CFD Simulation of Cavitation Bubble Collapse near a Rectangular Groove Wall

Yan Yu¹, Changguo Sun¹, Yazhe Liu¹, Yang Liu², Dongsheng Xia²*¹

¹Naval Architecture and Ocean Engineering College, Dalian Maritime University, 116026 Dalian, Liaoning, China
²Transportation Engineering College, Dalian Maritime University, 116026 Dalian, Liaoning, China

Corresponding author: xia@dlmu.edu.cn

Abstract. The collapse process of a vapour bubble near to a rectangular groove wall is numerically simulated based on the Volume of Fluid (VOF) method with considering bubble surface tension, liquid viscosity and gas compressibility. Under the conditions of different stand-off parameters, evolution of the bubble profile, the jet velocity and the shock pressure around the bubble are obtained. Variation of the maximum jet velocity and the shock pressure around the bubble with the stand-off parameters is analyzed. Moreover, the impact pressure on the groove wall induced by the jet and the shock wave is also discussed, respectively. Cavitation erosion of the groove wall is explored for different stand-off parameters.

1. Introduction
Cavitation as a complex hydrodynamic phenomenon has been a major concern in a variety of fields including hydraulic machinery [1], biomedical treatment [2-3], and ocean engineering. For the majority of the hydraulic machinery cases, cavitation usually resulting in vibration, noise, increase of hydrodynamic drag and damage of component surface is undesirable. On the other hand, cavitation could play a positive role in aspects such as degradation of organics [4], acceleration of chemical reactions [5], disruption of cells [5], and cleaning of surfaces [6]. For most of cavitation cases, the negative or positive effects of cavitation are closely associated with the destructive action of cavitation bubbles in the vicinity of a solid surface. Therefore, understanding the interaction between bubbles and walls is very important to inhibit or utilize cavitation.

Rayleigh firstly established the bubble dynamics equation for a bubble in an infinite liquid field [7], in which liquid viscosity and liquid compressibility was neglected. It is predicted that emission of shock waves as the bubble volume is compressed to the minimum at the final collapse stage. The shock wave propagates in the flow field, causing intense impulse pressure on the nearby wall. Mørch [8] reported the shock waves generated at the collapse of bubble clusters. Brujan [9] found that the shock wave pressure is 1.3±0.3GPa at a distance of 68μm from the bubble wall. The shockwave theory is recognized as one of main mechanisms for occurrence of cavitation erosion. In addition, the micro-jet theory is another important one, which was firstly proposed by Kornfeld and Suvorov [10] in 1944. It is believed that a bubble near a wall might behave an asymmetrical collapse characteristic, leading to generate a liquid jet towards to the wall. After the liquid jet penetrates the bubble, it finally impacts on the wall. The micro-jet theory was experimentally proved the earliest by Benjamin and Ellis in 1966.
[11]. A lot of experimental and numerical research achievements relevant to the micro-jet phenomenon have been reported up to now, which provides information enriching the micro-jet theory [12-14]. Although a great number of researches concerning dynamics of cavitation bubbles close to walls have been performed experimentally and numerically for decades, few researches focusing on the interaction between bubbles and walls with special profiles are reported. In this paper, collapse of a cavitation bubble near to a rectangular groove wall is numerically investigated by the homogeneous multiphase model and volume of fluid (VOF) method. It is anticipated to obtain valuable information understanding the effect of wall profile on collapse behavior of a bubble adjacent to the wall.

2. Numerical Method
The physical description of the problem discussed in this paper is the dynamics of a spherical vapor bubble close to a rigid boundary with the micrometer-scale rectangular groove structure. It is assumed that only vapor, not other gas such as non-condensable gas, is contained in the single bubble. The vapor inside the bubble is compressible and assumed to behave like an ideal gas. The liquid surrounding the bubble (e.g. water) is considered as an incompressible newtonian fluid. The fluid flow state is laminar. The viscosity of water and vapor is considered in simulation. The effect of gravity is neglected. Moreover, mass transfer between vapor and water is not taken into account in the paper.

2.1. Governing Equations
The volume of fluid (VOF) method based on homogeneous flow theory was adopted to simulate collapse of the cavitation bubble close to the rigid boundary with micrometre-scale groove structure. In the model, the flow is assumed as a homogenous mixture fluid consisting of liquid and vapor, neglecting inter-phase relative motion. The two components share the same physical velocity and pressure.

The continuity, momentum, and energy equations for the mixture flow are

\[
\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{v}) = 0
\] (1)

\[
\frac{\partial (\rho_m \mathbf{v})}{\partial t} + \nabla \cdot (\rho_m \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot \left[ \mu_m (\nabla \mathbf{v} + \nabla \mathbf{v}^T) - \frac{2}{3} \mu_m (\nabla \cdot \mathbf{v}) I \right] + F_s
\] (2)

\[
\frac{\partial (\rho_m E)}{\partial t} + \nabla \left[ (\rho_m E + p) \right] = \nabla \cdot (k \nabla T)
\] (3)

where \( p \) is pressure, \( \mathbf{v} \) is velocity vector, \( \rho_m \) and \( \mu_m \) denotes the density and dynamic viscosity of the mixture, respectively, \( E \) is total energy, \( T \) is temperature and \( k \) is thermal conductivity. In equation (2), \( I \) represents the identity matrix.

The surface tension effect of bubble-water interface is included with the continuum surface force (CSF) model[15], which appends the surface tension term \( F_s \) to the source term of the momentum equation (2) as a volume force.

\[
F_s = \sigma \frac{\rho_m k \nabla \alpha_l}{0.5(\rho_v + \rho_l)}
\] (4)

where \( \sigma \) is the surface tension, \( \kappa \) is the curvature of the interface, and \( \alpha_l \) is the volume fraction of the liquid phase in every cell.

The transportation equation of the volume fraction of liquid phase \( \alpha_l \) is given by

\[
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \mathbf{v}) = 0
\] (5)
As $\alpha_l=1$ or 0 for a grid cell at some time, it means that the grid cell is filled with pure liquid phase or pure gas phase. As $0<\alpha<1$, it represents that the interface between the two phases is located in this grid cell.

The density and the dynamic viscosity of the mixture are defined as

$$\rho_m = \alpha_l \rho_l + (1-\alpha_l) \rho_g$$

$$\mu_m = \alpha_l \mu_l + (1-\alpha_l) \mu_g$$

where the subscripts l and g correspond to liquid and vapor, respectively. The value of the liquid density is constant. Considering the compressibility of vapor, the density in the gas phase, $\rho_g$, is calculated by the ideal gas state equation.

$$\rho_g = \frac{M_m}{R_g T} p$$

where $R_g$ and $M_m$ represent the universal gas constant and the gas molar mass, respectively.

2.2. Computational Domain, Mesh Generation and Boundary Conditions

To decrease computational cost, a 2D axisymmetric model is applied in the numerical simulation because the collapse process has the axial symmetric characteristics. The geometry of the computational model and the boundary condition is shown in figure 1. The rectangular groove shape is the commonly adopted one in a variety of researches concerned with configuration effect. The topology characteristics of rectangular groove wall may be represented by three geometric parameters, i.e. the width $w$, depth $d$ and interval $s$ of groove unit. In this paper, a distributed rectangular groove is arranged on the bottom rigid boundary. The width $w$, depth $d$ and interval $s$ of rectangular groove unit are designed to be 40 μm ($w=d=s=40\mu m$), respectively. The rectangular groove boundary is set as wall with no-slip boundary condition. The left boundary is set as axisymmetric boundary. The pre-existing bubble is placed along the axisymmetric axis at a distance from the solid wall. Considering the profile characteristics of rectangular groove wall, two non-dimensional stand-off parameters $\gamma_1$ and $\gamma_2$ are defined as

$$\gamma_1 = h_1/r_{max}$$

$$\gamma_2 = h_2/r_{max}$$

where $h_1$ and $h_2$ are the distance of the initial bubble center to the wall and the bottom of groove, respectively, and $r_{max}$ is the maximum bubble radius.

Both of the top and right boundaries are set as pressure outlet with the value being equal to 101kPa. The initial radius of cavitation bubble denoted by $r_0$ is chosen as 0.1mm. The initial pressure in the bubble is set as the saturated vapor pressure of 4247Pa at the ambient temperature of 300K. To decrease the effect of pressure boundary on the computational result, the distance between any two opposite boundaries is designed to be 150 times as the initial size of bubble.

![Figure 1. Computational model.](image-url)
The whole domain is meshed with the structured quadrilateral cells by using ICEM CFD. The zone surrounding the bubble (I zone) is finely meshed in order to capture the details of the dynamics for bubble collapse. The minimum cell size in I zone is about 5×5μm². The total cell number of the whole domain is more than 240 thousand.

2.3. Discretization Scheme and Solution Algorithm

The finite volume method is adopted to discretize the governing equations. The PISO algorithm is employed for the pressure-velocity coupling in unsteady computation. The gradient term is discretized with the least squares cell based scheme. The liquid volume fraction at a cell interface is computed using Geo-Reconstruct scheme. The density term, the momentum term and the energy term are discretized by the second-order upwind scheme, respectively. The pressure term is calculated by the body-force-weighted scheme. The temporal term is discretized with the first-order implicit scheme. The time step was carefully adjusted between 10⁻⁹s and 10⁻¹⁰s during calculation to guarantee convergence.

3. Results and Discussion

In this paper, five cases with different stand-off parameters γ₁ and γ₂ as listed in Table 1 are investigated. Figure 2(a) depicts evolution of the bubble shape during collapse for γ₁=2.0 and γ₂=4.0. Initially, the bubble volume is compressed continuously due to pressure difference between the inside and outside of bubble. As t=9.96μs, the top part of bubble-liquid interface is completely flat and a high pressure zone with the maximum pressure of 1.41MPa exists above the bubble (figure 2(b)). Subsequently, liquid starts to penetrate inside the bubble forming a jet and the bubble takes on a heart-like shape (figure 2(a)). The jet passes through the bubble with the maximum jet velocity of 240m/s at t=10.2326μs, which results in forming a toroidal bubble (figure 2(c)). Then the toroidal bubble performs oscillation. A shock pressure of 5.23GPa is found around the bubble at t=10.2468μs (figure 2(d)). The groove wall below the bubble is subjected to an intensive impact pressure of 230MPa.

After the jet passes through the bubble, a high pressure appears ahead of the jet during the jet moves. The high pressure ahead of the jet continuously decreases as the jet velocity decays. At t=16.400μs, the jet starts to enter the rectangular groove with the jet velocity of 40m/s. As the jet moves downwards, the liquid fluid inside the groove flows outwards along the wall boundary, which leads to form two opposite-sense vortexes inside and outside the groove, respectively. At t=19-20μs, the jet reaches the groove bottom wall with the speed of 20-24m/s (figure 2(e)). The groove wall is subjected to a water-hammer pressure of 30MPa.

### Table 1. Simulation conditions.

| Case | Stand-off Parameter | Graphical Depiction |
|------|--------------------|---------------------|
| 1    | γ₁=1.0, γ₂=1.4     | ![Image](image1.png) |
| 2    | γ₁=1.1, γ₂=1.5     | ![Image](image2.png) |
| 3    | γ₁=1.5, γ₂=1.9     | ![Image](image3.png) |
| 4    | γ₁=2.0, γ₂=2.4     | ![Image](image4.png) |
| 5    | γ₁=3.0, γ₂=3.4     | ![Image](image5.png) |
rebounds a behavior oscillation. The high pressure of 104MPa is formed around the bubble when the bubble and the jet velocity reaches the maximum value of 130m/s (figure 3(c)). Finally, the toroidal bubble maximum pressure of 0.89MPa appears above the bubble (figure 3(b)). Then a jet starts to penetrate the wall. At *t* between the bubble and wall is impeded (figure 3(a)). The bubble center continuously interface is significantly inhibited comparing with one of the upper part because motion of the liquid 2 collapsing bubble at (a)

Figure 2. (a) Simulated bubble profile evolution; (b–e) Pressure field and velocity vector field around

For the case of *γ*_1=1.1 and *γ*_2=1.5, initially the deformation of the bottom part of bubble-liquid interface is significantly inhibited comparing with one of the upper part because motion of the liquid between the bubble and wall is impeded (figure 3(a)). The bubble center continuously approaches to the wall. At *t*=10.16µs, the top of bubble becomes flat meanwhile a high pressure zone with the maximum pressure of 0.89MPa appears above the bubble (figure 3(b)). Then a jet starts to penetrate inside the bubble. When the jet passes through the bubble at *t*=10.96µs, a toroidal bubble is formed and the jet velocity reaches the maximum value of 130m/s (figure 3(c)). Finally, the toroidal bubble behaves oscillation. The high pressure of 104MPa is formed around the bubble when the bubble rebounds at *t*=11.1244µs (figure 3(d)). The groove wall below the bubble is subjected to the impact pressure of 46MPa. As the jet reaches the groove wall at *t*=12.49µs, the jet velocity decays to be about 60m/s, which leads to a water-hammer pressure of 90MPa on the groove wall.

Figure 3. (a) Simulated bubble profile evolution; (b–d) Pressure field and velocity vector field around collapsing bubble at *t*=10.16µs, *t*=10.96µs and *t*=11.1244µs, respectively, for *γ*_1=1.1 and *γ*_2=1.5. (the black solid line is the bubble-liquid interface).
As the stand-off parameters $\gamma_1$ and $\gamma_2$ are further decreased to be 1.0 and 1.4, respectively, the bottom of bubble is very close to the top edge of groove wall at the initial condition as shown in Figure 4(a). At the beginning, the lower part of bubble contacts with the wall, which means that the bubble is absorbed to the wall. At $t=10.16\mu s$, the top of bubble becomes flat meanwhile a high pressure zone with the maximum pressure of 0.86MPa exists above the bubble (figure 4(b)). The lower part of bubble-liquid interface takes on a convex upward shape. Subsequently, a jet starts to penetrate through the bubble and the bubble appears to be the shape of saddle. At $t=11.01\mu s$, the jet passes through the bottom of bubble-liquid interface with the jet velocity of 135m/s (figure 4(c)). Compared to one in other cases, the jet is relatively wider. It is because the bubble is adsorbed to the wall, which makes the bottom of bubble keep a relatively large size in the jet forming process. Then an attached toroidal bubble is formed and continuously compressed. Finally, the attached bubble detaches from the wall to become a free one and behaves oscillation. It is observed that a pressure of 68.6MPa is formed around the bubble as the bubble rebounds at $t=11.2857\mu s$, which leads to the pressure impact of 46MPa on the groove wall (figure 4(d)). As the jet reaches the groove wall at $t=12.608\mu s$, the jet velocity decays to be 45-50m/s. It means that the groove wall is subjected to a water-hammer pressure of 67.5-75.0MPa.

![Figure 4](image_url)

**Figure 4.** (a) Simulated bubble profile evolution; (b–d) Pressure field and velocity vector field around collapsing bubble at $t=10.16\mu s$, $t=11.01\mu s$ and $t=11.2857\mu s$, respectively, for $\gamma_1=1.0$ and $\gamma_2=1.4$. (the black solid line is the bubble-liquid interface).

![Figure 5](image_url)

**Figure 5.** (a) The bubble collapse pressure and the maximum jet velocity, and (b) The impact pressure induced by shock wave and the water hammer pressure induced by jet on the groove bottom wall for different $\gamma_1$ and $\gamma_2$.

Figure 5(a) presents variation of the bubble collapse pressure and the maximum jet velocity with $\gamma_1$ and $\gamma_2$. It is indicated from figure 5(a) that the maximum jet velocity is promoted with increasing the stand-off parameters for $1.0<\gamma_1<3.0$ and $1.4<\gamma_2<3.4$. It is because that the wall effects on motion of the liquid below the bubble is weakened with increasing the distance between the bubble and the wall, which causes that the bubble tends to collapse symmetrically and the volume of bubble is
compressed more before the jet is formed. The bubble collapse pressure is enhanced with increasing the stand-off parameters for $1.0 < \gamma_1 < 2.0$ and $1.4 < \gamma_2 < 2.4$. However, the bubble collapse pressure is found not to grow as $\gamma_1$ and $\gamma_2$ are further increased to be 3.0 and 3.4. Figure 5(b) shows variation of the impact pressure induced by shock wave and the water hammer pressure induced by jet on the groove bottom wall with $\gamma_1$ and $\gamma_2$. Even though the impact pressure on the groove wall induced by jet is higher than one induced by shock wave for $1.0 < \gamma_1 < 1.1$ and $1.4 < \gamma_2 < 1.5$, the shock wave still has a noticeable effect on the groove wall in this condition. Therefore, the cavitation erosion of groove wall should be resulted from both of jet and shock wave in the cases of 1 and 2. Moreover, the impact pressure on the groove wall induced by shock wave is considerably enhanced meanwhile one by jet is decreased as $\gamma_1$ and $\gamma_2$ are increased to be 1.5 and 1.9, respectively. For $1.5 < \gamma_1 < 3.0$ and $1.9 < \gamma_2 < 3.4$, the impact pressure induced by shock wave is far greater than one induced by jet on the groove wall. Therefore, the shock wave is mainly responsible for the cavitation erosion of groove wall for $1.5 < \gamma_1 < 3.0$ and $1.9 < \gamma_2 < 3.4$.

4. Conclusion
The dynamics of a micrometer-sized vapor bubble close to a rectangular groove wall is numerically simulated based on the volume of fluid method. Evolution of the bubble profile presents the distinctly different characteristics with changing the stand-off parameters $\gamma_1$ and $\gamma_2$. The maximum jet velocity is promoted with increasing $\gamma_1$ and $\gamma_2$. The bubble collapse pressure is enhanced with increasing $\gamma_1$ and $\gamma_2$ for $1.0 < \gamma_1 < 2.0$ and $1.4 < \gamma_2 < 2.4$. However, the bubble collapse pressure is not to grow as $\gamma_1$ and $\gamma_2$ are further increased. The cavitation erosion of groove wall should be caused by both of jet and shock wave for $1.0 < \gamma_1 < 1.1$ and $1.4 < \gamma_2 < 1.5$. The shock wave is mainly responsible for the cavitation erosion of groove wall for $1.5 < \gamma_1 < 3.0$ and $1.9 < \gamma_2 < 3.4$.

Acknowledgments
This work was financially supported by Natural Science Foundation of Liaoning Province of China (2015020136 and 201602090) and the Fundamental Research Funds for the Central Universities (3132019162).

References
[1] D. Suo, Z. Jin, X. Jiang, P.A. Dayton, Y. Jing, Appl. Phys. Lett., 110, 023703, (2017)
[2] C. G. Duan, V. Y. Karelin, Abrasive Erosion and Corrosion of Hydraulic Machinery. London: Imperial College Press, (2003)
[3] Y. Zhang, Y. Zhang, Y. L. Wu, Pro. Inst. Mech. Eng., Part C: J. Mech. Eng. Sci., 231, 1181-1204, (2017)
[4] R. Balachandran, Z. Patterson, P. Deymier, S. A. Snyder, M. Keswani, Chemosphere, 147, 52-59, (2016)
[5] K. Nakajima, D. Nishioka, M. Hirao, M. So, Y. Goto, H. Ogi, Ultrason. Sonochem., 36, 206-211, (2017)
[6] G. L. Chahine, A. Kapahi, J. K. Choi, C. T. Hsiao, Ultrason. Sonochem., 29, 528-549, (2016)
[7] L. Rayleigh, Philos. Mag. 34, 94-98, (1917)
[8] K. A. Morch, Erosion. London: Academic Press, (1979)
[9] E. A. Brujan, T. Ikeda, Y. Matsumoto, Exp. Therm. Fluid Sci., 32, 1188-1191, (2008)
[10] M. Kornfeld, L. Suvorov, J. Appl. Phy., 15, 495-506, (1944)
[11] T. B. Benjamin, A. T. Ellis, Philos. Trans. R. Soc. London A Math. Phys. Eng. Sci., 260, 221-240, (1966)
[12] M. S. Plesset, R. B. Chapman, J. Fluid Mech., 47, 283-290, (1971).
[13] W. Lauterborn, H. Bolle, J. Fluid Mech., 72, 391-399, (1975)
[14] N. K. Bourne, J. E. Field, J. Fluid Mech., 244, 225-240, (1992)
[15] J. U. Brackbill, D. B. Kothe, C. Zemach, J. Comput. Phys., 100, 335-354, (1992)