NUMERICAL SIMULATION OF CAVITATING FLOW IN HYDRAULIC CONICAL VALVE[*]

Gao Hong, Fu Xin, Yang Huayong
State Key Laboratory of Fluid Power Transmission and Control, 310027,Hangzhou, P.R.China
SD19908023@sfp.zju.edu.cn

ABSTRACT

The paper describes a simplified numerical analysis of turbulent flow field inside hydraulic conical valves using RNG k-epsilon turbulent model. The simulations are under unsteady, incompressible and cavitating conditions. The Reynolds numbers based on the uniform velocity and inlet diameter are 6.5×10^4 and 1.0×10^5. The fluid is water, and the structure of the flow passage in the valve is simplified as a two dimensional axisymmetric geometrical model. The velocity and pressure fields and the cavitation distribution are presented. The effects of flow rate, valve opening as well as cone shape on flow field are numerically investigated and compared.

INTRODUCTION

Cavitations are generally considered as a harmful phenomenon in the hydraulic transmission system, cavitation noise has frequently posed serious problems of environment pollution and vibration hazards, at the same time, the efficiency of a system is reduced due to cavitation, especially dynamic performances are deteriorated. Valve is a noise source in hydraulic system, so the study on cavitating flow in valves is considerably required.

Much experimental research has been published on the flow characteristics of throttle valves, poppet valves and others in many literatures and studies on the relationship between cavitation and discharge, and thrust force and pressure distributions in the valves [1-3], but the results is still rather limited in accuracy and scope, and that further work is necessary.

By the advent of the computational Fluid Dynamics (CFD), many numerical simulations of flow fields inside the valves have been investigated. Under the assumption of non-cavitating condition, Vaughan et al. [4] performed simulation of flow through poppet valves, and results were compared with experimentally visualized flow patterns, qualitative agreement between simulated and visualized flow patterns was good. However, error in the prediction of jet separation and reattachment resulted in quantitative inaccuracies. These errors were due to the limitations of the upwind differencing scheme employed and the representation of turbulence by the k-ε model. Ueno et al. [5] investigated experimentally and numerically the oil flow in a pressure control valve under the assumption of non-cavitating condition, and concluded that the main noise of the testing valves is generated from cavitation, and the noise is influenced by the valve configuration, and the influence of various configurations of valve elements were discussed. Henrik L.Sørensen [6] described experimental and CFD analyses of the interior flow characteristics and flow force in conical seat valve in a specific range of 500 to 4600 in Re. However, the various and sometimes complex valve geometries mean that their results do not always correct very well at high

---

[*] This project is supported by SRF for ROCS, SEM and National Natural Science Foundation of China (59835160)

Reynolds numbers. Thus a continuing investigation into the performance of valves under cavitating condition is extraordinary necessary.

In this paper, based on the RNG k-epsilon turbulent model, numerical simulations of flow field in hydraulic conical valves are performed under unsteady, cavitating conditions. The cavitating flow characteristics are analysed and compared. The results provide beneficial reference for further research.

SIMULATION OF CONICAL VALVE

(1) Grid Generation and Boundary Condition

The structure of the valve is simplified as a two dimensional axisymmetric geometrical model shown in Fig.1. The topology and the grid of the fluid domain are created in an advanced grid preprocessor, gambit. In order to enable the features of the flow field to be better resolved, the solution-adaptive grid feature of Fluent is engaged in refining the original grid based on the geometry, boundary and numerical solution data of volume fraction gradient of water vapor. The adaptive final grids are presented in Fig.2. The boundaries of the valve are specified as inlet velocity, outlet pressure and wall boundary. The inlet velocity is set to 3m/s, 5m/s and 8m/s, respectively. The outlet pressure is set to 0.1Mpa. The initial conditions are as follows: v= inlet velocity and p=0.The wall is assumed to be adiabatic, and therefore no heat transfer between the fluid zone and the solid zone is considered. The fluid is water with ρ =998.2kg/m^3, μ =0.001003kg/m-s.

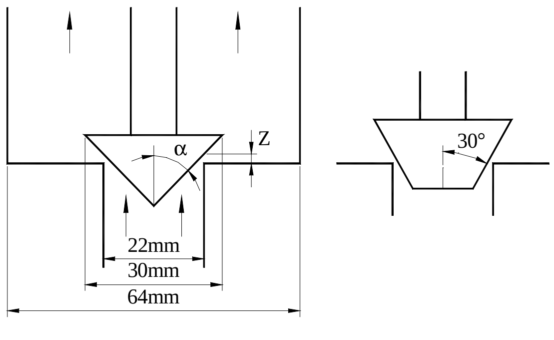


Fig. 1 Shapes of conical valve

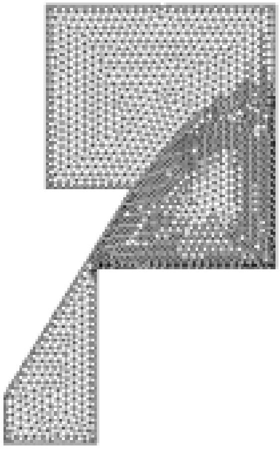


Fig. 2 Solution-adaptive gird of the fluid domain

(2) Computational Result and Discussion

The computation was performed at cone angle, α, of 30°, 45°. The Reynolds number Re, based on the uniform velocity and inlet diameter, was 6.5×10^4, 1.0×10^5. The computation at α=30° with truncated cones was performed with different front face area in order to see the effect of cone shape on the cavitating flow field. In addition, cavitating flow field for different valve opening and flow rate at α=30°, 45° obtained respectively.

A simulated flow pattern is shown in Fig.3 (a) in which the velocities at alternate nodes are shown. It can be seen that the simulated jet contracts downstream of the orifice and then travels at the angle of the cone face. It is deflected by the side wall and two recirculation zones are formed on either side of the jet. A smaller, intense recirculation zone occurs behind the rear face of the cone. Vortex occurs in two recirculation zone. The velocity magnitude contour and total pressure contour are shown in fig.3 (b, c). The simulated flow velocity in the orifice region is higher, but pressure is lower. The predicted results agree with the Bernoulli equation. Fig.4 (b) shows the corresponding cavitation distribution. It is noted that the region of cavitation inception corresponds to the region of low pressure and the center of the vortex agrees with the maximum volume fraction of water vapor. Because of the viscosity action and strenuous motion of fluid particle in the vortex region, flow energy is dissipated.

A series of cavitation distribution results are presented in Fig.4. The numerically obtained contours show volume fraction of water-vapor distribution. As it appears, the presence of water vapor indicates that cavitation has occurred and mainly occupied the jet and vortex zone. Fig.4 (a) and (b) show the contours of volume fraction of water vapor for 30° and 45°with equal valve opening (z=2mm) and flow rate (v=5m/s). The cavitation inception area and the maximum volume fraction of water vapor indicate a decrease with increasing core angle. Comparing Fig.4 (b) with (c), it can be observed that the increase of flow rate would be considerably prone to cavitation for 45° conical valve with equal valve opening (z=2mm). From Fig.4 (d), (e) and (f), it is clear that the truncated cones are effective in suppressing cavitation and with the increase of the truncated length the cavitation is reduced. Comparing Fig.4 (a) with Fig.4 (d), it can be found the intensity of cavitation is reduced with increasing of valve opening.

CONCLUSION

(1) Recirculation is an important feature of the flow through conical valves. The cavitating flow characteristics are more sensitive to intense recirculation zone in the orifice region than the recirculation which occurs downstream of the valve.

(2) Local energy loss in the conical valve primarily arises form vortical motion and the intensity of vortex affects the local pressure distribution most severely.

(3) The numerical achieved cavitation distribution indicates the cavitation occurs primarily in the jet and vortex region and the intensity of cavitation inception varies with the magnitude of the vortex.

(4) The flow rate, valve opening and shape of cone have a considerable effect on the intensity of the cavitation.

(5) The flow field in the conical valve is complex, which has great effect on flow energy loss and vibration noise. The analyses in this paper are helpful for further valve structure design of energy saving and noise reduction.

REFERENCES

[1] Inaguma, y. and Hibi, a., A study on cavitation in two-stage hydraulic restrictors. J. Jpn. Hydro. Pneum. Soc., (in Japanese), 1988, 9(7), 572.

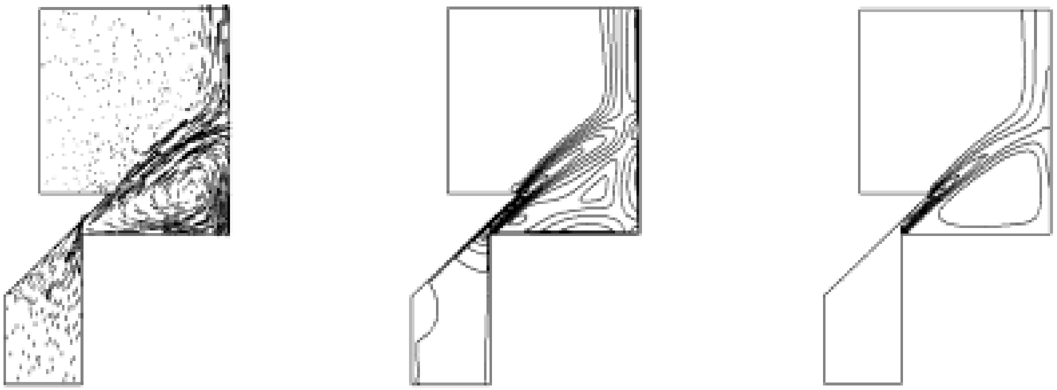
[2] Oshima, S., An experimental study on the several poppet valves with difference in shape. JHPS Intern. Sympo. on Fluid Power, Tokyo, 1989, 359.

[3] Johnston, D. N., Edge, K. A. and Vaughan, N. D., Experimental investigation of flow and force characteristics of hydraulic poppet and disc valves. Proc. Instn. Mech. Engrs, Part A, 1991, 205, 161.

[4] Vaughan, N. D., Johnston, D. N. and Edge, K. A., Numerical simulation of fluid flow in poppet valves. Proc. Instn. Mech. Engrs, Part C, 1992, 206, 119-127.

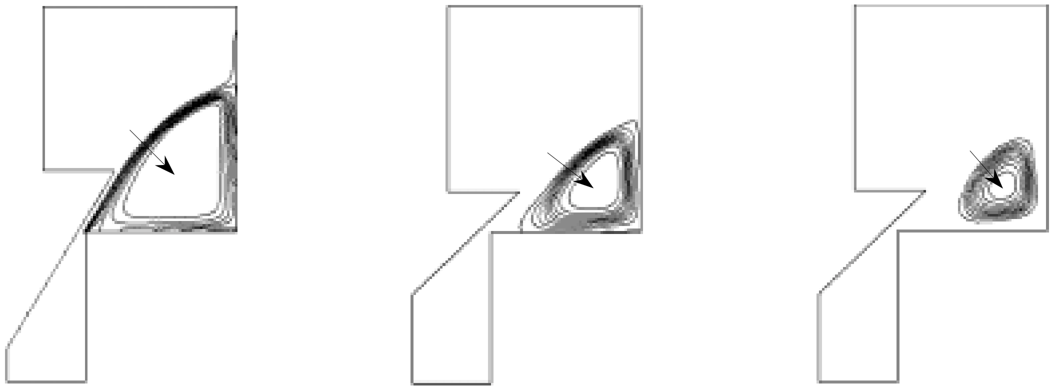
[5] Ueno, H., Okajima, a., Tanaka, H. and Hasegawa, t., Noise measurement and numerical simulation of oil flow in pressure control valves. JSME International Journal Series B, 1994, 37(2), 336-341.

[6] Henrik L.Sørensen, Experimental and numerical analysis of flow force compensation methods for hydraulic seat valve. The Sixth Scandinavian International Conference on Fluid Power 1999, Tampere, Finland, 471-482.

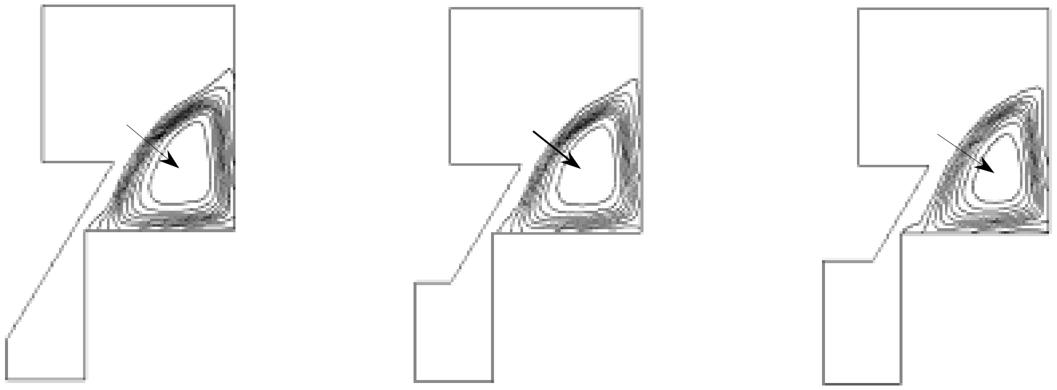


(a) Simulated velocity vectors (b) Contours of velocity magnitude (c) Contours of total pressure

Fig. 3 Flow patterns and contours
$\alpha = 45°$, z = 2 mm, v = 5 m/s



(a) $\alpha = 30°$, z = 2 mm, v = 5 m/s (b) $\alpha = 45°$, z = 2 mm, v = 5 m/s (c) $\alpha = 45°$, z = 2 mm, v = 3 m/s



(d) $\alpha = 30°$, z = 3 mm, v = 5 m/s (e) $\alpha = 30°$, z = 3 mm, v = 5 m/s
Diameter of front face = 10 mm (f) $\alpha = 30°$, z = 3 mm, v = 5 m/s
Diameter of front face = 14 mm

Fig. 4 Contours of volume fraction of water vapor