

A Preliminary Calculation of Three-Dimensional Unsteady Underwater Cavitating Flows Near Incompressible Limit

Yang-Yao Niu* and Wei-Yuan Hsu

Department of Mechanical Engineering, Chung-Hua University, Hsin-Chu, Taiwan, 30067, ROC.

Received 9 October 2007; Accepted (in revised version) 10 December 2007

Communicated by Kun Xu

Available online 29 May 2008

Abstract. Recently, cavitated flows over underwater submerged bodies have attracted researchers to simulate large scale cavitation. Comparatively Computational Fluid Dynamics (CFD) approaches have been used widely and successfully to model developed cavitation. However, it is still a great challenge to accurately predict cavitated flow phenomena associated with interface capturing, viscous effects, unsteadiness and three-dimensionality. In this study, we consider the preconditioned three-dimensional multiphase Navier-Stokes equations comprised of the mixture density, mixture momentum and constituent volume fraction equations. A dual-time implicit formulation with LU Decomposition is employed to accommodate the inherently unsteady physics. Also, we adopt the Roe flux splitting method to deal with flux discretization in space. Moreover, time-derivative preconditioning is used to ensure well-conditioned eigenvalues of the high density ratio two-phase flow system to achieve computational efficiency. Validation cases include an unsteady 3-D cylindrical headform cavitated flow and an 2-D convergent-divergent nozzle channel cavity-problem.

AMS subject classifications: 76B10, 76D05, 65M10

Key words: Cavitation, incompressible, Roe Riemann solver, dual-time stepping.

1 Introduction

Cavitation is one of important phenomena in the hydrodynamics problems. Cavitation is well known to associate with three aspects: formation, growth and collapse of bubbles within the body of a liquid due to the process of nucleation in a liquid flow where

*Corresponding author. *Email addresses:* yniu@chu.edu.tw (Y.-Y. Niu), m09408021@chu.edu.tw (W.-Y. Hsu)

the pressure falls below the vapor pressure. This phenomenon usually arises in flows around solid bodies, the turning corner of convergent nozzle, leading edge of the blades and tip leakage. It strongly affects the flow field to induce severe noise and structural erosion which result in supplementary maintenance operations of the turbomachinery and pump.

One pattern of cavitating flows might occur partially on the body surface or it might grow until it becomes very long compared with the body dimensions. It is always denoted as the super-cavitation which has been successfully formulated as near steady-state and incompressible flow models [1]. However, in high speed supercavitating flows around underwater projectiles, dramatic increase of void propagation associated with unsteady mass transferring dynamics gives rise to the cloud collapse physics. The large transient variation of interfacial dynamics such as density, void fraction and viscosity gradients around the interfaces naturally attract unsteady formulation of numerical method constructed. Several numerical methods had been proposed in the past in order to simulate the related phenomena in cavitation flows, such as the VOF method [2,3], the level set method [4] and the interface capturing method [5]. Recently, one common approach to combine the VOF model with the interface capturing scheme to constitute the so-called Eulerian approach based on separated equations to describe different fluids or phases. Several different Eulerian formulations are usually used to capture the interface, namely the multi-fluid model [6–9], the mixture model [10–17] and the ghost cell model [18].

The multi-fluid model containing individual particles, droplets or bubbles is through the so-called two-fluid model, in which the time or space ensemble average process is applied to both the continuous and disperse phase. Two sets of Navier-Stokes equations are used to describe both phases of fluids with additional inter-phasic terms for the exchange of momentum and energy between phases. Since each phase has its own velocity and temperature, the two-fluid model allows both mechanical and thermal non-equilibrium to be considered in the modeling; in that respect, it represents a more general model for two-phase flows. Two-fluid model have also been utilized for natural cavitation. However, in super-cavity flows, the gas-liquid interface is known to be nearly in dynamic equilibrium; for this reason, we do not pursue a full two-fluid level of modeling. In the mixture model, the mass, momentum and energy of the multi-phase flow are described by a set of Navier-Stokes equations. Usually among the mixture model, a single continuity equation is considered with the abrupt variations of density between vapor and liquid phases through a condensation-evaporation process. Such single-continuity-equation-homogeneous type mixture methods have become fairly widely used for sheet and supercavitating flow analysis [1, 10–16]. Here, a time-accurate preconditioned type three-dimensional multiphase Navier-Stokes analysis near the limit of incompressible flow is extended based on the mixture flow model for the prediction of underwater cavitation problems. A dual-time stepping method is used for the unsteady computations by introducing artificial time terms in the governing equations and highly accurate resolution of phase interfaces will be introduced.

The paper is organized as follows. In Section 2, the governing equations will be pre-