Oral presentation | Fluid-structure interaction

Fluid-structure interaction-III

Mon. Jul 15, 2024 4:30 PM - 6:30 PM Room A

[3-A-02] The moving rigid-body simulation with immersed boundary method and general pressure equation on multi-GPU cluster

*Jun-Yang Ji[1], Chao-An Lin[1] (1. Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 30013, Taiwan)

Keywords: Immersed boundary method, general pressure equation, multi-GPU acceleration

The moving rigid-body simulation with immersed boundary method and general pressure equation on multi-GPU cluster

Jun-Yang Ji* and Chao-An Lin*
Corresponding author: s111033802@m111.nthu.edu.tw
* Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 30013, Taiwan

1 Introduction

The fluid-structure problem is extensively studied. To tackle the challenges posed by fluid-structure interaction (FSI), researchers have proposed the Immersed Boundary Method (IBM). Liao et al. [1] developed the velocity-interpolation forcing method to predict the velocity field near the immersed boundary and directly compute the body force. This approach maintains second-order accuracy. For incompressible flow, the projection method [2] is prevalently used to couple the pressure and velocity fields. However, this method is time-consuming due to the solution of the pressure Poisson equation. To address this issue, Toutant [3] developed the General Pressure Equation (GPE) to achieve more accurate results. In the current study, the IBM, based on Liao et al. and combined with GPE, is employed to simulate two classic moving-body problems: the oscillating cylinder in a fluid at rest and the freely settling sphere problem. Both cases are implemented on a multi-GPU cluster to accelerate the simulation process.

2 Problem Statement

In the present study, two velocity and pressure coupling approaches are selected. The first is General Pressure Equation [3], the continuity equation is replaced by a general pressure equation to couple the pressure and velocity fields:

$$\frac{\partial p}{\partial t} + \rho c_s^2 \nabla \boldsymbol{u} = \frac{\gamma \nu}{Pr} \nabla^2 p$$

where c_s represents the artificial speed of sound, γ is the heat capacity ratio, ν is the kinematic viscosity. The second is projection method [2]. To implement this method on GPU, the four steps fractional-step method is used [4]. The fractional step method divides Navier-stokes equation into four steps:

$$\frac{\boldsymbol{u}^* - \boldsymbol{u}^n}{\Delta t} = -\nabla \cdot (\boldsymbol{u}^n \boldsymbol{u}^n) - \frac{1}{\rho} \nabla p^n + \nu \nabla^2 \boldsymbol{u}^n$$

$$\frac{\boldsymbol{u}^{**} - \boldsymbol{u}^*}{\Delta t} = \frac{1}{\rho} \nabla p^n$$

$$\frac{1}{\Delta t} \nabla \cdot \boldsymbol{u}^{**} = -\frac{1}{\rho} \nabla^2 p^{n+1}$$

$$\frac{\boldsymbol{u}^{n+1} - \boldsymbol{u}^{*}}{\Delta t} = -\frac{1}{\rho}\nabla p^{n+1}$$

where $\boldsymbol{u}^{*}$, $\boldsymbol{u}^{**}$ are intermediate velocities. First step, solve intermediate velocity $\boldsymbol{u}^{*}$ by Navier-Stokes equation with 3rd order explicit Runge-Kutta scheme. Second, the intermediate velocity $\boldsymbol{u}^{*}$ is obtained by removing pressure field. Third, the n+1 time step pressure field is solved by sub-iterating pressure Poisson equation. Finally, the new pressure field is added to revise the intermediate velocity.

To solve the velocity near the solid boundary, the velocity-interpolation forcing method developed by Liao et al. [1] is adopted. In Fig. 1 (a), the definition of each point is illustrated, while Fig. 1 (b) represents the $\boldsymbol{u}_F$, which is interpolated from $\hat{\boldsymbol{u}}_A$ and $\boldsymbol{U}_{move}$. Here, $\hat{\boldsymbol{u}}_A$ is the predicted velocity calculated from ordinary Navier-Stokes equation, and $\boldsymbol{U}_{move}$ is the moving velocity of solid body. The interpolated $\boldsymbol{u}_F$ is then utilized to estimate the immersed boundary force:

$$\boldsymbol{f}^{n+1} = \rho\frac{\boldsymbol{u}_F - \hat{\boldsymbol{u}}}{\Delta t}$$

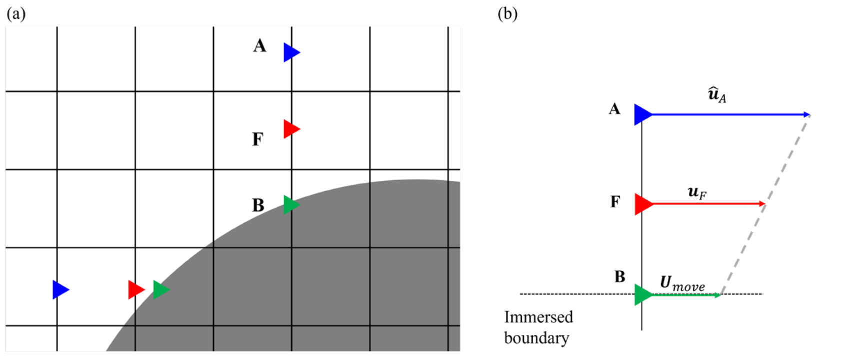

Figure 1: The schematic of velocity interpolation forcing method. (a) The location of nodes; (b) the velocity vectors.

3 Preliminary Results

The first result compares the oscillating cylinder in a fluid at rest with the experimental result from Düstch et al. [5]. In Fig. 2, the velocity profile results with Ma = 0.01 are more closer to the reference result than the Ma = 0.05 case. The second result examines the settling sphere problem in comparison with the experimental result from Ten Cate et al. [6]. Fig. 3 shows the position and velocity profiles at four Reynolds numbers. There is not much different between the results of two velocity-pressure coupling methods.

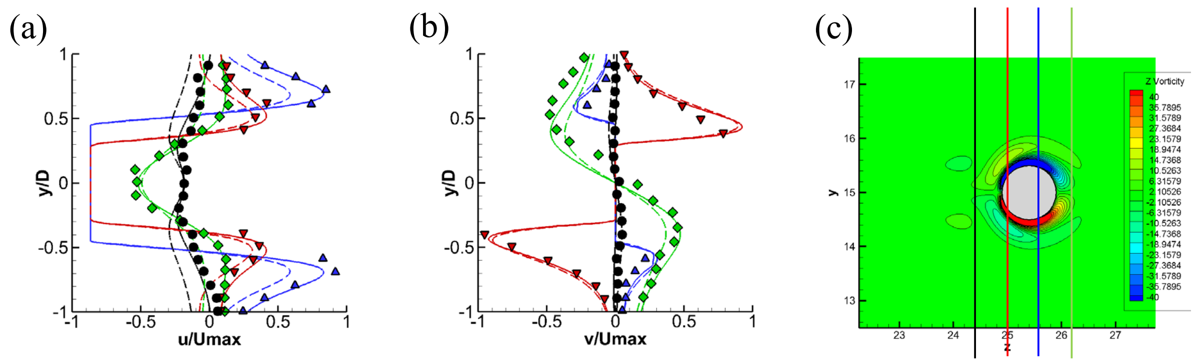

Figure 2: The velocity profiles of different Mach numbers (Ma = $\boldsymbol{U_{max}}/c_s$) in GPE equation, solid line for Ma = 0.01; dashed line for Ma = 0.05, symbols are Ten Cate et al. experimental result. (a) u velocity; (b) v velocity; (c) extracted position

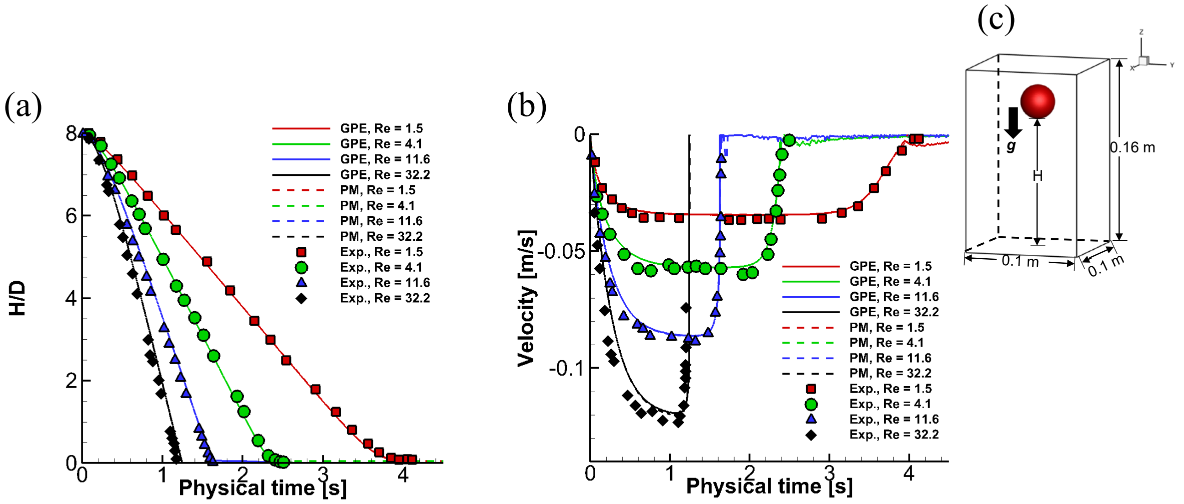

Figure 3: The (a) position, (b) velocity, and (c) schematic of single sphere settling problem.

References

[1] Liao, C. C., Chang, Y. W., Lin, C. A., & McDonough, J. (2010). Simulating flows with moving rigid boundary using immersed-boundary method. Computers & Fluids, 39(1), 152-167.

[2] Chorin, A. J. (1968). Numerical solution of the Navier-Stokes equations. Mathematics of computation, 22(104), 745-762.

[3] Toutant, A. (2017). General and exact pressure evolution equation. Physics Letters A, 381(44), 3739-3742.

[4] Choi, H., & Moin, P. (1994). Effects of the computational time step on numerical solutions of turbulent flow. Journal of Computational Physics, 113(1), 1-4.

[5] Dütsch, H., Durst, F., Becker, S., & Lienhart, H. (1998). Low-Reynolds-number flow around an oscillating circular cylinder at low Keulegan–Carpenter numbers. Journal of Fluid Mechanics, 360, 249-271.

[6] Ten Cate, A., Nieuwstad, C. H., Derksen, J. J., & Van den Akker, H. E. A. (2002). Particle imaging velocimetry experiments and lattice-Boltzmann simulations on a single sphere settling under gravity. Physics of Fluids, 14(11), 4012-4025.