Numerical Simulation of Cavitation Flows around NACA66 Hydrofoil by Adaptive Mesh Refinement

Minsheng Zhao1, Decheng Wan1*, Zhonghua Li2

1 Computational Marine Hydrodynamics Lab (CMHL), State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
2 Nanjing Hydraulic Research Institute, Nanjing, China

Corresponding author

ABSTRACT

The main attention of present work is to investigate the ability of a adaptive mesh generation method in accurate cavitation simulation. A numerical study of the cavitation around NACA66 hydrofoil has been carried out. The simulation results such as cavitation shape, the lift coefficient C_l, and the drag coefficient C_d are analyzed and compared with each other. It is found that the adaptive mesh refinement can effectively capture the change of cavitation morphology, especially the cavitation shedding process, while the the calculation cost is generally lowered.

KEY WORDS: OpenFOAM; cavitation flow; adaptive mesh refinement; NACA66 hydrofoil

INTRODUCTION

Cavitation is a physical phenomenon which occurs when the local pressure of liquid drops below the limit of static vapor pressure. This severe vaporization has been a research hotspot in fluid mechanics field for it leads to problems such as pressure fluctuation, increased drag force, vibration, and noise( Ghorbanim et al., 2015; WU J et al., 2005). On the other hand, it is required to be able to utilize cavitation in some cases like supercavitation torpedo. At present, the speed of ships is generally increased, and the occurrence of cavitation appearing in hydraulic devices including pumps, turbines, and propellers is inevitable. Therefore it is important to study cavitation through numerical methods.

Cavitation around hydraulic machineries have been studied for decades. A number of experiments were carried out to investigate cavitation phenomena, because it was the most efficient way, and possibly the only way. Attached cavitation that formed on stationary hydrofoils with significant sweep was tested by Crimi(1970), Bark(1986), and Ihara, Watanabe & Shizukuishi(1989). The unsteady cavitation in mixed-flow pumps have been experimentally and numerically investigated by Kobayashi and Chiba(2010) with data close to the ones obtained with other researchers. Among the different types of cavitation around hydraulic machineries, the tip vortex cavitation and the sheet cavity are the focus of propeller cavitation study, (Oprea and Bulten, 2009; Bensow and Bark, 2010). When the ship hull is also concerned, the impacts of the irregular wake behind the ship to the propeller cavitation is a research emphasis with experimental and numerical methods. The unsteady cavitation characteristics have been studied through experimental research with a simplified hydrofoil in the cavitation tunnel, from standard NACA hydrofoils (Wu et al., 2015; Zhang et al., 2014) or Clark-Y (Huang et al., 2014; Kato, 2011) to special types of profiles like a plano-convex hydrofoils (Coutier et al., 2007; Le et al., 1993).

Experimental method is the most direct way to study cavitation, however, this method has disadvantages that can not be ignored. Firstly, the scale effect is one of the technical constraints of the experimental results. Secondly, the re-entrant jets can not be described through high speed cameras or other experimental equipments, yet it is critical to the periodical shedding of the cavitation and causes the collapse of a sheet cavity (Foeth, 2008). In addition to the above, the most important reason to use numerical methods instead of experimental methods is that the latter can be extremely resources consuming. Hence the numerical simulation of cavitation is becoming more popular in recent years.

At first the potential flow was used in computational fluid dynamics areas, including cavitation study. This approach, however, has difficulties dealing with vertical structures. In the last years, the viscous CFD method is being used in numerical simulations. In recent years, the numerical simulation of cavitation flow by solving viscous fluid dynamics control equation (N-S) is the development trend. Solving the governing equations of viscous fluid mechanics numerically can only not consider the viscosity effect of the flow process, but also save the trouble of including the nonphysical cavity closure hypothesis.

In the development of this method, the research of numerical cavitation model is always the focus of it. There are three popular cavitation