Experimental and computational capture of high frequency oscillation induced by inception cavitation in submerged shear layer

Yuchuan Wang¹, Lei Tan², Xinyang Liu³, Diyi Chen¹*

1. College of Water Resources and Architectural Engineering, Northwest A&F University, Yangling, Shaanxi, China;
2. State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing, China
3. School of Water Conservancy, North China University of Water Resources and Electric Power, Zhengzhou, Henan, China

wyc@nwafu.edu.cn, diyichen@nwsuaf.edu.cn

Abstract. Cavitation is a peculiar phenomenon in the liquid medium flow, which is the phenomenon of that the local pressure in the flow field is lower than the local vaporization pressure. It's well-known that bubble collapse will cause high pressure oscillation in flow, which is the fundamental of the cavitation. In present study, 200 kHz high frequency pressure transducer is used to capture the pressure oscillation induced by inception cavitating bubble in submerged shear layer of axis-symmetric cavity. Consistent with boundary conditions in experimental measurement, large eddy simulation (LES) and Schnerr-Sauer (S-S) cavitation model are employed to simulate inception cavitation flow at condition Re=2×10⁵ and σ=0.59. The computational results show good agreement with experimental results. The cavitating bubble is slender, just like “rice grain shape”, and the collapse of bubble induces the instantaneous and significantly pressure fluctuation in cavity.

1. Introduction
Cavitation is a fundamental problem in fluids science and engineering. The general recognition about cavitation is that, when local pressure is lower than vapor pressure in flow field, liquid phase transform into vapor phase and form bubbles in liquid. These bubbles collapse and induce high frequency pressure fluctuation [1] as they move to high pressure region in flow field. In present article, we carried out experimental and computational study to capture high frequency pressure fluctuation of cavitation inception in submerged shear layer.

Numerous studies [2, 3] had investigated the cavitating jet flow. Two kinds can be summarized: one is cavitating cloud shedding from sharp edge orifice [4-7], in which relationship between pressure fluctuation and cavitation cloud shape is studied; another is to investigate the vortex structure [8] associating with cavitation inception.

When water jet flow through an axis-symmetrical cavity, water fill with the cavity. Thus, the water jet is surrounded by stationary water, which provides a good chance to observe cavitation of submerged shear layer. The jet entrain water in cavity flowing through cavity, which will result in pressure decrease
in cavity. Because of abundant vortex in shear layer, where is lower pressure region, cavitation will firstly occur in vortex center of shear layer.

2. Experimental Setup
The test rig is an open horizontal loop, shown in figure 1, which consists of water supply pump 8, pipe booster pump 7, electromagnetic flow meter 6, axis-symmetrical cavity, high frequency dynamic pressure transducer 1, data acquisition and video camera. The discharge rate and working pressure $p_0$ are controlled by valve 5, which means the variation of velocity of working jet $u_1$. The maximum value of $u_1$ can reach 50m/s. At every measurement condition, flow rate, working pressure, cavity pressures at different places are collected at the same time.

![Figure 1. Schematic diagram of experimental setup.](image)

Figure 1. Schematic diagram of experimental setup.

Figure 2 shows the structure of axis-symmetrical cavity, which is made of polymethyl methacrylate (PMMA), thus flow patterns can be observed inside cavity. Five parts (inlet nozzle, axis-symmetrical cavity, pressure measurement hole, impinging body and outlet nozzle) are clamped by four stainless screw piles between two stainless disks.

![Figure 2. The structure of axis-symmetric cavity.](image)

Figure 2. The structure of axis-symmetric cavity.

The frequency of dynamic pressure transducer is 200KHz, that can totally capture 300ms pressure signal. To obtain smooth curve, 5 points adjacent-averaging method is used to process data.

3. Numerical Model and Methods

3.1. Numerical model
The cavity geometries for computation, shown in figure 3(a), is the same as the cavity employed in experiments, which structure parameters are $120^\circ$ impinging body, the outlet diameter of the first nozzle $d_1=11\text{mm}$, the outlet diameter of the second nozzle $d_2/d_1=1.55$, chamber diameter $d_c/d_1=10.91$ and
chamber length \( L_c/d_1 = 9.93 \). Figure 3 shows the computational domain and partial grids in rectangular block. The central part of chamber is refined at size 0.25mm to capture more flow details. The computational domain is meshed 2.25 million hexahedral structural grids, shown in figure 3(b).

3.2. S-S cavitation model
Because of abundant vortex structure in shear layer, large eddy simulation (LES) provided by commercial CFD code Fluent [9] is employed to simulate submerged jet flow in cavity. The S-S cavitation model [7,9] is used to calculate the mass evaporation and condensation rates in vapor transport equation, that is described as follows.

\[
\frac{\partial}{\partial t} (\alpha \rho_v) + \frac{\partial (\alpha \rho_v u)}{\partial x_i} = m_v - m_c \tag{1}
\]

Where \( \alpha \) is vapor volume fraction, \( \rho_v \) is vapor density, \( u \) is velocity, \( m_v \) is mass evaporation rate and \( m_c \) is mass condensation rate. They are calculated by equations (2) and (3).

\[
\dot{m}_v = \frac{\rho_v \rho}{\rho} \alpha (1 - \alpha) \frac{3}{R_e} \left( \frac{2}{3} \frac{p_v - p}{\rho_v} \right) \quad p_v \geq p \tag{2}
\]

\[
\dot{m}_c = \frac{\rho_v \rho}{\rho} \alpha (1 - \alpha) \frac{3}{R_e} \left( \frac{2}{3} \frac{p - p_v}{\rho_v} \right) \quad p \leq p_v \tag{3}
\]

3.3. Numerical methods
The finite volume method is used to discretize the computational domain. The diffusion term, convection term and transient term in governing equation are discretized by the second central differencing scheme, bounded central differencing scheme and the second implicit differencing scheme, respectively. The spacing term in vapor transport equation is discretized by QUICK scheme. The Coupled algorithm is used to couple pressure-velocity equations.

The condition of \( Re = u_1 d_1/\nu = 2 \times 10^5 \) and \( \sigma = 0.59 \) at measuring point is simulated. The inlet boundary of computational domain is set as pressure inlet. The total pressure is set as 276690 Pa and initial static pressure is set as 27040 Pa according to experimental results. The outlet boundary is set as pressure outlet,
the static pressure is given $98000\text{Pa}$. The time step is determined by the smallest grid scale and jet averaged velocity, namely $\Delta t = \Delta x / u_1$. Before the calculation, grid and time step independence are checked.

The averaged flow rate of measurement is $1.7405\text{ L/s}$, while computational averaged flow rate is $1.7327\text{ L/s}$. Therefore, the computational results are accurate enough to analyze the cavitating flow features.

4. Results and Analysis

4.1. Experimental results

Figure 4 and figure 5 show the time series of pressure curve of M1(figure 2) at cavitation inception stage and without cavitation condition in range of 10ms. The working condition of figure 4 is $Re=2.12\times10^5$, while it is $Re=1.72\times10^5$ in figure 5. The cavitation number is calculated by formula (4).

$$\sigma = \frac{p + p_{\text{atm}} - p_v}{0.5 \rho u_1^2}$$

(4)

where $p$ is measured pressure, $p_{\text{atm}}$ is atmosphere pressure in laboratory and $p_v = 3540\text{Pa}$ is vapor pressure.

Figure 4. The experimental time series curve of cavitation number at $Re=2.12\times10^5$.

Figure 5. The experimental time series curve of cavitation number at $Re=1.72 \times 10^5$. 
The oscillation appears at the moment of the smallest cavitation number, as shown in figure 4, which indicates that a cavitation event occurs. The curve in figure 5 is smooth and the values of cavitation number are larger than them in figure 4. The experimental result is the same as Gopalan’s [8] measurement.

4.2. Computational results
Figure 6 shows the computational time series curve of pressure at point M1 (figure 2) at condition \( \text{Re}=2\times10^5 \) in time range of 10ms. The sharp variations of pressure coefficient appear in curve, which indicate that the computation captures the high frequency pressure fluctuation induced by vapor bubble collapse. To investigate the relationship of pressure oscillation and vapor bubble shape, figure 7 illustrates the bubble shapes corresponding to pressure fluctuation in time range of \( t=3.0-3.7\text{ms} \). The bubbles are isolated by vapor volume fraction of 5%.

At the moment of \( t=3\text{ms} \), when the cavitation number reaches a certain low value, cavitation occurs in shear layer and generate a small bubble. The bubble grows with the cavitation number decreasing. At the moment of \( t=3.5\text{ms} \), the bubble volume reaches the maximum, while the cavitation number reaches minimum. Then, the bubble volume begins to reduce as the pressure increasing. The bubble collapses in time range of \( t=3.6-3.7\text{s} \), accompanying with cavitation number increasing rapidly. After bubble collapses \( (t=3.6\text{s}) \), the bubble can’t be isolated, and volume fraction reduces to nearly zero, as shown in figure 8. From figure 7, the bubble shape can be observed. It just likes a “grain rice”, not a spherical bubble, because the shear force deforms the bubble in shear layer.

![Figure 6](image)

**Figure 6.** The computational time series curve in time range 10ms at \( \text{Re}=2\times10^5 \).

Figure 8 shows the maximum vapor volume fraction variation in range of pressure oscillation \( (t=2.8-4.9\text{ms}) \). This depicts the bubble volume in quality. The processes of bubble grow and collapse can be observed in figure 8. The interval of bubble growing is about 0.5ms, while collapsing interval is about 0.2ms.

4.3. Discussions
Comparing figure 4 with figure 6, it can be found that the measured pressure fluctuation induced by vapor bubble collapse is far smaller than that is obtained by computation.

In fact, the pressure oscillation induced by cavitation is damping in propagation process. The pressure transducer is mounted on cavity wall, where is a distance away cavitation region. At the same time, we believe that the head structure with a small hole of pressure transducer also affect the pressure propagation. However, this pressure damping doesn’t consider in computation. This difference results in the discrepancy between measured pressure signal and computed pressure signal.
(a) Cavitation number curve in time range $t=2.5-5\text{ms}$.

(b) $t=3.0\text{ms}$

c) $t=3.0\text{ms}$

(d) $t=3.1\text{ms}$

e) $t=3.2\text{ms}$

(f) $t=3.3\text{ms}$

(g) $t=3.4\text{ms}$
Figure 7. The computational cavitation number curve (a) and enlarged computational bubble shape in shear layer at different moment (b-i).

Figure 8. The computational maximum vapor volume fraction curve in time range $t=2.5-5$ms

5. Conclusions
(1) Experimental measurements and computations can capture the pressure fluctuation induced by cavitation bubble collapse in submerged shear layer inside an axis-symmetrical cavity.
(2) The cavitation bubble is “rice grain” shape, because the shear force deforms it in shear layer.
(3) The amplitudes of pressure fluctuations obtained by computation and experiment show great discrepancy, because damping exists in experiments from cavitation location to transducer.

Acknowledgements
The research was supported by the Chinese National Foundation of Natural Science (Grant Nos. 51509209).

References
[1] Qin Z, Bremhorst K, Alehossein H, and Meyer T 2007 J. Fluid. Mech. 573 pp 1–25
[2] Ji B, Luo X, Wu Y, Peng X and Duan Y 2013 Int. J. Multiphas. Flow, 51 pp 33-43
[3] Huang B and Wang G 2011 Chinese Sci. Bull. 56 pp 1985-1992
[4] Peng G and Shimizu S 2013 J. Hydrodyn. 25 pp 502-509
[5] Soyama H 2005 ASME J. Fluids Eng. 127 pp 1095-1101
[6] Wright M M, Epps B, Dropkin A and Truscott T T 2013 Exp. Fluids 54 1541
[7] Yuan W and Schnerr G H 2003 ASME J. Fluids Eng. 125 pp 963-969
[8] Gopalan S, Katz J and Knio O 1999 J. Fluid Mech. 398 pp 1-43
[9] ANSYS FLUENT 12.0 2009 Theory Guide