Modeling Fluid-Structure Interaction in Cavitation Erosion: Preliminary Results

Yves Paquette$^1$, Christian Pellone$^1$, Marc Fivel$^2$, Giovanni Ghigliotti$^1$, Eric Johnsen$^3$, Jean-Pierre Franc$^1$

$^1$LEGI, Univ. Grenoble Alpes, CNRS, F-38000 Grenoble, France
$^2$SIMAP, Univ. Grenoble Alpes, CNRS, F-38000 Grenoble, France
$^3$Mechanical Engineering Department, University of Michigan, Ann Arbor, MI 48109, USA

E-mail: Yves.Paquette@legi.grenoble-inp.fr

Abstract. This paper is devoted to cavitation erosion modeling. It presents some recent numerical developments made in the code initially developed in collaboration with E. Johnsen and collaborators at University of Michigan [1] in order to account for fluid-structure interaction. The considered test case is that of a single air bubble collapsing near a wall due to an incident shock wave in the surrounding water. In our investigation, we focus on the code implementation and optimization and the bubble implosion mechanism. The paper is focused on the various events occurring during bubble collapse and the computation of the time evolution of the pressure distribution. The influence of the amplitude of the incident wave and the distance of the bubble to the wall are investigated.

1. Introduction

Cavitation phenomena and erosion prediction are not totally understood. Recent efforts have been made in order to compute as accurately as possible the collapse of a single bubble [1], [2] or of a cloud of bubbles that is supposed to generate cavitation erosion on neighboring walls [3]. Experimental results are also available and new developments in measurements techniques provide experimental data about single bubble dynamics and the pressure field due to bubble collapse [4], [5]. To improve the prediction technique, 2D simulations using a high-accuracy CFD code have been made, that are able to capture shock waves with high reliability in particular due to bubble implosion. These simulations are able to compute the wall load with great accuracy that will be passed to a solid mechanics solver to investigate the effect of the fluid / structure interaction on the cavitation erosion process. To achieve this goal, we have developed a 2D CFD code for a single bubble collapse jointly with University of Michigan. This code allows us to compute the pressure signal due to bubble collapse, that is presented and discussed in the present paper.

2. CFD Code FSI4CAV

The CFD two-dimensional code FSI4CAV (Fluid Structure Interaction for CAVitation) is developed jointly by University of Michigan [1] and LEGI and SIMAP Laboratories of Grenoble University. The main goal is to model a single cavitation bubble that collapses close to a wall including the shock waves generated during the collapse.
2.1. Physical Modeling

The code is based on four conservation equations [1]: mass, momentum (2 equations in 2D) and energy, completed by the Fourier law for heat transfer, the expression of the viscous stress and an equation of state for both gas and liquid [6]. To solve these equations, a fifth-order accurate finite volume WENO reconstruction is used for spatial discretization [7], and the HLL Riemann solver [8] is used to compute the fluxes at each cell interface for each conservation equation. Viscous stress and thermal conductivity are computed directly from the respective equations. Time marching is made with a fourth-order accurate Runge-Kutta scheme. In the initial version of the code [1], the mesh was cartesian with uniform spacing. Cell nodes could not be changed during simulation, which makes it challenging to account for the deformation of the wall. The code was modified in order to accommodate the deformation of the wall so that cell size is no longer constant. Then cells, and particularly cells close to the wall, can be deformed, and in our case this is due to the high pressure caused by the bubble collapse applied to the compliant wall. This space dependency required many modifications in the WENO process since all formulas will now depend on cell size.

The code simulates the dynamics of two fluids: water and air, without mass transfer and phase change. The same system of equations is solved in each cell for both water and air. The calculation starts with a single air bubble in water near a wall at atmospheric pressure. To make the bubble collapse, a shock wave is generated at infinity that moves towards the bubble and is reflected by the wall. A typical behavior of the bubble collapse is shown in figure 1.

![Figure 1. Bubble collapse during simulation. Above: pressure field, below: density.](image-url)

After being hit by the shock wave, the bubble starts collapsing around 275 ns. As expected, a micro-jet develops in direction of the wall. As a result, the bubble is split into two parts and the micro-jet hits the wall. In this paper, we will focus specially on the pressure increase on the wall due to the jet impact.
2.2. Pressure Signal

We consider here the case of a bubble of 50µm in initial radius, the bubble interface is at 50µm from the wall and there is a resolution of 0.25µm per cell. Figure 2 presents the time evolution of pressure on the wall, where the shock wave due to the bubble collapse hits the wall, compared to the time evolution of the maximum pressure in the whole domain.

![Figure 2. Pressure signal during bubble collapse.](image)

Three peaks are visible on the maximum pressure in the whole domain (blue curve). The first one is due to the incident wave. This wave hits first the top-right and bottom-right borders of the domain. At the top and the bottom of the domain, the incident wave crosses only water, while in the middle part of the domain, it crosses air where propagation speed is lower. That is why, top right and bottom right borders are hit first by the incident wave. The maximum pressure is twice the pressure of the incident wave because of cumulation of the incident and the reflected wave. There is also a pressure increase in the middle part of the right border (red signal) which is however attenuated by the crossing of the bubble. The second peak is related to the microjet that develops during the collapse of the bubble. Just before the bubble collapses, the microjet hits the most right interface of the bubble and generates a shock wave associated to the second pressure peak. This shock wave will further hit the wall and cause the third peak visible on both pressure signals in Figure 2. This last peak is the most interesting for coupling, it corresponds to the maximum load applied to the material during the bubble collapse.

2.3. Incident Shock Wave dependence

Various simulations were conducted in order to investigate the influence of (i) the magnitude of the incident pressure shock wave and (ii) the distance between the bubble right interface and the wall. The following results were obtained with a resolution of 0.50µm per cell.

(i) The pressure magnitude of the incident wave varies between 100 atm and 1200 atm. We focus on the collapse duration and the overpressure ratio between the maximum pressure on the wall and the pressure of the incident wave.
Table 1. Dependence on incident wave.

| Incident Wave (MPa) | 100  | 200  | 600  | 1200 |
|--------------------|------|------|------|------|
| Collapse Time (ns)  | 772  | 405  | 289  | 203  |
| Overpressure ratio  | 6.0  | 6.2  | 5.4  | 4.8  |

As expected, the collapse time decreases while the pressure of the incident wave increases. It is due to the fact that the velocity of the microjet increase with the amplitude of the incident wave. As for the overpressure ratio, Table 1 shows that this ratio decreases when the incident wave pressure increases. However this ratio is still in the same order of magnitude: between 5 and 6 times the incident pressure.

(ii) Then, considering an incident wave of 1200 atm, the distance of the bubble to the wall was changed between 1 bubble radius and 0. The closer the bubble is to the wall, the higher the impact pressure on the wall. This is partly due to the fact that the attenuation of the shock wave becomes more important when the interface moves away from the wall.

Table 2. Dependence on distance interface-wall.

| Distance interface-wall (bubble radius) | 0 | 0.25 | 0.5 | 1 |
|-----------------------------------------|---|------|-----|---|
| Pressure maximum on wall (MPa)          | 1430 | 1100 | 916 | 822 |

3. Conclusion and perspectives

The code 2D FSI4CAV was developed in order to model the fluid-structure interaction for cavitation erosion prediction purposes. This code allows us to compute the different events occurring during a single bubble collapse. It was shown that the maximum pressure in the whole domain presents three successive peaks. The first peak is due to the impact of the incident shock wave on the wall. The second one is due to the impact of the microjet on the bubble interface. It appears within the fluid. The shock wave that results from this impact propagates to the wall and generates a third peak when impacting the wall. The amplitude of this peak is significantly larger than the magnitude of the incident way and the ratio varies between 5 and 6 for the different cases investigated in this paper. The pressure distribution will now be passed to a solid code in order to model the wall deformation and an iterative procedure will be developed in order to update the computational domain after deformation.

Acknowledgments

This work has been partially supported by the LabEx Tec 21 (Investissements d’Avenir - grant agreement nANR-11-LABX-0030) and ONR grant N00014-12-1-0751.

References

[1] S. A. Beig, E. Johnsen, Maintaining interface equilibrium conditions in compressible multiphase flows using interface capturing 2015, submitted J. of Comp. Phys.
[2] E. Johnsen, T. Colonius, Numerical simulations of non-spherical bubble collapse 2009 J. Fluid. Mech. 629 231-262
[3] K.-H. Kim, G. Chahine, J.-P. Franc,A. Karimi, 2014 Advanced Experimental and Numerical Techniques for Cavitation Erosion Prediction,Vol. 106, Springer, Netherlands, 345-372
[4] Y.-C. Wang, Y.-W. Chen, Application of piezoelectric PVDF film to the measurement of impulsive forces generated by cavitation bubble collapse near a solid boundary 2007 Exp. Thermal and Fluid Science 32 403-414
[5] A. Jayaprakash, C.-H. Hsiao, G. Chahine, Numerical and Experimental Study of the Interaction of a Spark-Generated Bubble and a Vertical Wall 2012 J. of Fluids Eng. 134
[6] O. Le Métayer, J. Massoni, R. Saurel, Modelling evaporation fronts with reactive Riemann solvers 2005 J. of Comp. Phys. 205 567-610
[7] X. D. Liu, S. Osher, T. Chan, Weighted Essentially Non-Oscillatory Schemes 1994 J. of Comp. Phys. 115 200-212
[8] E.F. Toro, M. Spruce and W. Speares, Restoration of the contact surface in the HLL-Riemann solver 1994 Shock Waves 4 25-34