International Journal of Heat and Mass Transfer 111 (2017) 1–16

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

A numerical investigation in characteristics of flow force under cavitation state inside the water hydraulic poppet valves

Mingxing Han, Yinshui Liu *, Defa Wu, Xufeng Zhao, Huaijiang Tan

State Key Laboratory of Digital Manufacturing Equipment and Technology, School of Mechanical Science and Engineering, Huazhong University of Science and Technology, Wuhan 430074, China

ARTICLE INFO

Article history:
Received 29 November 2016
Received in revised form 14 March 2017
Accepted 28 March 2017

Keywords:
Poppet valve
Cavitation
Flow force
Water hydraulic

ABSTRACT

Compared with the oil hydraulic systems, cavitation is more serious in water hydraulic systems due to the higher saturated vapor pressure of water. Poppet valve is one of important hydraulic components and cavitation is easy to happen due to the sharp pressure drop caused by throttling. This paper presents a numerical investigation into the flow force and cavitation characteristics inside water hydraulic poppet valves. Three kinds of typical structures of poppet valves are selected in the research. The effects of geometric parameters and backpressure of the poppet valves on flow characteristics, cavitation and flow force have been analyzed. Considering the axisymmetric structure of the valves, a half of 2D mixture model is selected and a two-phase mixture model is adopted in the calculation. The accuracy of the numerical models has been validated by comparing the simulation results with the experiment data. The results reveal that two-stage throttle valve (TS valve) can effectively suppress the occurrence of cavitation while the flow force of TS valve is much bigger than that of other valves. By comparing the simulation results under two different boundary conditions (including with backpressure and without backpressure), it seems that cavitation could slightly decrease the flow force.

1. Introduction

Water hydraulic systems that use water as a pressure medium could be a good solution for the environmental and safety problems of the most oil hydraulic systems. Water hydraulic systems have been widely used in the fields of steel and glass production, ocean exploration, food and medicine processing, and coal mining [1–5].

Since the saturated vapor pressure of water is higher than that of oil, cavitation is much easier to occur in water hydraulic components, especially in throttle valves. In the water hydraulic throttle valves, the cavitation bubbles are formed when the static pressure of fluid drops below the vapor pressure at a certain temperature. The formation and collapse of bubbles will reduce the lifespan of water hydraulic valves, lead to harsh noise, severe vibration and also result in local high temperatures and pressure. Therefore, cavitation will result in more power consumption. Previously, some valuable studies on cavitation phenomena inside the throttle valves were carried out by experimental and numerical methods. Oshima et al. [6] indicated that the sharp edged seat valve is better than the one with a chamfered seat in reducing cavitation. Liang et al. [7,8] studied the effects of inlet pressure fluctuations on the cavitation characteristics in poppet valves and the effects of cavitation on momentum thrust in water jet propulsion system of autonomous underwater vehicles separately. Liu et al. [9,10] and Nie et al. [11] focused on the investigation of cavitation characteristics in the two-stage poppet valve through experimental and numerical method. Amirante et al. [12] evaluated the effects of cavitation on the performance of a proportional valve by means of experimental and numerical investigation.

Previous research has shown that cavitation phenomenon has a significant influence on the performance of the hydraulic valves. And most of the previous work has focused on the influence of cavitation on the flow rate, flow coefficient, noise and erosion. However, as so far, very little research regarding the influence of cavitation on the flow force has been done. The variation of flow force is nonlinear in the poppet opening and closing process, and it has great effects on the precise control of proportional and servo valve [13]. This paper aims at studying characteristics of flow force under cavitation state, and thus quantifying the effects of cavitation upon the flow force acting on the cone. The slide valve has been selected as the research object to study the flow characteristic because it was widely used in oil hydraulic system. However, among various types of valves (slide valve, ball valve, disc valve, and poppet valve, etc.), poppet valves have obvious advantages

* Corresponding author.
E-mail address: liuwater@hust.edu.cn (Y. Liu).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.03.100
0017-9310/

such as no-leakage, strong anti-pollution capability, and ease of manufacture when water is used as a pressure medium [14–16]. Considering the typicality of poppet valves, this paper will select three kinds of typical structures as the research object.

The CFD (computational fluid dynamics method) has become an important tool to simulate the characteristic of the flow field inside the hydraulic valves (pressure distribution, velocity field and volume of vapor, etc.). Many researchers have done a lot of studies on the cavitation or flow force by using CFD, and verified the accuracy of the results of CFD simulation through the relevant experiments [13,17,18]. Cho et al. [19] studied the pressure distribution on the surface of a valve plug by CFD simulation. Lisowski et al. [20] proposed a method for calculating the flow forces using CFD method on a three-dimensional model and reduced the flow forces by introducing additional parallel and compensatory channels inside the body of the valve. Hutli et al. [21] indicated that the cavitation process is deeply affected by the flow structure and pressure distribution. However, the detailed correlations between cavitation and flow force inside the water hydraulic poppet valves still remain unclear. In the present work, the cavitation phenomena and flow force characteristic inside three different poppet valves have been investigated through computational flow analysis using CFD software Fluent 14.0, and the effects of dynamic parameter (backpressure with various pressure-difference and valve opening) and geometric parameters (cone angles) on cavitation and flow force are analyzed. Based on the CFD simulation results (pressure distribution, velocity field and volume of vapor, etc.), this paper attempts to further investigate the flow force characteristics under cavitation state, and study the influence of cavitation on flow force inside the poppet valves. This investigation will provide guidance for the design and application of water hydraulic valves both in reducing flow force and avoiding the occurrence of cavitation.

2. Mathematical modeling

CFD calculation is based on the basic governing equations of the fluid dynamics (including Flow Continuity equation, Navier Stokes equation and Energy Conservation equation, etc.) for a given geometry model with specified conditions on the boundary of models. Among the variety segregated algorithms available in Fluent (SIMPLE, SIMPLEC, PISO), the SIMPLE algorithm was selected to solve the governing equations since its memory-efficient, robustness and successful implementation in many cases [8,12,22]. Before the CFD simulation can be carried out, the type of flow inside the valve must be determined. Due to the small opening of the water hydraulic poppet valves in this study (the maximum valve opening is 1.2 mm), there are no conditions for the formation of laminar flow and the cavitation phenomenon occurs inside the valve. Therefore, a turbulent flow model and multiphase model have been applied in this work.

2.1. Model of turbulence

Fluent 14.0 software allows a turbulence model to be chosen from a variety of models available including $k-\varepsilon$, $k-\omega$ and Reynolds. Many studies [23,24] indicated that there is little difference among these turbulence models in predicting cavitation inception for the turbulent flows simulation. In this study, the Realizable $k-\varepsilon$ model is employed for the simulation study. The turbulence kinetic energy k and the dissipation rate ε are given as:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{1}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}(\rho\varepsilon u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\rho C_1 S\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b+S_\varepsilon \tag{2}$$

Where $C_1=\max[0.43,\ \frac{\eta}{\eta+5}]$, $\eta=S\frac{k}{\varepsilon}$, $S=\sqrt{2S_{ij}S_{ij}}$. In the equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. C_1 and $C_{1\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms. Turbulent viscosity μ_t is calculated as follows:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{3}$$

where C_μ is a constant. The model constants are: $C_{1\varepsilon}=1.44,\ C_2=1.9,\ \sigma_k=1.0,\ \sigma_\varepsilon=1.2$.

2.2. Multiphase model

In this study, two-phase flow is discussed. Water has been set as the main phase, and the second phase is vapor. The mixture considered as a 'single phase' [25,26]. The continuity and momentum equations can be given as [27]:

$$\frac{\partial\rho}{\partial t}+\frac{\partial\rho u_j}{\partial x_j}=0 \tag{4}$$

$$\frac{\partial\rho u_i}{\partial t}+\frac{\partial\rho u_i u_j}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\left[(\mu+\mu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)\right] \tag{5}$$

where δ_{ij} is the Kronecker symbol $\delta_{ij}=1$ for $i=j$. The mixture density ρ and viscosity μ are:

$$\rho=\alpha\rho_v+(1-\alpha)\rho_l \tag{6}$$

$$\mu=\alpha\mu_v+(1-\alpha)\mu_l \tag{7}$$

where ρ_v and ρ_l are vapor and liquid density respectively. μ_v and μ_l are vapor and liquid kinetic viscosity respectively. The vapor volume fraction α can be got by: $\alpha=f\frac{\rho}{\rho_v}$. f is the vapor mass fraction.

2.3. Model of cavitation

The cavitation occurs when the static pressure of fluid drops below the vapor pressure at a certain temperature. The mixture flow is considered as a homogeneous vapor-liquid mixture [26]. When the cavitation occurs, the liquid-vapor mass transfer (evaporation and condensation) is governed by the vapor transport equation [28]:

$$\frac{\partial}{\partial t}(\alpha\rho_v)+\nabla\cdot(\alpha\rho_v\vec{V}_v)=R_e-R_c \tag{8}$$

where $\vec{V}_v$ is the vapor velocity, R_e and R_c are the mass transfer source terms respectively. Assuming that all the bubbles in a system have the same size, Zwart et al. [29] got the final form of the cavitation model as follows:

$$\begin{cases} R_e=F_{vap}\frac{3\alpha_{nuc}(1-\alpha_v)\rho_v}{\Re_B}\sqrt{\frac{2(p_v-p)}{3\rho_e}} & if\ p\leqslant p_v \\ R_c=F_{cond}\frac{3\alpha_v\rho_v}{\Re_B}\sqrt{\frac{2(p-p_v)}{\rho_l}} & if\ p\geqslant p_v \end{cases} \tag{9}$$

where the bubble radius is $\Re_B=10^{-6}$ m, the nucleation site volume fraction is $\alpha_{nuc}=5\times10^{-4}$, the evaporation coefficient is $F_{vap}=50$ and the condensation coefficient is $F_{cond}=0.01$. Singhal et al. [28] indicated that the phase change threshold p_v could be given by: