Pressure Propagation of Impinging Jet with Cavitation by Numerical Analysis

To cite this article: Daisei Kanamori et al 2017 IOP Conf. Ser.: Mater. Sci. Eng. 249 012019

View the article online for updates and enhancements.
Pressure Propagation of Impinging Jet with Cavitation by Numerical Analysis

Daisei KANAMORI1, Fumihiro INOUE2, and Yutaka OHTA3

1 Graduate Student, Waseda University, Ookubo Shinjuku Tokyo, Japan, d.kanamori@ruri.waseda.jp
2 Department of Mechanical Engineering, Shonan Institute of Technology, Tsujidou, Fujisawa, Kanagawa, Japan
3 Department of Applied Mechanics and Aerospace Engineering, Waseda University, Ookubo Shinjuku Tokyo, Japan

Abstract. In recent years, cavitating jet has attracted attention as an application of water jet technology. In its application, it is important to clarify the jet flow structure and the behavior of bubble cloud collapse. Therefore, in order to visualize the cavitating jet flow structure and elucidate the behavior of collapsing of cavitation bubble clouds, we conducted numerical simulations with gas-liquid two-phase media model. We validated the numerical model by comparing the numerical results with the theoretical and experimental results and had a good agreement. In the case of gas-liquid two-phase free jet, cavitation bubble clouds emit periodically and transfer at a regular speed. And some bubble clouds merge with a preceding bubble clouds. Comparing with liquid single-phase jet, the core region is maintained to the further downstream and we show the usefulness of the cavitating jet. In the case of gas-liquid two-phase impinging jet, after a cavitation bubble cloud collides with wall, it is broken by applying pressure and generates a shock wave. At this time, the impact pressure becomes maximum. Thereafter, the shock wave affects other cavitation bubble clouds and break these. The collapsed cavitation bubble cloud rebounds and collapses again near the collision wall surface.

1. Introduction

Water jet in air or water is widely used in various situations such as cleaning, excavation, cutting, and destruction due to collision effect on the object [1, 2, 3, 4, 5, 6]. In recent years, cavitating jet in water has attracted attention as an application of water jet technology. Cavitation is capable of generating a very high impact pressure in the collapse of cavitation bubbles and has been applied to new fields such as decomposition of organic compounds and sterilization of sewage. In the case of cavitating jet, shear layer is generated near the nozzle outlet and velocity gradient changes abruptly between a high flow velocity region due to the high-pressure water and a surrounding fluid. And an unsteady vortex is generated in the shear layer. When the pressure inside the vortex decreases to the saturated vapor pressure of the fluid, vortex cavitation occurs. Vortex cavitation develops as it moves downstream, or it coalesces with other vortex cavitations to form a cavitation bubble cloud. It is known that the cavitation bubble cloud is periodically released and exhibits unsteady behavior.

In application of cavitating jet, it is important to clarify jet structure of jet and behavior of bubble cloud collapse. Until now many experimental researches have been conducted concerning the shape and size of the nozzle, the standoff distance, and the cavitation number etc. [7]. Other Studies have also been
made to elucidate the jet structure and breakdown of the cavitation bubble cloud using the Schlieren method, high-speed video camera, and PIV measurement [8, 9]. However, due to the influence of the cavitation bubble cloud, it is very difficult to observe its internal structure.

In this study, we conducted a numerical simulation with introducing a gas-liquid two-phase local homogeneous medium model which is effective for unsteady numerical analysis to visualize the jet structure of the cavitating jet and clarify the behavior of the bubble cloud collapse and to evaluate the impact pressure by the collision jet on the wall.

2. Computational Procedure

2.1. gas-liquid two-phase local homogeneous medium model
In this study, the gas-liquid two-phase local homogeneous medium model developed by Okuda et al. was used [10]. This model treats the gas-liquid two-phase medium as a homogeneous pseudo-single-phase medium with finite voids locally. That is, it is assumed that innumerable and infinitely fine bubble particles or droplet particles are homogeneously distributed in the micro-volume element in the gas-liquid two-phase medium. With the assumption of local homogeneity, the mixed density \( \rho \) of the two-phase medium is calculated as follows by using the gas phase density \( \rho_g \), the liquid phase density \( \rho_l \) and the local void fraction (volume fraction) \( \alpha \).

\[
\rho = (1 - \alpha) \rho_l + \alpha \rho_g \tag{1}
\]

We use an equation for the state of an ideal gas for the gas phase and Tammann's state equation for the liquid phase.

\[
p_g = \rho_g R_g T_g
\]

\[
p_l + p_c = \rho_l K_l(T_l + T_0)
\]

where \( p \) and \( T \) are pressure and temperature. Also, \( R_g = 287.0 \text{ J/(kg/K)} \), \( K_l = 472.2 \text{ J/(kg/K)} \), \( p_c = 1944.6 \text{ MPa} \), and \( T_0 = 3836.93 \text{ K} \) are the gas constant, the liquid constant, the liquid pressure constant, and temperature constant. Suppose that the pressure and temperature of the gas and liquid phases are in equilibrium, the equation of state of the homogeneous medium is written as follows.

\[
\rho = (1 - \alpha) \frac{p + p_c}{K_l(T + T_0)} + \alpha \frac{p}{R_g T} \tag{3}
\]

Here, the following relational expressions hold between the local void fraction \( \alpha \) and the quality (mass fraction of gas phase) \( Y \).

\[
\rho(1 - Y) = (1 - \alpha) \rho_l \quad \rho Y = \alpha \rho_c \tag{4}
\]

From these expressions, the state equation of the gas-liquid two-phase medium is given as follows using \( Y \).

\[
\rho = \frac{p(p + p_c)}{K_l(1 - Y)p(T + T_0) + R_g Y(p + p_c)T} \tag{5}
\]

2.2. Governing Equations
The governing equations are an equation of continuity for a gas-liquid two-phase medium, a 2-D compressible gas-liquid two-phase Navier-Stokes equation assuming an isothermal process, and an equation of continuity for a gas phase. Equations are discretized by a finite volume method. When the above three equations are summarized in vector form, these can be written as follow.
\[
\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = \frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y}
\]

(6)

\(Q\) is a vector of conserved quantity, \(E\) and \(F\) are flux vectors representing convection terms, and \(E_v\) and \(F_v\) are flux vectors representing viscous terms, which are given as follows.

\[
Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho Y \end{bmatrix}, \quad E = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uY \end{bmatrix}, \quad F = \begin{bmatrix} \rho v \\ \rho v^2 + p \\ \rho vu \\ \rho vY \end{bmatrix}, \quad E_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{yx} \\ 0 \end{bmatrix}, \quad F_v = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ 0 \end{bmatrix}
\]

(7)

\[
\tau_{xx} = \frac{2}{3}\mu \left( 2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right), \quad \tau_{xy} = \frac{2}{3}\mu \left( 2 \frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} \right), \quad \tau_{yx} = \tau_{xy} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)
\]

Where \(u\) and \(v\) are velocity components in the X and Y directions, respectively. And \(\tau\) is a viscous stress. It should be noted that the viscosity coefficient \(\mu\) of the gas-liquid two-phase local homogeneous medium is evaluated by the following equation to calculate the viscous stress.

\[
\mu = \left( 1 - \alpha \right) (1 + 2.5\alpha) \mu_t + \alpha \mu_g
\]

(8)

The convective flux was evaluated using flux difference splitting (FDS), and extended to a third order using the monotonic upwind scheme for conservative laws (MUSCL) interpolation. The viscous flux was discretized as the second-order central difference with Gauss’ theorem. For the time integration second-order Runge–Kutta method was used.

2.3. Computational Domains and Boundary Conditions

In this study, 2-D free jet field and impinging jet field are analysed. And axial symmetric flow is assumed. The nozzle orifice diameter \(D\) is 1.0 mm and the minimum grid width is 0.1 mm at the nozzle orifice portion. Overviews of the computational domains are shown in figure 1, and the grids are thinned for easy viewing.

An overview of the computational domain used for free jet field analysis is shown in figure 1 (a). The computational domain is taken from 15\(D\) upstream to 60\(D\) downstream with respect to the nozzle exit and the width is 20\(D\) in the radial direction. A multiblock grid is adopted, and the computational domain is divided into three regions, a nozzle front region, a nozzle orifice region, and a jet flow region. The number of grid points in the axial direction and radial direction in each region are 55 \(\times\) 50, 55 \(\times\) 15, and 355 \(\times\) 305, respectively. Therefore, the total number of grid points is about 11 million points. For free jet analysis, analysis was conducted under two conditions. In the case of the liquid single-phase state, the nozzle inlet pressure \(p_1 = 0.2\) MPa and the outlet pressure \(p_2 = 101.325\) kPa. The cavitation number \(\sigma = (p_2 - p_c) / (p_1 - p_2)\) is calculated to be 1.0. In the case of the gas-liquid two-phase state, the nozzle inlet pressure \(p_1 = 1.0\) MPa and the outlet pressure \(p_2 = 101.325\) kPa, cavitation number \(\sigma\) is calculated to be 0.11. At the inflow boundary, total pressure was fixed and free inflow conditions was set. At the outflow boundary, static pressure was fixed and free flow condition was set. At the wall boundary, adhesive condition was applied to velocity and von Neumann condition was imposed on density and pressure.

An overview of the computational domain used for impinging jet flow analysis is shown in figure 1 (b). Unlike the free jet analysis, a collision wall with a width of 10\(D\) was set at the position of the standoff distance \(X/D = 10D\) which is the first peak of the erosion curve. The computational domain is divided into four regions, a nozzle front region, a nozzle orifice region, a collision wall front region, and a collision wall upper region. The number of grid points in the axial direction and radial direction in...
each region are $55 \times 50$, $55 \times 15$, $105 \times 305$, and $255 \times 105$, respectively. Therefore, the total number of grid points is about 6.2 million points. In the impinging jet analysis, analysis was carried out only in the gas-liquid two-phase state with the cavitation number $\sigma = 0.11$. The boundary conditions were the same as free jet analysis.

![Diagram](a) Free Jet

![Diagram](b) Impinging Jet

Figure 1. Overview of the computational domain

3. Results and Discussion

3.1. Free Jet Analysis

First we confirm the validity of the analysis model by comparing with Geortler’s and Tollmien’s theoretical curves [11, 12]. The velocity distribution in the cross section at each standoff distance $X/D$ is shown in figure 2. In this graph, the distance $Y$ from the jet axis and the axial flow velocity are nondimensionalized by the half-width of the jet $b$ and the maximum axial flow velocity $u_m$. $b$ is the distance from the jet axis where the axial flow velocity $u$ is equal to $0.5u_m$. In the case of the liquid single-phase state (figure 2 (a)), the velocity distribution for each cross section shows a similar distribution and is in good agreement with Geortler’s and Tollmien’s theoretical curves. On the other hand, in the case of gas-liquid two-phase state (figure 2 (b)), velocity distributions have quadratic curve-like distribution, but it cannot be said that they are similar to each other. Differences from the theoretical curves are found both in the jet flow inner and outer regions, and the difference is larger in the outer region ($1.0 < b$). This is thought that flow becomes complicated due to a vortex cavitation.
Figure 3 shows the axial flow velocity distribution of the jet flow field. Jet flows from the left to the right of the page. According to figure 3 (a), it can be said that the flow diffuses stable. In addition, the initial length of the initial region $X_c$ is about 7.0 mm. On the other hand, in the case of gas-liquid two phase state (figure 3 (b)), the jet does not show a stable diffusion appearance and the jet width is suppressed to about 2.0 mm. It is considered that shear vortices are formed at the outer edge of the main jet. Further, the initial length $X_c$ gets longer to about 20.0 mm and the core region is maintained further downstream. In other words, it can be said that the usefulness of the cavitating jet was demonstrated such that jet energy can be propagated to the downstream by using the cavitating jet and the standoff distance $X/D$ can be lengthened.

With respect to the half-width of the jet $b$, we can write $b = C_2X$ from the theoretical equation of the jet. The slopes $C_2$ obtained from Goertler’s and Tollmien’s theoretical solutions are 0.114 and 0.097. In addition, Rajaratnam set $C_2 = 0.10$ as an experience value. In the liquid single-phase analysis, the slope $C_2$ was calculated to be 0.103, and the validity of this analysis model could be confirmed quantitatively.
Figure 4 shows the time course of the pressure distribution in the range up to the standoff distance $X/D = 30$ in order to clarify the periodic behavior of the cavitation bubble cloud released from the nozzle exit. The pressure at the deep blue part reaches saturated vapor pressure of water, $2332$ Pa, which indicates that cavitation occurs. Looking at the pressure distribution, we can see the cavitation bubble clouds occur periodically. The generated cavitation bubble clouds advect at a substantially constant speed like the path A, C, and E. The advection velocity is the same as the main stream velocity at that point. It has also been observed that some cavitation bubble clouds catch up with the bubble clouds generated earlier like the path D and F and coalesce.

![Figure 4. Time course of the pressure distribution](image)

3.2. Impinging Jet Analysis

Figure 5 shows absolute velocity distribution and pressure distribution. Please be aware that the length of the flow velocity vector on figure 5 (a) is corrected to be a uniform length, not shows the actual magnitude of the velocity. From figure 5 (a), the jet collides with the wall surface at the standoff distance $X/D = 10$ and flows along the wall as a wall jet. Poreh and Cermak [13], Bradshow and Love [14], Tani and Komatsu [15], Poreh et al. [16], and Beltaos and Rajaratnam [17] made the wall impinging jet experience and reported that the wall impinging jet radially spreads along the wall surface as a radial wall jet after hitting the wall. However, in this analysis, the jet didn’t diffuse radially. This is thought to be that the turbulence model is not used in this analysis, that is, mixing and diffusion action by turbulence are not taken into consideration and cavitation. Also, looking at the pressure distribution (figure 5 (b)), cavitation bubble clouds are periodically generated like the free jet flow field. Cavitation bubble clouds also exist after the jet collides with the wall surface.
Next, we investigated the propagation of the shock wave and pressure wave caused by the cavitation bubble cloud collapse when the cavitation jet collides and to evaluate the impact pressure on the wall surface. Figure 6 shows the time course of the pressure distribution, figure 7 shows the pressure distribution along the Y axis on the collision wall. Unlike the free jet flow field, cavitation bubble clouds are adhering to the wall without coalescence (figure 6). At the time $t = 0$, the cavitation bubble cloud collides, the pressure rises, and the void fraction decreases. A decrease in the void fraction means an increase in sound velocity, and the region transits from the supersonic region to the subsonic region. And then a shock wave is generated and the bubble cloud collapses ($t = 2$). At this time, the impact pressure increases to the maximum of 1.4 MPa (figure 7). The generated shock waves propagate with collapsing other bubble clouds and generate new shock waves and pressure waves. However, shock waves and pressure waves are strong in the direction along the wall surface and can be seen to be propagating with avoiding bubble clouds existing on the upstream side ($t = 4$). They propagate further upstream ($t = 6, 8$) and extend to the interior of the nozzle ($t = 10, 12$). Since new cavitation bubble clouds are released after the shock waves and pressure waves arrive at the nozzle part, it is suggested the relationship between the arrival of these waves and the emission of new cavitation bubble clouds. After that, the pressure fluctuates gets smaller, the crushed cavitation bubble clouds raise the void fraction again due to the rebound phenomenon, and become cavitation bubble cloud ($t = 14 \sim 20$). Ultimately, it collapses again on the collision wall surface and generates a shock wave ($t = 22 \sim 26$). In general, it is known from the erosion test results that erosion marks made by cavitation jet have ring-shaped, suggesting that this cavitation bubble cloud collapse pattern contributes to the formation of ring-shaped erosion marks. After that, periodic unsteady behavior is shown as well.

(a) Absolute velocity distribution with vector  
(b) Pressure distribution

**Figure 5.** Flow pattern of impinging jet
Figure 6. Time course of pressure distribution

Figure 7. Pressure distribution on the collision wall
4. Conclusions
We introduced the gas-liquid two-phase local homogeneous medium model and conducted a numerical simulation to visualize the jet structure of the cavitating jet and clarify the behavior of the bubble cloud collapse and to evaluate the impact pressure by the collision jet on the wall.

The results can be summarized as follows:

1. The numerical simulation results of the free jet flow field in the liquid single-phase state and the gas-liquid two phase state using the gas-liquid two-phase local homogeneous medium model showed good agreement with the theoretical solution and experimental results. The effectiveness of the simulation model could be shown.
2. Cavitating jet had smaller jet width than water jet in liquid single-phase state, and it was possible to propagate jet energy further downstream. Cavitatin bubble clouds were periodically emitted and advected at constant speed. Some bubble clouds coalesced with the preceding bubble clouds.
3. Cavitatin bubble clouds also showed cyclic behavior even in collision jet field, and they collapsed at wall collision and generated shock waves and pressure waves. It is suggested that these waves correlate with cavitatin bubble cloud generation.
4. After bubble clouds collapsed at the wall, generated shock waves again near the erosion due to the rebound phenomenon. It is thought that the characteristic ring-shaped erosion marks in the erosion test is formed by these impact pressures.

References
[1] Yamauchi A and Shimizu S 1987 Erosion Due to Impingement of Cavitating Jet *J.Fluids Engineering* **109** 442-447
[2] Soyama H, Yamaguchi Y, Adachi Y, Sato K, Shinoda T and Oba R 1993 High-Speed Observation of the Cavitatin Cloud around a High-Speed Submerged Water-Jet *Transactions of the Japan Society of Mechanical Engineers Series B* **59** (562) 1919-1924
[3] Kato H 2003 Environmental Protection Using Cavitation *Technology Turbomachinery* **31** (11) 648-654
[4] Tsujimura 1997 *J. Jet flow engineering* **14** (2) 21-25
[5] Yokota 1999 *J. Jet flow engineering* **16** (1) 17-22
[6] Soyama 1996 *J. Jet flow engineering* **13** (1) 25-32
[7] Shimizu S 1996 Effects of nozzle shape on structure and drilling capability of premixed abrasive water jets *Japan Society of Mechanical Engineers Series B* **62** (603) 84-91
[8] K K Ooi 1985 *J. Fluid Mech.* **151** 467-390
[9] Oguma Y, Peng G and Shimizu S 2017 *J. Jet flow engineering* **33** (1) 10-18
[10] Okuda K and Ikohagi T 1996 Numerical Simulation of Collapsing Behavior of Bubble Clouds *Japan Society of Mechanical Engineers Series B* **62** (303) 38-43
[11] Nomura Y 1981 *Jet flow* (Tokyo: Morikita Publishing Co.)
[12] Sakouchi T 2004 *Jet flow* (Tokyo: Morikita Publishing Co.)
[13] Poreh M and Cermak J E 1959 Flow characteristics of a circular submerged jet impinging normally a smooth boundary *Proc. 6th Mid-Western Conf. on Fluid Mechanics* 198-212
[14] Bradshaw P and Love EM 1961 The normal impingement of circular air jet on a flat surface (London: Aeronautical Research Council R and M) **3205** 1-8
[15] Tani I and Komatsu Y 1964 Impingement of around jet on a flat surface *Proc. 11th Conge. Appl. Mech. Munich* 672-676
[16] Poreh M, Tsuei, Y G & Cermak and J E 1967 Investigation of a turbulent radial wall jet *Trans. ASME. J. Appl. Mech.* **34** 457-463
[17] Beltaos S and Rajaratnam N 1973 Impinging circular turbulent jets *Proc. ASCE J. Hydraul Div.* **100** 1313-1328
[18] Fukushi A, Shin B R, Sato K and Ikohagi T 2000 Numerical Simulation of High-Speed
Submerged Water Jet with Cavitation. *Jet flow engineering* **17** (3) 16-21

[19] Fukushi A, Nakamori I and Ikohagi T 2002 Numerical Simulation of High-Speed Submerged Water Jet with Cavitation *Japan Society of Mechanical Engineers Series B* **68** (673) 46-53

[20] Inoue F, Outa E, Tajima K and Machiyama T 1993 An experimental Study on Control Valve Cavitation (3rd Report, Vortex Shedding and Cavitation Inception on a Contourde Plug) *Japan Society of Mechanical Engineers Series B* **59** (562) 131-138

[21] Sato K, Ohjimi S and Sugimoto Y 2009 Collapsing and Impulsive of Cavitation Clouds on Cavitating Water-Jet Impinging on Solid Wall *Japan Society of Mechanical Engineers Series B* **75** (750) 47-56