Energy Conversion and Management 100 (2015) 370–379

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Cavity shedding dynamics in a flapper–nozzle pilot stage of an electro-hydraulic servo-valve: Experiments and numerical study

Shengzhuo Zhang, Songjing Li *

Department of Fluid Control and Automation, Harbin Institute of Technology, Box 3040, Science Park, No. 2, Yikuang Street, Nangang District, Harbin 150001, PR China

ARTICLE INFO

Article history:
Received 28 December 2014
Accepted 16 April 2015
Available online 22 May 2015

Keywords:
Electro-hydraulic servo-valve
Flapper-nozzle pilot stage
Cavitation shedding
Pressure oscillation

ABSTRACT

The performance of an electro-hydraulic servo-valve is significantly influenced by the characteristics of the flow field in the flapper–nozzle pilot stage. Cavitation and pressure oscillations frequently occur in the flow field of flapper–nozzle pilot stage and these undesired flow-induced phenomena commonly lead to the appearance of high frequency noises and vibrations of the servo-valve. To obtain in-death understanding about these flow-induced phenomenon, numerical and experimental study of the unsteady cavitation phenomenon in a flapper–nozzle pilot valve is carried out. Large Eddy Simulation (LES) coupling with Schnerr and Sauer mass transfer cavitation model is utilized to simulate the unsteady cavitation shedding around the sharp edge of the flapper. The simulations are conducted using the commercial CFD code ANSYS/FLUENT 14. Meanwhile, the flow field is experimentally observed by using a high speed video camera. The recorded images of the transient cavitation patterns are verified with CFD simulation results. Then, the characteristics of pressure oscillations at the beginning and the end of the shedding path in the flow field are evaluated by CFD approach. The results show that the increment of inlet pressure intensifies cavitation in the flapper–nozzle pilot stage and induces the shedding phenomenon. And both the frequency and magnitude of pressure oscillations in the flapper–nozzle valve are enhanced by increasing the inlet pressure.

1. Introduction

Electro-hydraulic servo-valve is the critical component in most hydraulic control systems, since the control precision of the system always depends on the working performance of the servo-valve. In practice, a lot of present servo-valves adopt twin flapper–nozzle type for the pilot stage which mainly includes two nozzles and one flapper. According to the special structure of the pilot stage, the flow field within it has some distinct features such as small scale, high speed and high shearing. In some specific working conditions flow with cavitation could even occur, which is one of the most substantial factors affecting the performance of the servo-valve. More importantly, the pressure oscillations caused by the collapse of the cloud cavitation could not only disrupt the stability of the flapper but damage the surface of the nozzle and flapper. Therefore, investigating the cavitating flow especially the induced unsteady behaviors in the pilot stage is crucially significant to the structure optimization or performance improvement of the servo-valve.

A number of attempts have been made to study the working characteristics of the servo-valve in order to avoid the failure of the control process. The effect of Reynolds number on flow rate in the main stage was investigated by Pan et al. [1], the flow rate coefficient curve was fitted and matched well with the experiment results. Li and Song [2] reported that the dynamic response of the torque motor in the servo-valve could be improved significantly by using the magnetic fluids. Since neither of them worked on the properties of the flow field in the pilot stage, the influence of the cavitating flow on the performance of the servo-valve has not been released until Aung and Li's research [3], which analyzed the jet flow field and cavitation pattern in pilot stage of a servo-valve with different Reynolds numbers. One of their observations was that the jet flow attached the surface of the flapper with low *Re* and it separated and became free jet when the *Re* getting higher. Furthermore, the sheet cavity in the flow filed was observed both numerically and experimentally. However, only the steady characteristics of the cavitation under different conditions were studied in their research, such as the variation of the shape and size of the cavitating area with *Re*. Although understanding the mechanism of cavitation in hydraulic components and systems is always a hot topic in fluid dynamics [4–7], little research has been done in the area of servo-valve. Particularly the unsteady behaviors of the

---

* Corresponding author. Tel.: +86 1 379 663 5768; fax: +86 451 86418318.
E-mail addresses: zsz2046@163.com (S. Zhang), lisongjing@hit.edu.cn (S. Li).

http://dx.doi.org/10.1016/j.enconman.2015.04.047
0196-8904/

Nomenclature

Latin letters

C_s	Smagorinsky constant
d	distance to the closest wall
g	gravitational acceleration
h	local grid scale
k	von Karman constant
L_s	mixing length for sub-grid scale
n	bubble number density
p	pressure
R	mass transfer rate
S	rate of train tensor
u	velocity of fluid
t	time
x	Cartesian coordinate

Greek letters

α	volume fraction
δ	Kronecker delta
μ	dynamic viscosity of fluid
μ_t	turbulent viscosity
ρ	density of fluid
τ	sub-grid stress tensor

Subscripts

c	condensation
e	evaporation
i, j	Cartesian tensor indices
l	liquid
m	mixture
v	vapor

cavitating flow in the pilot stage and its interaction to the performance of the servo-valve are presently unclear.

At the same time, the unsteady cavitating flow has been investigated extensively in many other areas with various approaches [8,9]. Stanley and Barber [10] investigated the re-entrant jet mechanism for periodic cloud shedding cavitation experimentally, which showed a constant presence of a liquid sublayer between the cloud cavitation and the nozzle wall. The travelling pressure wave driven by the previous bubble collapse was regarded as the reason of the periodic break off and shedding of the cavity. Danlos et al. [11] applied the Proper Orthogonal Decomposition (POD) method to sequences of sheet cavity images, in order to identify the cavitation regimes (sheet cavity or cloud cavitation regimes). The results showed that compared to energy content of the first mode, the phase portraits and Lissajous figure were more significant to detect different regimes. In addition, the shedding frequency of cavitation clouds in the submerged water jet was measured by several researchers. Hutli and Nedeljkovic [12] concluded that the shedding frequency decreased with the increasing injection nozzle pressure. Although Nishimura et al. [13] arrived at the opposite results, both of them indicated that the shedding frequency could be as high as thousands of Hertz.

Since the periodic shedding of the cloud cavitation is often accompanied by high frequency pressure fluctuation, which is considered to be a destructive form of cavitation and could cause damage to hydraulic machinery [14,15], it is essential to investigated the dynamic characteristics of cavitating flow from the perspective of induced pressure fluctuations. Leroux et al. [16] measured the pressure fluctuations on the surface of a NACA66 foil under different cavitation regimes (stable and unstable). The frequency of the pressure oscillations induced by the unstable cavitation shedding was found to be lower than the frequency with stable cavitation. Dai et al. [17] studied the pressure fluctuations caused by cavity shedding coupling with shear layer oscillations in a square surface cavity, which showed that the frequency of pressure fluctuations caused by bubble collapse was higher than that induced by shear layer oscillation.

Due to the rapid improvement of the computing abilities and limitations of measurement techniques, large amount of effort has been put into developing numerical models to predict unsteady cavitating flow. As the behaviors of cavitation and turbulence are all essentially unsteady in nature, it indicates that a strong bond often exists between them [18]. Therefore, the simulation accuracy of the cavitating flow is directly affected by the property of the turbulence model. Compared to the traditional RANS approach, Large Eddy Simulation has been proven to be superior to simulate the unsteadiness of the cavitation, as it is often more capable of reproducing large unsteadiness motion of the flow field, resulting in only the small unresolvable scales being modeled [19,20]. Investigations of the unsteady cavitating flow with LES scheme have been the subjects of numerous computational studies, such as cavitating flow around different hydrofoils or cavitation induced by various hydraulic devices [21–23]. Based on the present researches and theories, it can be concluded that LES can provide greater accuracy of the simulation for unsteady cavitating flow.

Thus the idea of this work is to simulate the unsteady cavitation phenomenon using the LES scheme and a mass transfer cavitation model in the pilot stage of the servo-valve. The whole process of the cavitation formation, shedding and collapse is carried out under different boundary conditions. And the induced pressure oscillations are also calculated aiming to find out the reason for the damage of the servo-valve components.

2. Working principle of flapper–nozzle pilot stage

A close-up schematic illustration of the two-stage electro-hydraulic servo-valve is shown in Fig. 1. It basically consists of the following 3 parts, the torque motor, the pilot stage and the main spool. As shown in Fig. 1, without working current the flapper stays at the neutral position, which means the distances between flapper and two nozzles are the same. Hence the flapper may balance the pressure on both sides of the main spool. Once the working current is applied on the coils, the torque motor will produce a proportional toque, force the flapper move closer to one nozzle. As a result, the flow area of the other nozzle will increase which makes a pressure difference between the two nozzles. Since the nozzle is linked with the main spool chamber, the main spool will move in the opposite direction of the flapper and have a certain opening. On the other hand, the flapper is connected with the main spool by the feedback spring. Thus, the flapper will return to the neutral position driven by the main spool until the pressure on the two sides of the main spool rebalances. Consequently, one input working current of the torque motor