European Journal of Mechanics B/Fluids 30 (2011) 26–40

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

journal homepage: www.elsevier.com/locate/ejmflu

Numerical study of unsteady turbulent cavitating flows

Eric Goncalvès

LEGI-Grenoble INP, 1025 rue de la Piscine, 38400 St Martin d'Heres, France

ARTICLE INFO

Article history:
Received 8 April 2010
Received in revised form
27 July 2010
Accepted 29 August 2010
Available online 9 September 2010

Keywords:
Cavitation
Homogeneous model
RANS simulations
Turbulence model

ABSTRACT

The simulation of cavitating flows is a challenging problem both in terms of modelling the physics and developing robust numerical methodologies. Such flows are characterized by important variations of the local Mach number, compressibility effects on turbulence and involve thermodynamic phase transition. To simulate these flows by applying homogeneous models and Reynolds averaged codes, the turbulence modelling plays a major role in the capture of unsteady behaviours. This paper presents a one-fluid compressible Reynolds-Averaged Navier–Stokes (RANS) solver with a simple equation of state (EOS) for the mixture. A special focus is devoted to the turbulence model influence. Unsteady numerical results are given for Venturi geometries and comparisons are made with experimental data.

1. Introduction

The simulation and prediction of cavitating flows is of critical importance for the efficient design and performance of many engineering devices (turbomachinery, turbopumps in rocket propulsion systems, hydrofoils, fuel injectors, marine propellers, nozzles, underwater bodies, etc.). In most cases, cavitation is an undesirable phenomenon, significantly degrading performance, resulting in reduced flow rates, lower pressure increases in pumps, load asymmetry, vibrations, noise and erosion. In most industrial applications, cavitating flows are turbulent and the dynamics of the interface formed involves complex interactions between the vapour and liquid phases. These interactions are not well understood in the closure region of cavities, where a distinct interface may not exist and where the flow is unsteady.

Various numerical models, within the framework of the continuum modelling method, have been developed to investigate cavitating flows. This method makes no attempt to track the liquid and vapour interface but treats the flow as two phases with an averaged mixture density, which continuously varies between the liquid and vapour extremes. In its implementation, there are different approaches according to the assumptions made: equilibrium models versus non-equilibrium models and homogeneous models versus two-fluid models.

The two-fluid approach assumes that both phases co-exist at every point in the flow field and each phase is governed by its own set of conservation laws. These models can take into account the physical details occurring in the cavitation phenomenon such as mass exchange, thermal transfer and surface tension. However, transfer terms have to be known; such quantities are usually very difficult to obtain.

On the contrary, the one-fluid method treats the cavitating flows as a mixture of two fluids behaving as one. The governing equations are composed of three conservation laws written for the mixture. These models are based on the assumption of local kinematic equilibrium between phases (the local velocity is the same for both phases). With the assumption of local thermodynamic equilibrium (local temperature, pressure and free Gibbs enthalpy equality between phases), they constitute the homogeneous equilibrium models (HEM). These model cannot reproduce strong thermodynamic or kinetic non-equilibrium effects but, because of its simplicity, it is often used for numerical simulations. An equation of state (EOS) is necessary to close the system. Different closure relations that link the pressure to the temperature and the density have been proposed: tabulated EOS [1–3], a mixture entropy maximization procedure with a relaxed pressure law [4], barotropic mixture laws [5–12] and equilibrium EOS [13,14].

When the non-equilibrium effect becomes important, additional equations are needed for an accurate prediction. Hybrid or reduced models are intermediate models between one-fluid and two-fluid models, based on four or five equations. A four-equation model is obtained by adding a mass equation for the vapour or liquid density including a cavitation source term. This model is very popular in simulating cavitating flows [15–23]. The main difficulty is related to the formulation of the source term and the tunable parameters involved in the vaporization and condensation process (different formulations and sets of parameters are presented in [24]).

E-mail address: Eric.Goncalves@legi.grenoble-inp.fr.

0997-7546/$ – see front matter © 2010 Elsevier Masson SAS. All rights reserved.
doi:10.1016/j.euromechflu.2010.08.002

In addition, the turbulence modelling plays a major role in the capture of unsteady behaviours. Cavitation sheets that appear on solid bodies are characterized by a closure region which always fluctuates with the existence of a re-entrant jet. This one is mainly composed of liquid which flows upstream along the solid surface. Moreover, due to the drastic decrease of the speed of sound in the mixture, large supersonic areas appear leading to compressibility effects on turbulence. These effects and interactions with two-phase structures are not yet well known and understood.

For usual applications, three-dimensional time-dependent computations obtained with large eddy simulations (LES) or direct simulations (DNS) are not yet tractable. The Reynolds decomposition is often used with an averaged statistical processing resulting in the RANS equations for the mean flow quantities. One fundamental problem with the RANS approach is that turbulence models are tuned by steady-state mean flow data. Moreover, the standard eddy-viscosity models based on the Boussinesq relation are known to over-product eddy viscosity, which reduces the development of the re-entrant jet and two-phase structures shedding [25]. The limitation of the turbulent viscosity is a determinant point to capture realistic cavitation sheets. Different methods have been investigated to limit or to correct standard turbulence models. An arbitrary modification was proposed by Reboud to reduce the turbulent viscosity [25] and has successfully been used by different authors [16,26–30]. Other corrections are based on the modelling of compressibility effects of the vapour/liquid mixture in the turbulence model. Correction terms proposed by Wilcox [31] in the case of compressible flows have been tested for unsteady periodic cavitating flows [26]. A sensitivity analysis of constants $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$, which directly influence the production and dissipation of turbulence kinetic energy, was conducted for a k–ε model and a cavitating hydrofoil case [32]. Finally, filter-based methods were investigated [23] by which the sub-filter stresses are constructed directly using the filter size and the k–ε turbulence closure.

The present work is part of a research aimed at developing a numerical tool devoted to cavitating flows. In previous works [33,34], an in-house finite-volume code solving the Reynolds-Averaged Navier–Stokes (RANS) compressible equations was developed with an homogeneous approach. First computations were performed to assess numerical aspects and thermodynamic constraints on EOS. This paper is a complement work in which particular emphasis is placed on the study of turbulence models and corrections for unsteady flow, such as eddy viscosity limiters and realizability constraints. Our goals in this study are

- to asses the sensitivity of a number of turbulence models in cavitating flows, originally developed for aerodynamic applications;
- to investigate the effect of various eddy viscosity corrections for flows involving different cavity sheet behaviours. This would help in gaining an insight into the range of predictions that we can expect with these models.

The paper is organized as follows: the theoretical formulation is summarized, including physical models and elements of the numerical methods. This is followed by sets of results on Venturi geometries and discussions.

2. Governing equations and models

The numerical simulations were carried out using an in-house CFD code solving the one-fluid compressible RANS system. First, we present the one-fluid formulation for Euler equations, with the associated preconditioned method. Secondly, the complete RANS system is presented.

2.1. The one-fluid Euler equations

The homogeneous model assumes strong coupling of the phases and moving at the same velocity components. The phases are assumed to share the same temperature T and the same pressure P. The evolution of the two-phase flow can be described by Euler equations that employ the representative flow properties as unknowns just as a single-phase problem. The mixture density ρ is defined by

$$\rho = \alpha\rho_V + (1-\alpha)\rho_L \tag{1}$$

where ρ_L and ρ_V are respectively the liquid and vapour densities. The void ratio α characterizes the volume of vapour in each cell: $\alpha = 1$ means that the cell is completely filled by vapour; inversely, a complete liquid cell is represented by $\alpha = 0$. Liquid and vapour phases are characterized by their thermodynamic properties. On each cell, the unknowns are calculated by averaging them by the volume occupied.

In order to simplify the formulation, we present below the one-dimensional equations, expressed in conservative variables $w = (\rho, \rho u, \rho E)$:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} = 0 \tag{2}$$

$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2 + P)}{\partial x} = 0 \tag{3}$$

$$\frac{\partial (\rho E)}{\partial t} + \frac{\partial (\rho u H)}{\partial x} = 0 \tag{4}$$

where $E = e + u^2/2$ denotes the total energy and $H = h + u^2/2$ the total enthalpy; e the internal energy, h the enthalpy, which are related to the void ratio and the liquid and vapour values:

$$\rho e = \alpha\rho_V e_V + (1-\alpha)\rho_L e_L \tag{5}$$

$$\rho h = \alpha\rho_V h_V + (1-\alpha)\rho_L h_L. \tag{6}$$

To close the system, an EOS and a thermal relation are necessary to link the pressure and the temperature to the thermodynamic variables. Pure phases follow the stiffened gas EOS and a barotropic law is considered for the mixture.

2.1.1. The pure-phase EOS—low Mach number preconditioning

For pure phases, we use the convex stiffened gas EOS (see [35]):

$$P(\rho, e) = (\gamma - 1)\rho(e - q) - \gamma P_\infty \tag{7}$$

$$P(\rho, T) = \rho(\gamma - 1)C_v T - P_\infty \tag{8}$$

$$T(\rho, h) = \frac{h - q}{C_p} \tag{9}$$

where $\gamma = C_p/C_v$ is the polytropic coefficient, C_p and C_v are thermal capacities, q the energy of the fluid at a given reference state and P_∞ is a constant reference pressure.

A well-known problem of compressible codes concerns the stiffness on the solution convergence when the Mach number becomes low. In this situation, the dominance of convection terms renders the system stiff and compressible solvers converge slowly. To overcome this difficulty, a preconditioned method is necessary. The physical acoustic waves are replaced by pseudo-acoustic modes that are much closer to the advective velocity, reducing the stiffness and enhancing the convergence. The preconditioned method is based on the modification of the derivative term by a premultiplication with a suitable preconditioning matrix. With the primitive variables $W = (P, u, e)$, the preconditioned Euler equations can be expressed as

$$P_e^{-1}\frac{\partial W}{\partial t} + A_e\frac{\partial W}{\partial x} = 0. \tag{10}$$