8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

NUMERICAL SIMULATIONS OF IMMERSED COLLISIONS OF TETHERED SPHRERS

***Ching-Biao Liao[1], Cheng-Hsin Chen[2]**

[1] Professor of Department of Water Resources Engineering and Conservation,
Feng Chia University, Taichung, Taiwan
No. 100 Wenhwa Rd., Seatwen, Taichung, Taiwan 40724
E-mail: cbliao@fcu.edu.tw
[2] Doctoral Student of Graduate Institute of Civil and Hydraulic Engineering, Feng Chia University, Taichung, Taiwan 40724
E-mail: p9317786@fcu.edu.tw

Key Words: *Pendulum, Immersed Collisions, Immersed Boundary Method, Volume of Solid*

ABSTRACT

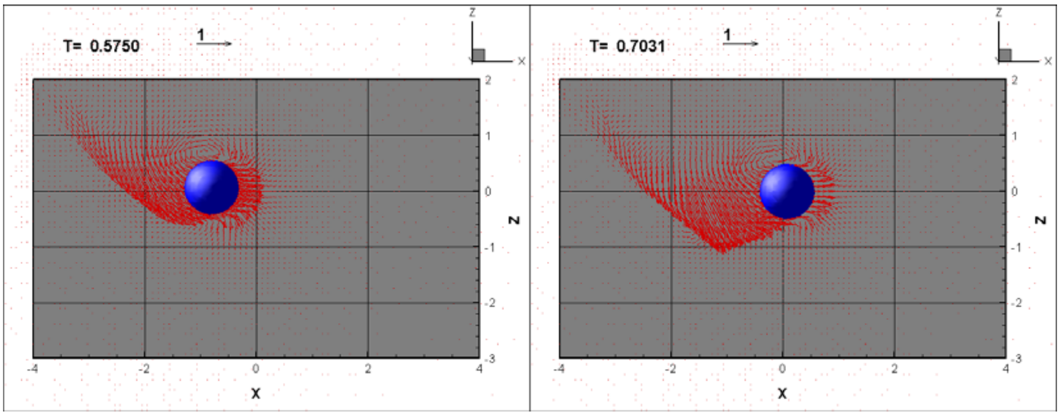
Fluid-structure interaction problems have been a challenging subject for computational fluid dynamics. Many novel and efficient immerse boundary methods have been recently applied successfully to solve difficult fluid-structure interaction problems. The simplified immersed boundary method is developed for modeling pendulum motion of a sphere and immersed collisions of tethered spheres. This problem was investigated experimentally by Hsu and Capart in 2007. The incompressible Navier-Stokes equations together with the IBM are discretized by finite difference method in simple staggered Cartesian grids. Projection method is employed here with pressure Poisson equation to replace divergence-free condition. Second-order Adams-Bashforth method is used for the time-integration of momentum equations. The overall accuracy in space and time is second order. As to the solid part, an immerse boundary method with direct forcing is applied to simulate a solid object moving in fluids. Volume of solid interpolation is employed to obtain further accuracy in space for flow near the solid without losing the easiness to compute the force exerted on solid.

The numerical method is validated by the simulation of three-dimensional lid-driven cavity flow with a sphere which located at center. The result shows that the accuracy of numerical model is about second-order. The motion of pendulum can be solved from the Newton second law of motion while consider the gravity force, buoyancy force, drag force, and added mass force. We use the experimental data of Hsu and Capart (2007) to verify the numerical model for this moving object problem. Figure 1 shows the computed flow field of a single pendulum. The flow pattern of immersed collisions of tethered spheres is shown in Figure 2. They are shown a good agreement compared with experimental results. In conclusion, our current method is efficient, straightforward, robust and successful in solving fluid-structure interaction problems.

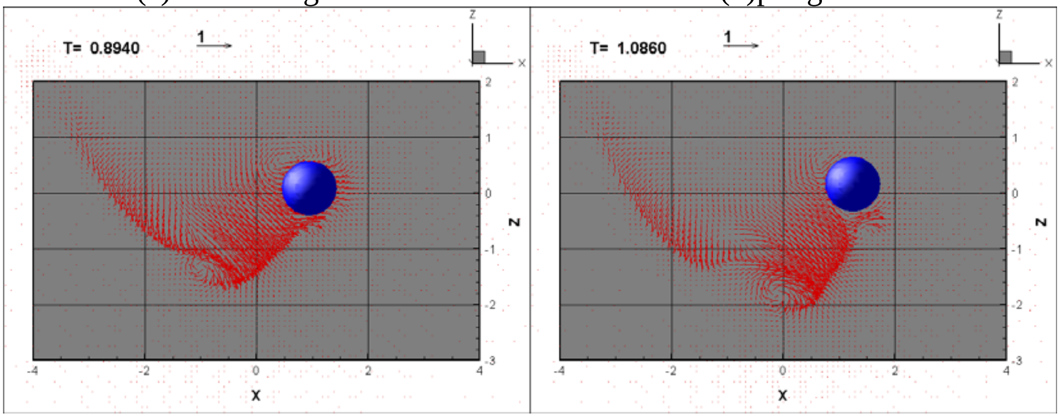
REFERENCES

[1] Hui-Chi Hsu and Hervé Capart, "Enhanced upswing in immersed collisions of tethered spheres", Physics of fluids, Vol. **19**, 101701, (2007).

[2] P. Alexander and E. Indelicato, "A semiempirical approach to a viscously damped oscillating sphere", *Eur. J. Phys.*, Vol. **26**, p.p. 1-10, (2005).

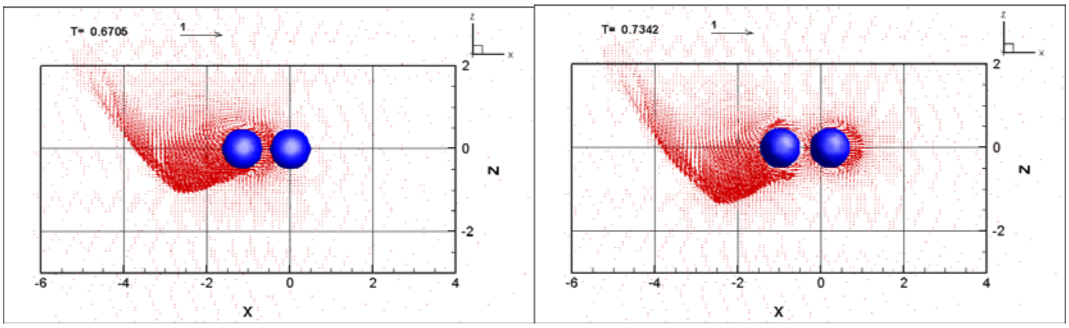


(a)downswing (b)perigee

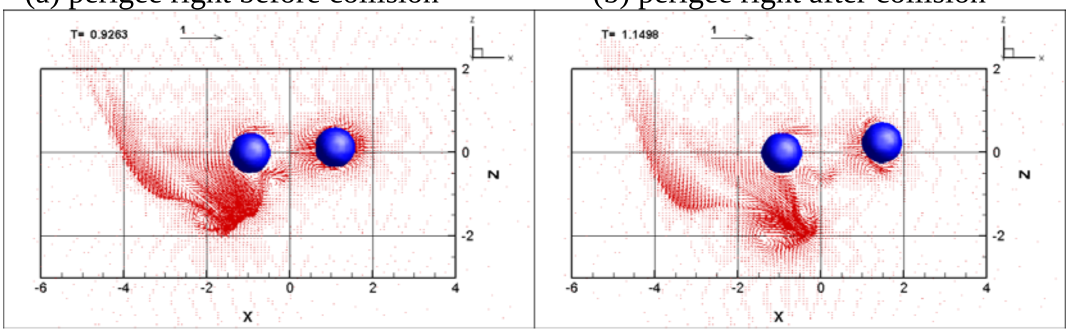


(c)upswing (d)apogee

fig. 1. Computed flow fields at successive stages in the motion of a single pendulum: (a) downswing, (b) perigee, (c) upswing, and (d) apogee.



(a) perigee right before collision (b) perigee right after collision



(c) upswing (d) apogee

fig. 2. Computed flow fields at successive stages in the motion of colliding pendulums: (a) perigee right before collision, (b) perigee right after collision, (c) upswing, and (d) apogee.