Numerical simulation analysis of cavitation of current limiting orifice plate

Lingying Ni, Bing Li* and Guangwei Geng

School of Petroleum Engineering, China University of Petroleum (East China), Shandong 266000, China

*Corresponding author’s e-mail: 851433954@qq.com

Abstract. In this paper, ANSYS-Fluent software is used to simulate the cavitation phenomenon that may occur in the current limiting orifice plate of the vertical oil inlet pipe with large drop in the underground water sealed oil depot. The influence of different orifice designs on the cavitation are calculated and summarized. The simulation results show that the shape of the orifice is an important factor affecting the cavitation intensity, the chamfering of the orifice can restrain the cavitation. The circular chamfer is better than the 45° chamfer to reduce the cavitation intensity, and the cavitation intensity will decrease with the increase of the thickness of the orifice chamfer.

1. Introduction

China is a big oil consuming country, with the rapid development of China's economy in recent years, China's import demand for oil resources is increasing day by day. The net import volume is expected to reach more than 50% in 2020. It is necessary to establish a national strategic oil reserve base as soon as possible. As China's national petroleum strategic reserve gradually adopts the form of underground water sealed oil depot, a series of energy storage technology problems need to be solved. The depth of the underground water sealed cavern of the National Petroleum Reserve is mostly more than 100m, which adopts pipeline transportation. When a large amount of oil is injected into the cavity of the vertical pipe with a large drop, the potential energy of the fluid in the vertical pipe is far greater than the resistance drop, and there is a surplus of the fluid energy. If measures are not taken to consume this surplus energy, complex changes will occur in the liquid flow of the large drop pipeline section [1], then partial flow may be occurred in pipeline. The increase in local flow rate will cause the transient change of the pressure inside the pipe or reach the vaporization pressure of oil at the conveying temperature. This will cause the gas separated out from the oil. At the same time, the unstable gas-liquid mixed flow (slump flow) has a certain pulsating impact on the pipeline, which may cause the pipeline to vibrate, the anchoring pier to fall off, or collide with other pipelines to cause potential engineering accidents.

The orifice plate is one of the widely used pressure reducing devices in oil storage field. But the relationship between the orifice plate's design parameters and the hydraulic cavitation phenomenon has not been fully studied. Cavitation is a complex hydrodynamic phenomenon unique to liquids. When the fluid flows to the restriction zone, the flow rate increases and the pressure decreases due to the sudden narrowing of the flow zone. If the pressure is lower than the vaporization pressure of the flowing medium at this temperature, not only the liquid will vaporize, but also the gas core contained inside the fluid will rapidly expand and precipitate, causing the flow to change from the original single-phase flow to the unstable multi-phase flow. This phenomenon is called hydrodynamic cavitation [2]. When the fluid carries the vacuole through the region with a higher pressure in the downstream, the bubble is
compressed and finally collapses sharply. The cavitation will generate a large instantaneous pressure when it collapses, when the collapse occurs near the solid surface, the cavitation in the liquid will produce a higher pressure, and the solid surface will be damaged under this repeated action. This phenomenon is called "cavitation" [3]. This phenomenon should be avoided as much as possible in projects. Some studies have analysed the effects of different inlet or outlet pressures and structural parameters of the orifice on cavitation intensity [4-6]. In this paper, the relationship between chamfer design parameters and cavitation phenomena on the orifice plate is discussed, which will provide some references for the optimal design of the orifice plate.

2. Mathematical model and computational strategy

2.1 Turbulence model

The flow separation phenomenon with the adverse pressure gradient often occurs in the downstream zones of the orifice plate. The SST k-ω turbulence model has higher calculation accuracy and lower calculation cost in simulating the flow problem with the adverse pressure gradient flow separation. The model was proposed by Menter, combining the advantages of the k-ω model in the near-wall zones and the k-ε model in the far-field zones. Therefore, this model has a high calculation accuracy. The kinetic energy k and the dissipation rate ω in this model are defined as follows:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} (\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k + S_k
\]

(1)

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} (\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega
\]

(2)

Where \( \tilde{G}_k \) is the turbulent kinetic energy due to the mean velocity gradient, calculated from \( G_k \); \( G_\omega \) is the turbulent dissipative energy; \( \Gamma_k \) and \( \Gamma_\omega \) are the effective diffusivity of k and \( \omega \), respectively; \( Y_k \) and \( Y_\omega \) are the dissipation of k and \( \omega \) due to turbulence; \( D_\omega \) is a cross-diffusion term; \( S_k \) and \( S_\omega \) are user-defined source terms.

2.2 Computational strategy

In this paper, ANSYS-Fluent software is used for the simulation. The cavitation phenomenon of the orifice plate in the large drop vertical tube is numerically simulated. The 2D axisymmetric calculation is used in the whole simulation process. The calculation used the pressure-based solver, mixture multiphase flow model. The gravity effect is considered. We used SST k-ω turbulence model and the non-slip standard wall function boundary conditions, inlet and outlet are fixed with pressure boundary conditions. In order to ensure the accuracy of the solution, the ordinary single-phase calculation is first performed in each case. After the calculation is converged, the single-phase calculation result is used as the initial condition for the start of cavitation. The Schnerr-Sauer cavitation model is activated by the cavitation option in the phase interaction and the SIMPLE algorithm was used for pressure velocity coupling. To improve calculation accuracy, 2nd order discretization is applied to the momentum, pressure and turbulent quantities in each instance, the other adopts the default option.

The orifice plate model used in the simulation is shown in Figure 1. The diameter (d) of the orifice plate is 40 mm and the thickness (h) is 36 mm. There are three types of orifice edge shapes: non-chamfering, round chamfering and 45 ° chamfering. Each chamfer shape is divided into different sizes, as shown in Figure 2, where the chamfering radius (thickness) of \( r_0 \) to \( r_3 \) is 0mm, 1.2mm, 3.8mm, 6.4mm, respectively; the thickness of the chamfer of \( l_0 \) to \( l_3 \) is 0mm, 1.2 Mm, 3.8mm, 6.4mm respectively.
3. Results & discussion

3.1. Influence of orifice chamfer

The chamfering of the orifice will have a significant impact on the flow field and is an important factor affecting the cavitation inside the orifice. Figure 3 compares the two orifice shapes of the orifice plate, which are unchamfered orifice (left) and an orifice with a chamfer radius of 3.8 mm (right). Figure 3(a) shows the velocity distribution for both cases. Due to the sudden contraction of the orifice, both flow separation occurs at the inlet edge of the orifice and a large velocity is produced, with a maximum velocity exceeding 46 m/s, the velocity at this time is sufficient to drive the local pressure below the vaporization pressure. This will lead to the occurrence of cavitation. But we can find that inside the chamfered orifice, the cavity caused by flow separation is smaller. Smooth orifice is shown to inhibit the development of cavity formation. The corresponding pressure distribution cloud diagram is shown in Figure 3(b). The minimum pressure of the unchamfered orifice reached -446354pa, which is located near the wall of the orifice inlet. The lowest pressure in the same position inside the round chamfer orifice is -115178pa. The chamfer causes the minimum pressure inside the orifice rose significantly. The above factors will reduce the amount of bubbles generated due to cavitation. It can be seen from Figure 3(c) that inside the chamfered orifice, the turbulent energy dissipation value is smaller, which is related to the decrease in the amount of cavitation generation. Figure 4 is a scatter plot of the vapor
phase volume fraction. It can be seen that the chamfering the orifice will reduce the volume of the vapor phase in the orifice. The presence of the chamfer of the orifice inhibits the occurrence of cavitation.

![Figure 3](image1.png)

**Figure 3.** (a) Velocity distribution (b) Pressure distribution (c) Turbulence kinetic energy distribution

3.2 Influence of orifice chamfer type

The type of the orifice chamfer will also affect the flow field, which will affect the occurrence of cavitation. Figure 5 to Figure 7 are the scatter plots of vapor volume fraction inside the orifice plates with two different orifice chamfers. It can be seen from the comparison that when the thickness of orifice chamfer is the same, the volume of vapor phase inside the round chamfered orifice is smaller than that of the 45° chamfered orifice. The volume of the vapor phase decreases with the decrease of the sharpness of the orifice chamfer. This shows that the round chamfered orifice has a better inhibitory effect on cavitation than the 45° chamfered orifice.

![Figure 4](image2.png)

**Figure 4.** Vapor volume fraction scatter plot (a) r=0mm (b) r=3.8mm

![Figure 5](image3.png)

**Figure 5.** Vapor phase volume fraction scatter plot r = 1 = 1.2mm (a) round chamfered orifice (b) 45° chamfered orifice
3.3 Influence of orifice chamfer thickness

Taking the circular chamfer with good cavitation suppression effect as an example, the thickness of the orifice chamfer is controlled by changing the curvature radius of the chamfer. Figure 8 shows the pressure distribution and turbulent kinetic energy distribution inside the orifice with the curvature radius of 1.2mm, 3.8mm and 6.4mm respectively. The simulation results show that the inside of the orifice with a chamfer radius of 6.4 mm has the smallest negative pressure zones and the lowest turbulent energy dissipation value, followed by the orifice with a chamfer radius of 3.8mm and 1.2mm respectively. With the increase of the orifice chamfer thickness, the negative pressure zones and the turbulent energy dissipation value in the orifice will be reduced, which indicates that the number of bubbles generated in the orifice is reduced. Figure 9 shows the corresponding vapor volume fraction scatter plots with three different orifice chamfer radius. It can be seen that the gas phase volume decreases with the increase of the chamfer thickness of the orifice. The increase of the thickness of the orifice chamfer will reduce the cavitation intensity.

![Figure 8. Effect of radius of chamfer curvature on cavitation (a) pressure distribution (b) turbulent kinetic energy distribution](image-url)
4. Conclusion

In this paper, the numerical simulation of the cavitation phenomenon in the orifice plate of the vertical flow inlet of the groundwater sealing oil reservoir is carried out. The influence of different orifice design parameters on the cavitation phenomenon is calculated. The numerical simulation results show that: The shape of the orifice edge of the orifice plate is an important factor in the occurrence of cavitation. Unless restraint measures are taken, the sharper orifice edge of the orifice plate will cause severe flow separation, and the resulting negative pressure cavity will induce cavitation; The existence of chamfer of the orifice will reduce the separation strength and inhibit the occurrence of cavitation; The circular chamfer has a better effect than the 45° chamfer in suppressing cavitation; Increasing the chamfer thickness of the orifice is beneficial to reduce the cavitation strength. However, the increase of the chamfer thickness of the orifice will increase the difficulty of processing and reduce the decompression effect. Therefore, in actual project, the chamfering radius of the orifice should be appropriately increased according to the specific needs. The optimal design of the orifice should be achieved on the premise of ensuring the decompression effect.

References

[1] Ni, L.Y., Xie, C.L., Li, C.H. (2011) Hydraulic calculation of large drop pipe in cavern. J. Oil & Gas Storage and Transportation, 30(12): 917–918
[2] Zhang, F.H., Liao, Z.F., Tang, C.L., et al. (2004) Chemical effects of cavitation water jet. J. Journal of Chongqing University, 27 (1): 32–35.
[3] Wang, Z.Y. (2006) Numerical simulation of hydraulic cavitation based on FLUENT software. D. Dalian University of Technology.
[4] Wang, Y.J. (2017) Numerical simulation and experimental study of hydraulic cavitation based on orifice plate and jet. D. North University of China.
[5] W, J.Y. (2015) Numerical simulation of cavitation process of porous plate hydrodynamic cavitation generator. D. Tianjin University of Science & Technology.
[6] Zhang, Y. (2011) Hydraulic cavitation and CFD numerical simulation. D. Zhejiang University of Technology.