Numerical Simulation of Cavitation with Level-Set in NSMB

C.-K. Huang\textsuperscript{a}, Y. Hoarau\textsuperscript{a}, E. Goncalvès\textsuperscript{b}

\textsuperscript{a} ICUBE - University of Strasbourg, 2 rue Boussingault, 67000 Strasbourg, ck.huang@etu.unistra.fr
\textsuperscript{b} Institute P’ - ENSMA, 86961 Futuroscope Chasseneuil, eric.goncalves@ensma.fr

Résumé :
Cette étude présente la mise en œuvre et la validation des modèles de cavitation développés au LEGI (Laboratoire des Écoulements Géophysiques et Industriels) dans le solveur NSMB (solveur compressible structuré multiblocks parallèle avec maillage chimère). En général, la cavitation se réfère à des poches de gaz apparaissant dans un écoulement fluide. En d’autres termes, il s’agit d’un phénomène diphasique avec changement de phase. La cavitation se produit lorsque la pression d’écoulement est inférieure à la pression de vapeur saturante. La prédiction numérique de la cavitation reste un défi pour plusieurs raisons. La modélisation du changement de phase (thermodynamique) et les interactions avec la turbulence n’est pas encore totalement établie. Du point de vue de la modélisation, la grande majorité des codes dédiés à la simulation de la cavitation est basée sur une approche moyennée à la fois pour l’écoulement diphasique et la turbulence. Une hiérarchie de modèles existe, du modèle simple à trois modèles d’équations (un fluide ou modèle homogène) jusqu’au modèle à sept équations (deux fluides) qui restent plus adaptés pour des géométries simples ou des fluides nonvisqueux. La méthodologie Level-Set est une technique numérique pour capturer des interfaces mobiles. Elle est basée sur une fonction de distance signée à savoir la fonction Level-Set pour représenter l’interface. L’interface est définie par le contour de zéro d’une fonction de la distance signée continu. Elle est positive sur l’extérieur, négative à l’intérieur et nulle sur la limite. Dans cette étude, la modélisation de la cavitation est couplée à la technique Level-Set. Des simulations numériques sont effectuées sur des profils NACA, dans un venturi et dans un tube à choc.

Abstract :
This study presents implementation and validation of the cavitation models developed at the LEGI (Laboratoire des Écoulements Géophysiques et Industriels) in the NSMB solver (structured parallel multiblocks compressible solver with chimera grid). Generally, cavitation refers to gaseous cavities that appear in the liquid flow. That is to say, it is a phenomenon of two-phase flows with phase transition. Cavitation occurs when the flow pressure is less than the vapor pressure of the flow. Numerical prediction of cavitation remains a challenge for several reasons. The modeling of phase transition (thermodynamics) and the interactions with the turbulence is not fully established yet. From the point of view of modeling, the vast majority of computer codes dedicated to the simulation of cavitation is based on an averaged approach for both the two-phase flow and the turbulence. A hierarchy of models exists from simple model of three equation models (one-fluid or homogeneous) up to seven equation models (two-fluid) which remain more suited for inviscid and simple geometries. Level set methods is a numerical
technique for capturing moving interfaces. It is based on a level set function namely signed distance function to represent the interface. The interface is defined by the zero contour of a continuous signed distance function. It is positive on the exterior, negative on the interior and zero on the boundary. In this study the cavitation modeling is coupled to the Level-Set technique. Numerical simulations are performed on NACA foils, venturi and shock tube.

Key words : Cavitation, Level-Set, Chimera grids

1 Introduction

Cavitation is a phenomenon that occurs frequently in conventional hydraulic components such as pumps, valves, turbines and propellers. Over-speeds imposed by the local geometry, shear phenomena, acceleration or vibration may cause local pressure drops in the fluid. When the flow pressure is less than the vapor pressure of the fluid, there is a partial vaporization and vapor structures arises. The so formed structures are entrained by the flow and when they reach a higher pressure zone they condense and implode violently. Cavitation leads to significant loss of system performance, problems of instability of operation of machines and erosion of the component walls. It is thus a primary source of technical problems in the field of hydraulic turbomachinery, naval propulsion and space as well as in high pressure fuel injection.

In the development of a space launcher, cavitation is one of the most limiting factor generated by the hydraulic because it requires from the design phase the introduction of safety margins resulting primarily from an increase in pressure in the reservoirs. This increase in pressure requires an increase in the wall thickness which generates an increase in the structure. The magnitude of this increase in dry weight is 100 kg for 100 mbar of additional pressure, which corresponds about to 2% of the total weight of the largest telecommunications satellite built. Cavitation appears in the ergol turbo pumps of the launcher propellant and it generates falls of performances, instability of operation as well as mechanical loads on structures. The consequences can be tragic as the failure of the Japanese H-II launch vehicle in 1999. As for the shipbuilding industry, cavitation is one of the major constraints in the design of marine propellers. Noise, vibration, erosion as issues resulting of cavitation are very tricky. The appearance and disappearance of bubbles on the propeller blades create local pressure fluctuations that can be compared to shock waves because their violence. Moreover propeller produces a rotating flow in its wake. Sections of rudders that are placed behind the propeller are then in incidence and can cavitate violently at high speed. Cavitation is also very energetic and very noisy in the audible range. Depending on the type of cavitation frequencies and very specific signatures appear. This type of nuisance is obviously crucial for military vessels, as brought up to 100 km offshore by poorly controlled cavitation. The determination of cavitation instabilities regime is essential. In the hydraulic energy field, cavitation is a phenomenon limiting in the design phase of hydraulic machinery (pumps, turbines) and its consequences in terms of erosion of the walls are a very important nuisance (operating range and duration component life). Damage to solid walls is caused by very short pressure spikes (10ns to 1µs), high amplitude (∼ 1GPa), attributed to the impact of pressure waves emitted during the collapse of vapor structures. Knowledge of the dynamics of pockets is therefore very important. Also operating machinery instabilities related to the hydrodynamic coupling between the inter-blade channels are observed.
2 Numerical aspects

Numerical prediction of cavitation remains a challenge for several reasons. First the modeling of phase transition (thermodynamics) and the interactions with the turbulence is not fully established. Specific issues to numerical techniques in this type of flow also persist. On the issue of numerical architecture (compressible or incompressible low Mach preconditioning extended to variable densities), the question remains open. However, several studies have shown better capture reentraining jet of cavitation bubbles by compressible codes[1]-[4]. From the point of view of modeling, the vast majority of computer codes dedicated to the simulation of cavitation is based on an averaged approach for both the two-phase flow [5] and the turbulence. A hierarchy of models exist: simple model of three equation (one-fluid) models up to seven equation models (two-fluid) which remain more suited for inviscid and simple geometries [6]-[9]. Because of the difficulty of modeling nonequilibrium thermodynamics pattern during a phase transition, the existing models have systematic use of mechanical equilibrium assumptions (single pressure model) and thermal (single temperature model). The calculation of the void fraction by additional transport equation is increasingly used. In this case, the term of mass transfer between phases must be explicit. Several empirical formulations have been proposed [10]-[13] but still suffer from a calibration problem and thermodynamics inconsistency [14]. A new formulation is being tested in LEGI [15] and implemented/tested in this study. On the other hand, the turbulence-cavitation interaction is an under-known and documented phenomenon (due in particular to the difficulty of performing experimental measurements in cavitating flows). Compressibility effects on turbulence and the effects of the dispersed phase are also unknown. The usual codes are formulated in a Reynolds averaged to (or Favre) tensor turbulent closure model by a transport equation \( k - \varepsilon \) (Boussinesq hypothesis). These turbulence models are inadequate to correctly predict the dynamics of cavitation bubbles. Several solutions have been proposed and tested to reduce the eddy viscosity and improve the behavior of turbulence models. These corrections have shown some success, but do not take into account the dynamics of small scales [16]-[18].

In this study, several cavitation modeling are implemented in the NSMB solver in relation with the Level-Set technique. The models are validated on the shock tube, the venturi case and NACA12 foils.

The Navier-Stokes Multi-Block (NSMB) solver is a numerical software of NSMB consortium solving the finite volume Navier-Stokes equations (Vos et al., 1998). NSMB is a multi-block structured solver and parallelized able to solve the steady or unsteady Navier-Stokes equations in their compressible or incompressible version.

References


