Dynamics Of Collapsing Bubbles Near Walls

Sigrid Andreae[1], Josef Ballmann[1], Siegfried Müller[2], and Alexander Voß[2]

[1] Lehr- und Forschungsgebiet für Mechanik der RWTH Aachen, Germany
sigrid.andreae@lufmech.rwth-aachen.de, ballmann@lufmech.rwth-aachen.de
[2] Institut für Geometrie und Praktische Mathematik, RWTH Aachen, Germany
mueller@igpm.rwth-aachen.de, voss@igpm.rwth-aachen.de

Summary. The influence of an adjacent wall on a collapsing bubble is investigated numerically. For this purpose, the two phase fluid is modeled by a single equation of state using the van der Waals equation of state and applying the Maxwell construction in the mixture region. Thereby the possibility of phase transition is included. The numerical simulations are performed using an adaptive finite volume scheme where the grid refinement strategy is based on multiresolution techniques.

1 Introduction

The formation and collapse of vapor bubbles in a liquid is called cavitation. Lord Rayleigh discovered that pressure waves emitted during the process of cavitation [8] may damage solids, e.g., marine screw propellers. However, up to now little is known about the mechanisms causing the damage of the solid.

Cavitation is induced by a pressure drop in the liquid below vapor pressure. Such a pressure decrease may occur due to local acceleration of the liquid flow caused by geometrical constraints, e.g., if the liquid flows through a narrow orifice or around an obstacle. In case, the pressure drops below vapor pressure, the liquid bursts and creates a free surface filled with gas and vapor – the bubble. Due to changes in the flow field, the pressure in the liquid may increase again causing the bubble to collapse. The collapse is accompanied by strong shock and rarefaction waves running into the bubble and the surrounding liquid. The shock wave focuses in the center of the bubble. This leads to extreme physical states in the interior. In addition, the shrinking of the bubble leads to a compression of the vapor. Both effects evoke an increase of pressure which bulges the bubble. Hereby, a dynamic oscillation process is initiated which finally leads to the collapse of the bubble. If the collapse takes place next to a solid, the pressure distribution becomes asymmetric and a liquid jet develops [7] which is either directed towards or away from the solid boundary. The direction of the jet depends on the elasticity of the solid and on a ratio determined by the initial distance between the boundary and the center of the bubble divided by the maximum extension of the bubble. This has been shown experimentally by Brujan et al. in [1], [2].

In order to investigate experimentally the dynamics of a bubble collapse, the bubble is produced by a laser pulse. Thereby the fluid is heated in the focus of the laser and forms a small, hot gas bubble at very high temperature. This experimental setup provides an exact positioning of the bubble. The processes taking place in the interior of the collapsing and oscillating bubble and the prediction of onset and extent of the cavitation damaging are still subject of theoretical and experimental research. However, small time and space scales as well as the complicated dynamics make an experimental approach difficult. Therefore numerical investigations are needed to reveal information about the wave dynamics in the fluid as well as the damaging of the solid. Of particular interest are pressure contours and velocity vectors in the liquid phase as requested in [2].

The primary objective of the present work is to provide an accurate prediction of all occuring wave phenomena. This concerns waves running into the liquid and the center of the bubble as well as the interaction of these waves with a neighboring solid. Normally, a collapsing gas bubble in a liquid is modeled as a two phase flow problem which means for each fluid a different equation of state is used. This can be done by using two meshes, whereby each mesh has to be updated in every step. Other techniques are tracking the boundary using a level-set or a volume-of-fluid method. Consequently, the phase boundary is modeled as a mathematically sharp boundary without a transition zone