The 18th International Conference on Finite Elements in Flow Problems (FEF2015)
16-18 March 2015, Taipei, Taiwan

INCOMPRESSIBLE SMOOTHED PARTICLE HYDRODYNAMICS MODELLING FOR SOLID-FLUID INTERACTION BASED ON ARTIFICIAL COMPRESSIBILITY METHOD

Burniadi Moballa[1], Ming-Jyh Chern[1], and Symphony Chakraborty[1]

[1] Department of Mechanical Engineering,
National Taiwan University of Science and Technology, Taipei, Taiwan
mjchern@mail.ntust.edu.tw

Key words: *Incompressible Smoothed Particle Hydrodynamics, solid-fluid interaction, artificial compressibility*

When a solid submerged in or surrounded by a fluid, the fluid force acting on the solid induces deformation and movement which alter the flow of the fluid. Consequently, the force on the solid will be further changed and so on. This interaction problem, the so-called solid-fluid interaction, encompasses many important applications in engineering such as the stability of aircraft wings, response of a bridge to wind and response of an offshore structure to ocean wave. There are two basic challenges in simulation of solid-fluid interaction problems. The first challenge is how to model the solid geometry. The second challenge is how to predict the forces exerted by the fluid on the solid accurately.

The smoothed particle hydrodynamics (SPH) is a mesh-free and Lagrangian method. It was developed for astrophysics application and then successfully applied to hydrodynamics problems especially free-surface flow. The mesh-free nature of the smoothed particle hydrodynamics (SPH) method offers an advantage when used to model solid-fluid interaction. It is relatively easy to discretize the solid boundary using particle.

Accurate prediction of the fluid force implies the need of an accurate pressure solver. Originally, SPH method for fluid flow used a weakly compressible fluid model when applied to simulate incompressible fluid. The density is allowed to slightly change and the pressure is calculated using an equation of state. The pressure field predicted using this method is plagued by noise or oscillation. To remedy this problem, another approach using fractional step or pressure projection method was proposed by Cummins and Rudman [1]. In their method, pressure field is obtained by solving the pressure-Poisson equation (PPE) instead of using equation of state. However, the coefficient matrix of the PPE must be reconstructed at every time step because the particle configuration is changing due to the Lagrangian nature of the SPH method. Eulerian method like finite volume method does not suffer from this problem because the positions of the computational points do not change with time.

In the SPH method, the velocity and the pressure are defined at the same location as the consequence of the particle-based approach. This is similar to the co-located approach in the grid based method. Therefore, the pressure field predicted by ISPH method could also suffer from the same problem that happen co-located method, the so-called checkerboarding. If left untreated, the checkerboarding will limit the accuracy of the predicted pressure field.

In this study, we aimed to develop an incompressible SPH method using different approach. We applied the artificial compressibility method to solve the pressure field. Using the artificial compressibility method, the pressure is obtained by iterative correction thus eliminating the need to construct and solve the PPE directly. We also developed an SPH divergence operator incorporating the interpolation method proposed by Rhie and Chow [2] to overcome the checkerboarding problem.

We applied the method to simulate flow past circular cylinder between parallel walls. The time history of the drag coefficient and the steady state vorticity contours can be seen in Fig. 1. We validated the result predicted by the current ISPH method with previous work using well-established grid-based method. The comparison showed that the result predicted by the current ISPH method is in good agreement with previous work.

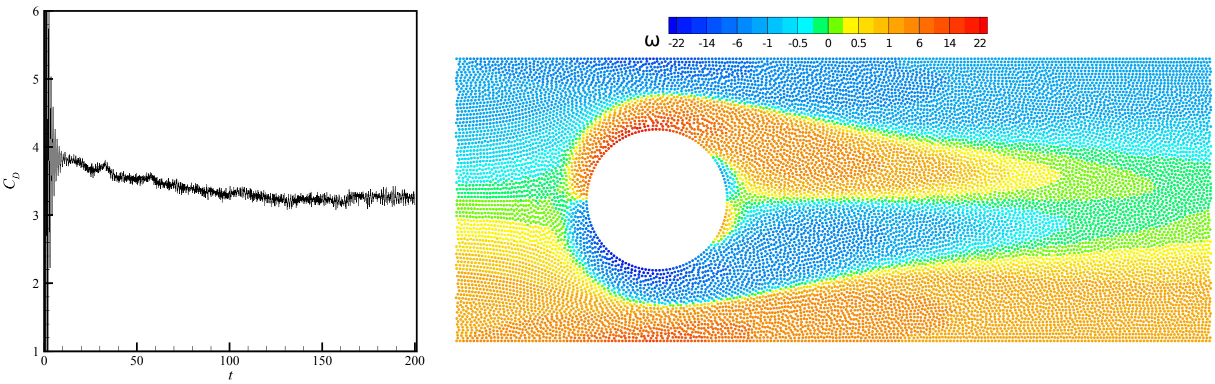

Figure 1: Flow past circular cylinder between parallel walls at $Re = 100$ and channel height to cylinder diameter ratio of 2.0. The average drag coefficient is 3.25. This value is in good agreement with the value obtained by [3] at 3.14. Left: time history of drag coefficient. Right: vorticity.

REFERENCES

[1] Cummins SJ, Rudman M (1999) An SPH projection method. *J. Comput. Phys.* **152**:584-607.

[2] Rhie CM, Chow WL (1983) Numerical study of the turbulent flow past an airfoil with trailing edge separation. *AIAA J.* **21**:1525-1532.

[3] Singha S, Sinhamahapatra KP (2010) Flow past a circular cylinder between parallel walls at low Reynolds numbers. *Ocean Engineering* **37**:757-769.