



JASS 2009 - Joint Advanced Student School, Saint Petersburg, 29. 03. - 07. 04. 2009
Modelling and Simulation in Multidisciplinary Engineering

Modelling and Computation of Compressible Liquid Flows with Phase Transition

Mihatsch M.
Technische Universität München,
Lehrstuhl für Fluidmechanik – Fachgebiet Gasdynamik,
D-85747 Garching Germany

Content

- 1 Introduction
- 2 Modelling
- 3 Numerical Results and Validation

Abstract

This paper gives a short overview about the modeling and simulation of compressible liquid flows with phase transition: flows with cavitation. First the difficulties of simulating cavitating liquid flows are explained and a modelling approach is presented. Subsequent the resulting numerical approach **CATUM** (**C**avitation **T**echnische **U**niversität **M**ünchen) is validated by comparison of experimentally and numerically obtained results of an analysed 3-D cavitating liquid flow including a investigation about cavitation erosion.

Introduction

The significance of understanding cavitating flows is undoubtedly related to its occurrence in various technical applications, such as hydraulic machinery and fuel injection systems, where due to the operating conditions cavitation is hardly avoidable [3-5]. In these applications cavitation can lead to unwanted effects like the drop of efficiency or even more dangerous ones like vibration or erosion. Instantaneous loads, caused by the collapse-like recondensation of cavitation patterns, are one of the driving mechanisms of cavitation erosion. In order to highlight one basic erosion mechanism Fig. 1 shows a sketch of a

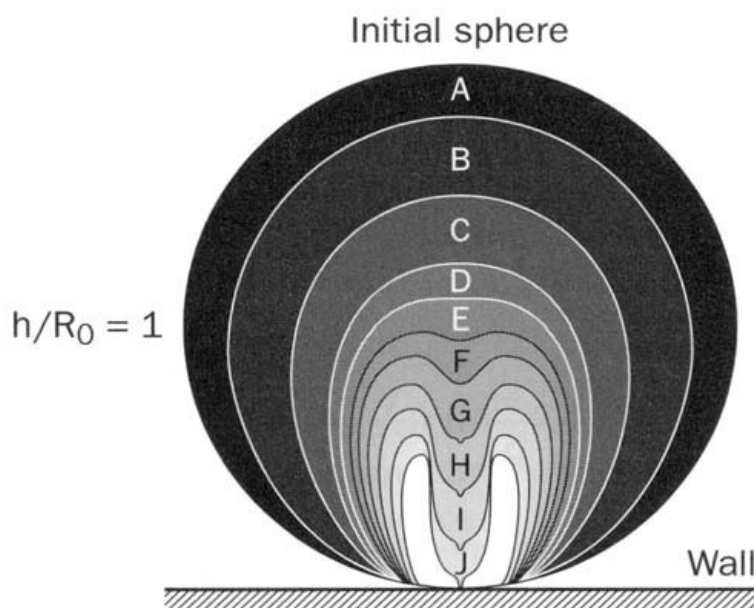


Figure 1: Sketch of the collapse mechanism and the jet formation of a cavitation bubble [6]

collapsing bubble at a wall [6]. The letters A to J correspond to subsequent instants in time but the time increments are strongly non-uniform. From a spherical collapse at the beginning until step C, a vertical flow direction becomes dominant starting around step E. The collapsing bubble deforms due to a micro jet, which is directed towards the wall. The impingement of the highly accelerated water jet at the wall (step J) produces pressure loads that are strong enough to cause erosion. This mechanism is directly related to the 'water hammer' or 'Joukowsky-shock'.

Modelling

Both, modelling and the simulation of cavitation are rather challenging due to the huge variations of the density and of the speed of sound (strong nonlinear behaviour!). The coexistence of vapour and liquid within the same flow field implies density ratios of the order of 10^5 . In saturated mixtures, the speed of sound drops even below the values of the pure phases. In a saturated mixture of water and vapour the speed of sound drops to the order of 1 m/s, where it is approximately 1500 m/s in pure water. Thus the Mach number varies from values near to 0 in pure water to values of the magnitude of 10 in cavitating two phase areas within the same global flow field. Furthermore, the time steps of the simulation are linked to the smallest cell size and to the fastest signal speed. This is due to the restriction that no information may travel through more than one cell during one timestep. That means that the smallest cellsize within the computational grid divided by the speed of sound of water defines the biggest time step that may be used in the simulation. With respect to the required resolution to resolve cavitation structures the time step drops to the order of nanoseconds.

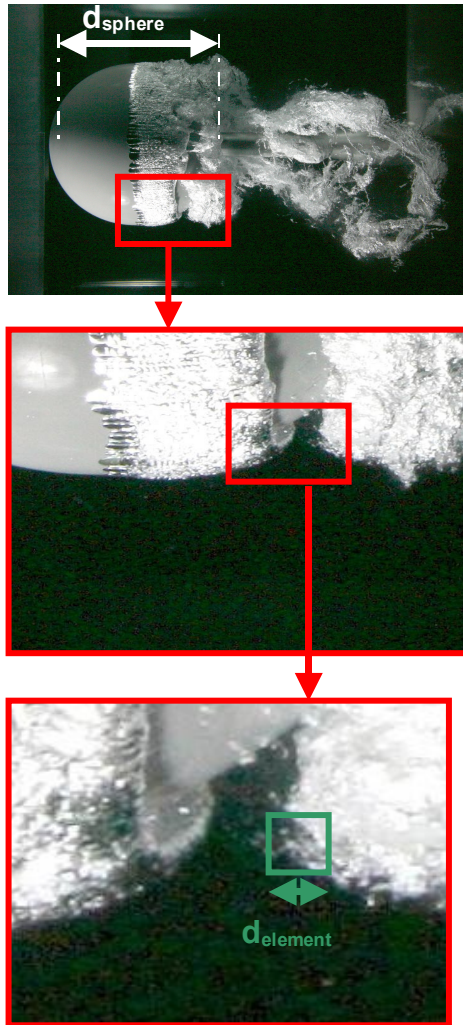


Figure 2: Cavitating flow around a sphere (top) and two zooms (middle, bottom) [7]

Figure 2 shows in the first picture the experimental setup of a cavitation flow around a sphere. In this picture three different length scales are shown: the diameter of the sphere ($d_{\text{sphere}}=1,5 \cdot 10^{-1}\text{m}$), the typical size of a finite volume, that would be used for numerical simulation of this setup ($d_{\text{element}}\approx 5 \cdot 10^{-3}\text{m}$) and the different sizes of cavitation pattern in the flowfield (ranging from 10^{-3}m to 10^{-5}m). As the single bubbles of the shown cavitation cloud structures are of some orders of magnitude smaller than the size of the numerical grid, single bubble dynamics can't be resolved. Instead the general physical approach to fulfill the integral conservation of mass, momentum, and energy is taken. The modelling of the two pure phases is achieved by the Tait-EOS for pure liquids and the ideal gas law for pure vapour. The two phase flow is modelled by a substitute fluid defined by the properties of the saturated single phases weighted by the void fraction. Due to the dominance of inertia effects within the considered two phase flows we neglect viscous effects and express the conservation principles by the Euler equations. However, the inclusion of dissipative mechanisms into the model is possible without restrictions.

Numerical Results and Validation

These presumptions are justified by comparing a simulation result with an experiment. Therefore we model and discretize an experimental setup consisting of a rectangular test section, where a prismatic body is located at the bottom wall. The mesh consists of $3 \cdot 10^6$ finite volumes and the simulation requires a computational time of 240 hours using 64 processors for 10^6 time steps with a step size of $2.9 \cdot 10^{-7}$ s. That leads to a physical simulation time of 0.29 s. As shown in Fig. 3, the distribution of the simulated vapor volume fractions (void) match the experiment in detail.

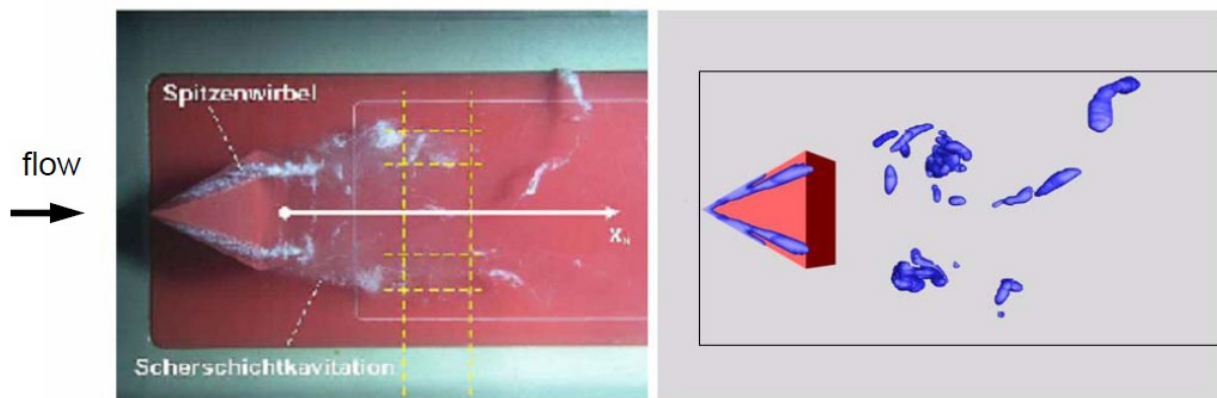


Figure 3: Top view of the prismatic body and the occurring cavitation structures - comparison of the experiment (left) to the numerical result (right)

The right picture is a time instant of the simulation, where the cavitation patterns are displayed by blue iso-surfaces of void fraction $\alpha = 0.1\%$. On the left side a picture of the experimental observation is shown. We observe weakly time depended cavitating tip vortices at the top of the prismatic body, as well as highly unsteady cavitating vortices in the shear layer downstream. Furthermore, we detect the transition of the shape of cavitating patterns from compact clouds in the near wake to tube-like structures in the far wake. It can be seen that even manifold and complex shapes of the cavitation pattern are well predicted by the simulation.

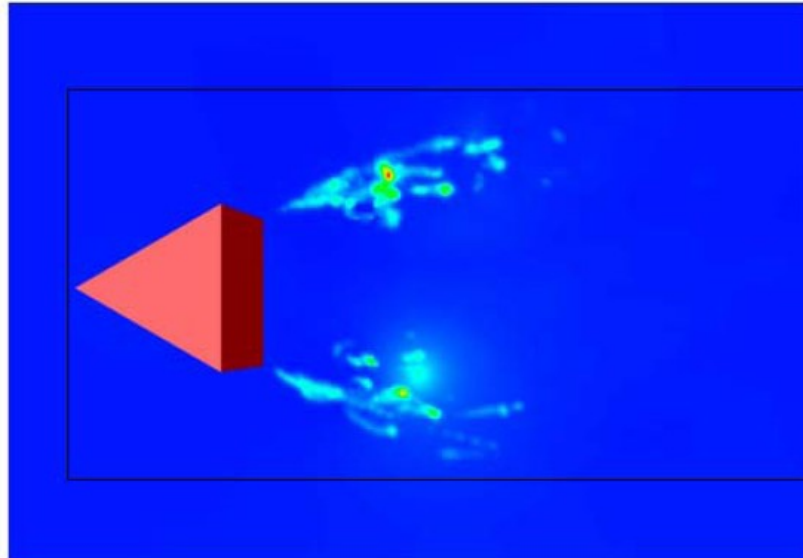


Figure 4: Top view of the prismatic body and the numerically obtained maximum pressure loads on the bottom wall, $p_{max}=70bar$

The areas indicated by yellow lines in the left picture of Fig. 3 are domains where intense erosion was experimentally observed. Figure 4 shows a top view of the simulation, where the highest pressures at the bottom wall over the whole simulation time are displayed with a maximum pressure $p_{max} = 70 \text{ bar}$. It shows that these areas match with the areas of intense erosion observed in the experiment. Thus we conclude that the CFD-Tool **CATUM** [8] is able to predict erosion sensitive areas within 3-D unsteady cavitating flows. With respect to the industrial relevance, the opportunity of predicting cavitation erosion enables design improvements of pumps or of ship propellers.



References (fundamentals of gas dynamics)

- [1] Schnerr, G.H., Lecture notes on 'Gasdynamik', Technische Universität München, 2007.
- [2] Schnerr, G.H., Lecture notes on 'Gasdynamische Strömungen mit Energiezufuhr und Phasenübergängen', Technische Universität München, 2007.

References (cavitation and CFD-Tool CATUM)

- [3] Schnerr, G.H., Sezal, I.H., Schmidt S.J., 'Numerical investigation of 3-D cloud cavitation with special emphasis on collapse induced shock dynamics', Physics of Fluids, vol. 20, issue 4, 2008.
- [4] Schmidt, S.J., Sezal I.H., Schnerr, G.H., Thalhamer M., 'Riemann Technique for the Simulation of Compressible Liquid Flows with Phase-transition at all Mach numbers – Shock and Wave Dynamics in Cavitating 3-D Micro and Macro Systems', 46th AIAA Aerospace Sciences Meeting and Exhibit, 7 - 10 January 2008, Reno, Nevada, USA, paper 2008-1238, 2008.
- [5] Schmidt, S.J., Sezal I.H., Schnerr, G.H., Thalhamer M., 'Shock waves as driving Mechfor Cavitation Erosion', Proc. 8th ISAIF – International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows, Lyon, France, July 2-6, 2007 (eds. Ottavy, X., Trebinjac, I.), paper ISAIF8-0044, pp. 931-940, 2007.
- [6] Plesset, M.S., Chapman, R.B., 'Collapse of an Initially Spherical Cavity in the Neighbourhood of a Solid Boundary', Journal of Fluid Mechanics, vol. 74, Issue 2, pp. 283-290, 1972.
- [7] Brandner, P. A., Walker, G. J., Niekamp, P. N. and Anderson, B., "An Investigation of Cloud Cavitation about a Sphere." In: 16th Australasian Fluid Mechanics Conference, 2 – 7 December 2007, Crown Placa, Gold Coast, Australia, 2007.
- [8] Further information about the CFD-Tool **CATUM**:<http://www.lhm.mw.tu-muenchen.de/gd/>