Numerical investigation of flow past a horizontal circular cylinder bounded by free surface and wall boundary using incompressible smoothed particle hydrodynamics

Burniadi Moballa and Ming-Jyh Chern

Department of Mechanical Engineering
National Taiwan University of Science and Technology
Taipei 10607, Taiwan
e-mail: d10103803@mail.ntust.edu.tw

ABSTRACT

The flow past a circular cylinder is one of the most studied problems in fluid dynamics. The configuration where the cylinder is placed in an infinite medium under uniform flow has been explored in detail. Many studies can also be found on the effect of proximity to a wall boundary. However, only a few studies can be found on the effect of both wall and free surface boundaries, the configuration which represents an open channel. The aim of the present study is to numerically investigate this configuration using the incompressible smoothed particle hydrodynamics method. Smoothed particle hydrodynamics (SPH) is a Lagrangian mesh-free method originally developed to simulate phenomena in astrophysics. Later it is found that the Lagrangian nature of SPH makes it suitable for free surface tracking. However, the lack of inlet and outlet algorithms and inaccurate pressure field due to noise and checkerboarding prevent wide application of the method to fluid-solid interaction problem. In the present study, we developed an incompressible SPH (ISPH) method based on the incremental projection to obtain more accurate pressure field. In the SPH method, the continuum is discretized into a set of particles. Each particle carries velocity and pressure. Therefore, the pressure and velocity are defined at the same location similar to the collocated grid. As a consequence, it also suffers from the checkerboarding problem found in collocated grid -based method. In order to treat this problem, we adopted the velocity interpolation method proposed by Rhie and Chow[1] to the current ISPH method. We implement a simple inlet and outlet treatment by dragging the particle that moves beyond the outlet back to the inlet. We conducted a parameter study to investigate the effect of Froude number and the gaps to the free surface and the bottom boundary to the flow pattern and hydrodynamic force while the Reynolds number is held constant.

REFERENCES

[1] Rhie, C.M. and Chow, W.L. Numerical study of the turbulent flow past an airfoil with trailing edge separation. *AIAA J.* Vol. **21**, pp. 1525-1532, (1983).