Commun. Comput. Phys.
doi: 10.4208/cicp.050412.140812a

Vol. **14**, No. 2, pp. 328-354
August 2013

Simulation of Wave-Flow-Cavitation Interaction Using a Compressible Homogenous Flow Method

J. G. Zheng[1], B. C. Khoo[1,2,3,*] and Z. M. Hu[3]

[1] *Department of Mechanical Engineering, National University of Singapore, Singapore 119260.*
[2] *Singapore-MIT Alliance, National University of Singapore, Singapore 117576.*
[3] *Temasek Laboratories, National University of Singapore, Singapore 117411.*

Received 5 April 2012; Accepted (in revised version) 14 August 2012

Abstract. A numerical method based on a homogeneous single-phase flow model is presented to simulate the interaction between pressure wave and flow cavitation. To account for compressibility effects of liquid water, cavitating flow is assumed to be compressible and governed by time-dependent Euler equations with proper equation of state (EOS). The isentropic one-fluid formulation is employed to model the cavitation inception and evolution, while pure liquid phase is modeled by Tait equation of state. Because of large stiffness of Tait EOS and great variation of sound speed in flow field, some of conventional compressible gasdynamics solvers are unstable and even not applicable when extended to calculation of flow cavitation. To overcome the difficulties, a Godunov-type, cell-centered finite volume method is generalized to numerically integrate the governing equations on triangular mesh. The boundary is treated specially to ensure stability of the approach. The method proves to be stable, robust, accurate, time-efficient and oscillation-free.

Novel numerical experiments are designed to investigate unsteady dynamics of the cavitating flow impacted by pressure wave, which is of great interest in engineering applications but has not been studied systematically so far. Numerical simulation indicates that cavity over cylinder can be induced to collapse if the object is accelerated suddenly and extremely high pressure pulse results almost instantaneously. This, however, may be avoided by changing the traveling speed smoothly. The accompanying huge pressure increase may damage underwater devices. However, cavity formed at relatively high upstream speed may be less distorted or affected by shock wave and can recover fully from the initial deformation. It is observed that the cavitating flow starting from a higher freestream velocity is more stable and more resilient with respect to perturbation than the flow with lower background speed. These findings may shed some light on how to control cavitation development to avoid possible damage to operating devices.

---

[*] Corresponding author. *Email addresses:* `mpezhen@nus.edu.sg` (J. G. Zheng), `mpekbc@nus.edu.sg` (B. C. Khoo), `huzm@imech.ac.cn` (Z. M. Hu)

AMS subject classifications: 35L40, 65M08, 65M22, 76T10

Key words: Cavitating flow, isentropic cavitation model, shock-flow-cavitation interaction.

1 Introduction

Vapour-filled cavities forming in a liquid flow when the pressure drops below the saturated vapour pressure is commonly referred to as cavitation. The cavitation phenomenon is undesired in most situations. An example is that the high pressure pulse and jetting following the collapse of cavitation can damage engineering devices such as propeller, turbomachinery, hydrofoil, etc. On the other hand, attempts also have been made to utilize cavitation effects to improve the performance of engineering machines. For example, the drag exerted on an underwater projectile can be reduced significantly by a supercavitating vapour enveloping the whole object, since the viscous drag is normally about 1000 times less in vapour than in liquid water. During the past few decades, investigation of cavitation has received increasing attention.

Numerically simulating cavitating flow is challenging in terms of modeling of complex physics in the interfacial region separating liquid and vapour phases as well as development of robust numerical methods. Up to now, quite a few numerical approaches have been developed to resolve the cavitating problems and can roughly be categorized into two families. One of them is the so-called sharp interface method where the interface separating the cavitation region and liquid flow is assumed to be a sharp discontinuity and tracked accurately using an iterative procedure, see [5,6] for a brief discussion. Unfortunately, the sharp interface method appears to be able to handle simple cavitating problems only. The other is the widely used diffuse interface method. Instead of attempting to locate the phase interface accurately, the method allows the flow to transmit from vapour to liquid phase smoothly, leading to a numerically diffused zone along the cavitation boundary. The diffuse interface method is becoming increasingly popular because it is simpler than the sharp interface method and can still model some physical details of the cavitating flows accurately. In practical implementation, one has two choices of governing equations, i.e. single-phase or two-phase formulation, on which the complexity of the diffuse interface method depends. For the strict two-phase model, both phases coexist at every point in the flow field and thus one has to solve the separate governing equations for each phase. The complex physics on the interfacial region such as heat and mass transfer, surface tension and restoration of pressure equilibrium between different phases can be taken into account in this type of model. But the governing equations usually involve transfer terms accounting for phase transition and interaction and may not be conservative, and therefore it is not trivial to efficiently solve them. Another disadvantage of the two-phase formulation is that it is difficult to know the parameters associated with phase transition a priori. On the contrary, one-phase model treats the cavitating flow as a single-fluid flow of the mixture of liquid and vapour phases. The flow is gov-