Numerical simulation of cavitations jet flow structure and impact pressure properties

Min-ge ZHANG¹, Xing-hua LIN², Jianguo WU²
¹College of Mechanical Engineering, Tianjin University of Science and Technology, Tianjin 300222, China,
²College of Mechanical Engineering, Hebei University of Technology, Tianjin 300130, China.

Abstract. Cavitations jet is a kind of continuous jet involved cavitations bubbles, and the impact force is far greater than normal continuous jet. In this paper, the structural characteristics of cavitations jet flow field and the distribution characteristics of the impact pressure are studied by the numerical simulation method, and the effect of inlet pressure on the extent of cavitations is also further analyzed. The results show that: in the flow field, the jet diverges to surrounding when it is near the component, and produces a noticeable cavitations region in this position. The impact pressure in the radial direction shows the Gaussian distribution in the flow field; the impact force on the rigid wall acts in an annular area, and the distribution curve has good self-mold resistance. These results coincide to the experimental results. Furthermore, the effect of inlet pressure on the extent of cavitations exists a certain limit.

1. Introduction

Cavitations is the phenomenon that producing cavitations bubbles when the pressure in partial flow system is less than the saturated vapor pressure, and cavitations jet is a kind of continuous jet involved of these cavitations bubbles[1-3].

With the development of computer technology, studying the cavitation jet through the numerical simulation becomes a new analysis method. Lu Zhaohui [4] studied the structural properties of flow field produced by two kinds of high-pressure pulsed water jet, squeeze and cut off, based on numerical simulation. Up to now, there has been clear evidence to prove the injection pressure as an important factor that influences the cavitation jet shape stability and the size of impact[5-7]. So it is interesting to clarify the relationship between the size of injection pressure and the distribution of impact pressure in cavitation flow quantificationally[8-10]. In this paper, using the Venturi nozzle as the physical model, studied the relationship between the size of injection pressure and of impact pressure through CFD methods.

2. Mathematical model

In present paper, the Fluent 6.3 software and the homogeneous equilibrium model was applied for cavitation phenomenon in the Venturi nozzle. The mixture model is selected to be the multiphase flow model, and the liquid water and water vapor are selected to be basic phase and the second phase respectively, that the case of velocity slip between them is neglected, so we consider the vapor/liquid
flow a homogeneous mixture flow. There is no differentiation about the basic equations of continuity and momentum between liquid and vapor.

In a flowing fluid with no velocity slip between the liquid and vapor, the progress of bubble growth and collapse is described by the Rayleigh-Plesset equation, which is as following (1).

\[ \frac{3}{2} \frac{R^2}{RR} \frac{dR}{dt} = \frac{p_l - p}{\rho_l} - \frac{2\Omega}{\rho_l R} - \frac{4\mu R}{\rho_l R} \]  

(1)

In the progress of solving governing equations, the body force due to gravity is considered. The PISO scheme with small relaxation factor is used for the pressure–velocity coupling to allow the higher pressure correction and to speed up convergence.

3. **Physical model and calculation method**

In present study, the scaling cavitations nozzle is selected to be the physical model. Its physical structure and type of boundary are shown as in Fig.1. The type of entrance is pressure-inlet, and outlet is set up on both sides in order to simulate an open basin. Assuming boundary wall-2 as a component surface, and use the parameters of its surface to express the effects of cavitations jet. The rest of the boundaries are rigid walls-1. Quadrilateral element is used in meshing, and the grids are encrypted in the narrowing part due to a sharp rise of velocity at the position.

![Figure 1. The physical model of a scaled cavitations nozzle](image)

4. **Results and analysis**

4.1 **Structural characteristics of cavitations jet flow field**

When the injection pressure is 2MPa, 4MPa, 6MPa and 8MPa respectively, the characteristics of cavitations jet in the flow field are studied. The distribution of the impact velocity on the surface of the plate was investigated as shown in Fig.2, where the color contour represents the magnitude of the velocity in the field. And the volume fraction contours of gas are shown in and Fig.3.

![Figure 2. velocity contour maps \(P_0 = 6MPa\)](image)  
![Figure 3. vapor fraction contours](image)

As shown in Fig.2, when the injection pressure is 6MPa, the flow detaches from the wall and forms a constricted jet when it enters the narrow segment, and the speed increases rapidly from 50m/s to 106m/s. There is an air gap between the jet and the wall, which envelops the jet all the way through the narrow segment. For this reason, there is a long constant velocity field, where the cavitation does not disturb the structure of this flow. The diameter of jet is close to the nozzle, but the jet shown in Fig. 1 is not a continuous column actually, and it can be observed apparently that consists of a continuous
region and a discrete region. The discrete region is the region where the jet is broken to both sides of the central axis, when the jet reaches the vicinity of surface. And the speed decreases rapidly until the jet reaches the surface, where the value of speed is 0 at the centre point called stagnation point. On the surface of simple, the flow field can be divided into two regions, which is impact zone and wall jet zone respectively. There is obvious water-pillow in the impact zone, and the speed of flow rose first and then fell along the radial in the wall jet zone.

Because the flow field is symmetrical, for the convenience, the half of the physical model is used to represent. As shown in Fig 3, the severest region of cavitations in the flow field is coinciding exactly with the beginning of discrete region except the inside of nozzle, and the position hardly changes with the injection pressure. This is because the jet is obstructed by the wall-2, and the interaction between constricted jet and wall-2 produces the recirculation flow region, where exists the large-scale vortices, as known to all, the lower static pressure usually appears in this region. Therefore, there is enough liquid tension in this region to produces cavitation bubbles. As the pressure increases, the shape of the cavitation region becomes thinner and longer, and the cavitations areas are also growing since the interaction because stronger. When the injection pressure reaches to 6MPa, the phenomenon of all-cavitations can be observed. However, in the continuous region mentioned above, there is almost no the phenomenon of cavitation, which is because the jet stays laminar even at remarkably high Reynolds numbers. Therefore, there is an obvious influence of injection pressure on the intensity of the cavitation field, but the position of cavitation is depended on the nozzle geometry.

4.2 The distribution characteristics of impact pressure

When a water jet impacts a flat solid, the impact pressure exerted on the solid boundary wall-2 by the liquid jet can be calculated using the water-hammer equation (2).

\[ P = \rho_l C_l v_m \left\{ \frac{\rho_s C_s}{\rho_l C_l + \rho_s C_s} \right\} \]  

(2)

Where \( \rho \) is the density and the \( C \) is sound speed. The subscripts \( l \) and \( s \) refer to the liquid and solid respectively. The \( V_m \) refers to the impact velocity. But compared to \( \rho_l C_l \), the value of \( \rho_s C_s \) is large. So the Eq. (2) can be reduced to Eq. (3) approximatively.

\[ P = \rho_l C_l v_m \]  

(3)

The impact pressure given by Eq.(3) will decrease rapidly when the release waves propagate into the jet from the circumference, and this impingement continues until a steady state is reached, at that time, the pressure approaches hydro-dynamical pressure, as shown as Eq.(4).

\[ P = \frac{1}{2} \rho v_m^2 \]  

(4)

4.2.1 The distribution characteristics of impact pressure in the flow field

In order to study the distribution characteristics of impact pressure in the flow field, the position which is 4mm far from the surface of sample are selected to make the distribution cures of the dynamic pressures in the radial as shown in Fig.4, when the injection pressure is 2MPa, 4MPa, 6MPa and 8MPa, respectively.

![Figure 4.](image1.png)  
**Figure 4.** the distribution cure of dynamic pressure

![Figure 5.](image2.png)  
**Figure 5.** the dimensionless dynamic pressure distribution curve and fitting curve
As shown in the Fig. 4, with the increasing of inlet pressure, the maximum impact pressure is gradually increased, and the distribution of dynamic pressure becomes steeper and the impact pressure is also more concentrated. Under the different inlet pressures, the distribution curves of dynamic pressure have an obvious self-mold property. By further analysis, the result shows that the distribution approximately is Gaussian distribution. When the inlet pressure is greater than 6MPa, there is a splitting near the peak, and with the increasing of inlet pressure, the degree of splitting is more obvious.

To make treatments of experimental data into dimensionless data, using $\frac{m}{PP}$ as the vertical axis, and $\frac{1}{2} \frac{x}{d_{1/2}}$ as the horizontal axis ($d_{1/2}$ is the width of half-peak). Then the distribution curve of impact pressure under all inlet pressure is substantially the same. When the inlet pressure is 6MPa, the distribution curve and fitting curve is shown in Figure 5.

Where the curve fitting equation is:

$$P / P_o = \exp[-3.334(x / d_{1/2})^2], \ R^2 = 0.9904$$ \hspace{1cm} (5)

This fitting equation is identical with the conclusion in literature [5]:

$$p / p_o = \exp(-\alpha \eta^2)$$ \hspace{1cm} (6)

Where $\alpha$ is the concentration factor of impact pressure, and with the value of $\alpha$ increasing, the degree of concentration is higher in the high speed water jet. In present paper, the changing of $\alpha$ with the injection, which is from 1MPa to 10MPa, is shown as Tab.1.

| $P_o$/MP | $P_{in}$/MP | $d_{1/2}/10^{-3}m$ | $\alpha$ | $R^2$ |
|---|---|---|---|---|
| 1 | 0.8752 | 1.63 | 3.145 | 0.9930 |
| 2 | 1.8886 | 1.50 | 3.255 | 0.9911 |
| 3 | 2.8277 | 1.64 | 3.409 | 0.9879 |
| 4 | 3.7863 | 1.70 | 3.512 | 0.9907 |
| 5 | 4.7119 | 1.70 | 3.377 | 0.9846 |
| 6 | 5.6687 | 1.70 | 3.334 | 0.9914 |
| 7 | 6.5990 | 1.83 | 3.159 | 0.9917 |
| 8 | 7.5769 | 1.76 | 3.080 | 0.9917 |
| 9 | 8.5224 | 1.76 | 3.157 | 0.9892 |
| 10 | 9.3165 | 1.90 | 3.300 | 0.9907 |

In the high speed water jet, with the speed increasing, the value of $\alpha$ is larger. But in the cavitation jet, the value of $\alpha$ fluctuates with the injection pressure, as shown in Fig. 6. This maybe caused by the influence of cavitation bubbles. With the increasing of injection pressure, the speed of jet and the degree of concentration is higher, but meanwhile, the cavitation will be stronger, due to the waves produced by cavitation bubbles can make the jet divulge, therefore, there is an balance of the interaction between water jet and cavitation bubbles. As shown in Fig. 7, when the injection pressure is 4MPa, the degree of concentration is the highest and then decreases with the increasing of injection pressure, where the lowest is at the position of 8MPa, but at the position between 5MPa and 6MPa, there is a platform, so the balance point maybe at the position where the injection pressure is 5.5MPa and the value of $\alpha$ is 3.35 in present paper.
4.2.2 The distribution characteristics of impact pressure on the surface of simple
When the inlet pressures are 2MPa, 4MPa, 6MPa, 8MPa and 10MPa respectively, the distribution curves of dynamic pressure on the surface of the components are shown in Fig.8(a).

As shown by Fig.8(a), the distribution of impact pressure on the surface of sample is quite different from the distribution inside the flow field along the radial direction. As mentioned above, there is water-pillow near the centre point, and due to the obstructive effect of water-pillow, the water jet needs to consume a lot of impact energy to overcome this resistance in this area, causing the loss of energy. So the water jet inside the field spreads to surrounding areas near the surface, and when the jet reaches the surface of sample, the impact pressure decreases to zero. But at the position where is 5mm far from the centre point, the impact pressure reaches to the maximum value, which is linear change with the injection pressure, as shown in Fig.8(b). The distribution curves under different injection pressures also show obvious self-mold property, just as the inlet pressure increases, the curves become steepening.

4.3 The effect of inlet pressure on the extent of cavitations
When the inlet pressures are 2MPa, 4MPa, 6MPa, 8MPa and 10MPa respectively, the distribution curves of volume fraction of gas on the surface of the component are shown in Fig.9.
As can be seen from Fig.9, the volume fraction of gas on the components shows a parabolic shape. In the vicinity of the central axis, the volume fraction of gas is small, and gradually increases toward the exit. With the increasing of the inlet pressure, the volume function of gas increases, but when the inlet pressure reaches 6MPa, the distribution curves of volume function are nearly same, in the other words, the increasing of volume function of gas is no longer apparent with the inlet pressure, indicating that there is a limit of the effects which are on the extent of cavitations by the inlet pressure.

In this paper, the cavitations number is used to express the effects of inlet pressure on the extent of cavitations.

\[ \sigma = (P - P_v) \rho \frac{v^2}{2} \]  

Where, the \( P \) is the surrounding pressure, namely the outlet pressure, and in this article is the standard atmospheric pressure; \( P_v \) is the saturated vapor pressure of water, and in this article is 2367.8Pa; \( \rho \) is the density of the liquid, and \( v \) is the velocity of jet. When the distance is 10mm and 15mm from the center axis, the curves of volume fraction of gas and the cavitations number are shown in Fig.10.

As can be seen from the Fig.10, when the inlet pressure increases, the cavitations number is gradually decreased, while the volume fraction of gas increases. This meets the law that when cavitations number is smaller, the volume fraction of gas is higher. With the inlet pressure increases, both curves become flattening, which validated the above results that there is a certain limit of effects of the inlet pressure on the extent of cavitations.

5. Conclusions
1. The speed decreases rapidly when it reaches to the near-surface of components, and at the position, there is a large cavitations region. With the inlet pressure increases, the area of cavitations region increases.
2. The distribution of dynamic pressure in the radial presents approximately Gaussian distribution in the flow field, and the maximum is in the vicinity of the central axis. There is a splitting phenomenon at the position of peak as the pressure increases.
3. On the surface of component, the maximum impact pressure is in the position where at a distance from the center axis, and the impact region is a ring.
4. There are effects of inlet pressure on the degree of cavitations, which in a certain range, with the increasing of inlet pressure, the degree of cavitations is higher. When the inlet pressure increases to a value, the effects are not significant any more.

References
[1] Zhu, B., Chen, H. (2012) Cavitating suppression of low specific speed centrifugal pump with gap drainage blade. *Journal of Hydrodynamics*, 24, 729-736.
[2] Yang, Q., Wang, Y., Zhang, Z.,(2011) Effects of non-uniform inflow on propeller cavitation hydrodynamics. *Chinese Journal of Hydrodynamics*, 26: 538-550(in Chinese).
[3] Ji, B., Luo X., Wu, Y.,(2013) Numerical analysis of unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil. *International Journal of Multiphase Flow*, 51:33-43.
[4] Lu, Z., (2012) CFD modeling on flow-field Structure of high pressure pulse water jet and its hard rock fragmentation mechanism. *Chongqing: Chongqing University*(in Chinese).
[5] Hitoshi S.,(2013) Effect of nozzle geometry on a standard cavitation erosion test using a cavitating jet. *Wear*, 297: 895-902.
[6] Hitoshi, S., Tsutomu, K.,(2011) Introduction of compressive residual stress into stainless steel by employing a cavitating jet in air. *Surface & Coating Technology*, 205: 3167-3174.
[7] Choi, J., Arvind J., Georges, L.,(2012) Chahine. Scaling of cavitation erosion progression with cavitation intensity and cavitation source. *Wear*, 278-279: 53-61.
[8] Wang, Y., Lu, Q., Rolf, D. R.,(2014) Simulating cavitating liquid jets using a compressible and equilibrium two-phase flow solver. *International Journal of Multiphase Flow*, 63: 52-67.
[9] Shuji, H., Takuya, H., Kenichi, S.,(2010) Prediction method for cavitation erosion based on measurement of bubble collapse impact loads. *Wear*, 269: 507-514.
[10] Peng, G., Shimizu, S.,(2013) Progress in numerical simulation of cavitating water jets. *Journal of Hydrodynamics*, 25: 502-509.