\Delta t}\nabla \vec{u}_a^* \tag{3}$$

Where P and ρ are particle's pressure and density. The velocity at the time $n+1$ $\vec{u}_a^{n+1}$ will result from the projection of $\vec{u}_a^*$. Therefore

$$\vec{u}_a^{n+1} = \vec{u}_a^* - \frac{\Delta t}{\rho_a}\nabla P_a^{n+1} \tag{4}$$

Finally, the particle positions are advanced in time,

$$\vec{r}_a^{n+1} = \vec{r}_a^n + \Delta t\left(\frac{\vec{u}_a^n + \vec{u}_a^{n+1}}{2}\right) \tag{5}$$

The particle approximation of the above relations based on SPH method is presented as [5]:

$$\vec{\nabla}P_a \approx \sum_b V_b\left(P_a + P_b\right)\nabla_d W_{ab} \tag{6}$$

$$\left(\nabla.\nu^e\nabla\vec{u}\right)_a \approx \sum_b V_b \frac{\left(\rho_a\nu_a^e + \rho_b\nu_b^e\right)}{\rho_a}\frac{\vec{r}_{ab}.\nabla_a W_{ab}}{\left(r_{ab}^2+\eta^2\right)}\left(\vec{u}_a - \vec{u}_b\right) \tag{7}$$

$$\left(\nabla^2 P\right)_a \approx \sum_b 2V_b \frac{\vec{r}_{ab}.\nabla_a W_{ab}}{\left(r_{ab}^2+\eta^2\right)}\left(P_a - P_b\right) \tag{8}$$

Where $\nabla_a W_{ab}$ is the normalized kernel gradient [6]. Finally, to stabilize the simulation, the particles are then shifted slightly and the hydrodynamic variables are corrected [7]. The direction and amount of shifting are determined from the arrangement of neighboring particles; the proposed $\Delta\vec{r}_a$ is defined as shifting particle vector which is calculated by:

$$\Delta\vec{r}_a = \varepsilon\sum_b V_b \vec{r}_{ab} W_{ab}.\left(I_{2\times 2} - \vec{n}_a \otimes \vec{n}_a\right) \tag{9}$$

Where $\vec{n}_a$ denotes normal vector to the free surface and ε can vary between 0.0 and 0.2. Ultimately, it is necessary to modify the flow field variables in the new position as follows:

$$\vec{V}_a^f = \vec{V}_a + \Delta\vec{V}_a \;,\; \Delta\vec{V}_a = \delta\vec{r}_{aa'}.\left\langle\nabla\vec{V}\right\rangle_a \tag{10}$$

$$p_a^f = p_a + \Delta p_a \;,\; \Delta p_a = \delta\vec{r}_{aa'}.\left\langle\nabla p\right\rangle_a \tag{11}$$

2- 2- TLSPH methodology

In this study, the Totally Lagrangian Smoothed Particle Hydrodynamics (TLSPH) method is used to simulate the elastic dynamic problem [8]. The momentum equation in reference configuration may be rewritten in its discredited form as:

$$\frac{d\vec{u}_a}{dt} = \sum_{b=1}^{N_b} V_b\left(\frac{P_a^s + P_b^s}{\rho_a} - P_{\Pi_{ab}}\right).\vec{\nabla}_0 W_{0ab} + \vec{a}^{F\to S} \tag{12}$$

where $\vec{a}^{F\to S}$ corresponds to the interaction force acting on the fluid F by structure S, P^s is the nominal stress tensor which is related to the Cauchy stress tensor by $P^s = |F|\sigma F^{-1}$ and $P_{\Pi_{ab}} = |F|\prod_{ab} F^{-1}$ is the artificial viscosity written in the initial configuration. It should be noted that F is the deformation gradient tensor.

2- 3- Fluid-structure coupling scheme

The structure particles are considered as a moving wall boundary for the fluid, providing velocity and position boundary conditions in the calculation of fluid's pressure field through solving a PPE by consideration of momentum and continuity equations. The interacting forces normal to the fluid-structure interface would be equal in magnitude and opposite in direction, in view of the fact that:

$$\vec{\nabla}p_{F\to S}.n_F = \vec{\nabla}p_{S\to F}.n_S \tag{13}$$

It should be noted that the interaction term $\vec{a}^{FS}$ is obtained based on the calculated pressure gradient at a typical structure particle S as follows:

$$a_{F\to S} = -\frac{\nabla p_s}{\rho_s} = -\frac{1}{\rho_s}\sum_{k\in\Omega}\nabla p_{ks} \;;\; \Omega = \Omega_F \cup \Omega_s \tag{14}$$

Where k represents neighboring particles of fluid and structure particles.

2- 4- Time stepping condition

Generally, in SPH the time step sizes are determined through the Courant–Friedrichs–Lewy (CFL) condition:

$$\Delta t = Cr.\frac{h}{U} \tag{15}$$

Where the Courant number, $Cr < 1.0$. In this study, for all the conducted FSI simulations, the maximum allowable time step size of the structure $\Delta t^s_{\max}$ is set $0.01\Delta t^F_{\max}$.

3- Results & Discussion

The FSI solver is applied for simulation of dam break with an elastic gate. The elastic gate is of $0.005m$ thickness with Young's modulus and Poisson's ratio of $12Mpa$ and , respectively. The fluid partition consists of water. Fig. 1 presents a set of snapshots corresponding to the simulation of dam break with elastic gate.

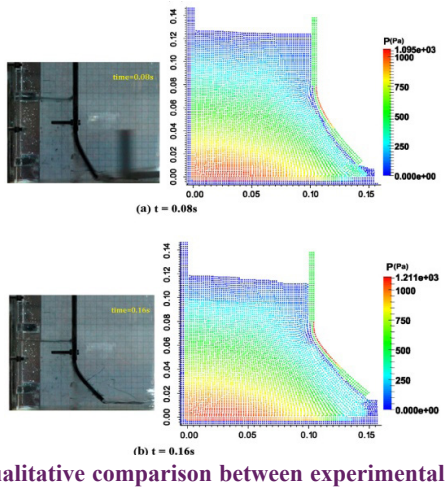

Fig. 1: Qualitative comparison between experimental photos[1] and their corresponding snapshots

4- Conclusion

In the present study, a novel coupled algorithm is developed based on the SPH method for simulating the FSI problems corresponding to incompressible fluid flows and elastic structures. An optimized particle regularization scheme is proposed for stable and accurate results. The proposed method is applied to the simulation of an FSI problem corresponding to a dam-break with an elastic gate. The simulation results are compared with experimental data.

5- References

[1] C. Antoci, M. Gallati, S. Sibilla, Numerical simulation of fluid–structure interaction by SPH, Computers & Structures, 85(11-14) (2007) 879-890.

[2] A. Rafiee, K.P. Thiagarajan, An SPH projection method for simulating fluid-hypoelastic structure interaction, Computer Methods in Applied Mechanics and Engineering, 198(33-36) (2009) 2785-2795.

[3] A. Khayyer, H. Gotoh, H. Falahaty, Y. Shimizu, An enhanced ISPH-SPH coupled method for simulation of incompressible fluid-elastic structure interactions, Computer Physics Communications, (2018).

[4] S.J. Cummins, M. Rudman, An SPH projection method, Journal of computational physics, 152(2) (1999) 584-607.

[5] S. Lind, R. Xu, P. Stansby, B.D. Rogers, Incompressible smoothed particle hydrodynamics for free-surface flows: A generalised diffusion-based algorithm for stability and validations for impulsive flows and propagating waves, Journal of Computational Physics, 231(4) (2012) 1499-1523.

[6] G.-R. Liu, M.B. Liu, Smoothed particle hydrodynamics: a meshfree particle method, World Scientific, 2003.

[7] R. Xu, P. Stansby, D. Laurence, Accuracy and stability in incompressible SPH (ISPH) based on the projection method and a new approach, Journal of computational Physics, 228(18) (2009) 6703-6725.

[8] J. Lin, H. Naceur, D. Coutellier, A. Laksimi, Geometrically nonlinear analysis of two-dimensional structures using an improved smoothed particle hydrodynamics method, Engineering Computations, 32(3) (2015) 779-805.