Computers & Fluids 88 (2013) 590–598

Contents lists available at ScienceDirect

Computers & Fluids

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

Experimental and numerical investigation of cavitation phenomenon in flapper–nozzle pilot stage of an electrohydraulic servo-valve

Songjing Li *, Nay Zar Aung, Shengzhuo Zhang, Junzhang Cao, Xinzhi Xue

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 9 June 2013
Received in revised form 6 September 2013
Accepted 15 October 2013
Available online 22 October 2013

Keywords:
Cavitation
Flapper
Nozzle
Pilot stage
Electrohydraulic servo-valve

ABSTRACT

Cavitation in the flapper–nozzle pilot stage is an important source for the noise, performance deterioration and even failure of electrohydraulic servo-valves. In this paper, experimental and numerical investigations of cavitation phenomenon appearing in the flow field between the flapper and nozzle of an electrohydraulic servo-valve are carried out. Experimental observations are conducted with variation of Reynolds number ranging from 630 to 2500 based on the nozzle inlet velocity and diameter. Images of cavitation phenomenon in the flow field are recorded and compared with CFD simulation results to confirm the occurrence and locations of cavitation sources. The computed numerical results show a good agreement with experimental observations. From both types of results, the nozzle inner wall tip, nozzle outer wall tip and flapper leading edge are shown as the locations of cavitation sources. At flow conditions with lower Reynolds numbers, onset cavitation and inception are found at the nozzle outer wall and the flapper leading edge. Further increasing of Reynolds numbers creates a separated flow and then jet flow. Attached cavity is found on the flapper curved surface together with the separated flow and cloud-like cavitation comes with the jet flow. Since numerical results can confirm all of the recorded observations, a reliable computational scheme is also provided by this paper.

1. Introduction

Electrohydraulic servo-valves are the focal point of discussion when the failures of hydraulic control systems occur. Since these devices are composed of electrical and mechanical parts, understanding of fatigue effects in those fields is necessarily important for the reliable design of the devices. Cavitation is the rapid formation of vapor bubbles caused by a transient reduction in local pressure field and it is a frequently encountered problem in hydraulic control devices. Usually, the collapse of cavitation bubbles subsequent to pressure recovery generates high-speed fluid jets with considerable energy that may damage the surface of hydraulic components [1,2]. Flow in the flapper–nozzle pilot stage of a servo-valve (Fig. 1) consists of orifice flow (vena contract), impinging jet flow and flow over bluff body where rapid pressure change mostly occurs. Therefore, the occurrence of cavitation phenomenon in the flow field of the pilot stage is very potential. Thus, damages by cavitation on the flapper, feedback spring and nozzles of servo-valves have been detected in many of practical applications.

Since understanding and prediction of cavitation phenomenon in hydraulic components and systems is crucially important, a number of intensive researches regarding cavitation behaviors in hydraulic components have been done. Martin et al. [3] conducted investigations of cavitation in spool valves with respect to valve opening, Reynolds number and dissolved gas content. One noticeable conclusion was that the effect of dissolved gas was minimal on cavitation if the content did not exceed that at atmospheric condition. After dealing with cavitation in a hydraulic poppet valve, Washio et al. [4] observed that flow separation and solid contact were two mechanisms of cavitation. From Lu's research about cavitation in spool valve [5], it was found that spiral cavity was formed by the swirling flow and the collapse of bubbles in the wake of spiral cavity produced cavitation noise.

Some other studies also showed that cavitation was also governed by geometries of devices and flow conditions. Gao et al. [6] investigated the cavitation phenomenon near the orifice of a poppet valve experimentally and numerically. Their results showed that the reduction in outlet area and the increase in outlet pressure could effectively suppress the cavitation. In the work carried out by Sandor and Suan-Resiga [7], it was observed that higher inlet pressures would produce more intense and more damaging cavitation. Annie Claude et al. [8] studied a cavitating flow in a flash valve numerically with and without cavitation model. He discussed that higher inlet pressure increased the cavitation intensity. Chern et al. [9] numerically investigated cavitation phenomenon in a globe valve with and without cages. They reported that cavitation appeared in the vortices existing inside the valve and at the

* Corresponding author. Tel.: +86 1 379 663 5768; fax: +86 451 86418318.
E-mail addresses: lisongjing@hit.edu.cn (S. Li), nay1572@gmail.com (N.Z. Aung), zsz2046@163.com (S. Zhang), cjz_mouse@126.com (J. Cao), rexxue@gmail.com (X. Xue).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.10.016

Nomenclature

Symbols

F	body force
f	phase fraction
g	gravitational acceleration (m s^{-2})
k	turbulent kinetic energy
p	pressure (Pa)
Re	Reynolds number
u	velocity of fluid (m/s)

Greek letters

ε	turbulent dissipation rate
ρ	density of fluid (kg/m^3)

Subscripts

dr	drift
m	mixture
v	vapor
l	liquid
g	gas
sat	saturation
t	turbulent

downstream region for the condition without cages, but disappeared in those regions for the condition with cages. Cavitation in a spool valve with U-grooves was studied by Zou et al. [10]. One of their observations was the cavitation inside the valve becoming more violent at deeper groove depth. Although characteristics of hydraulic valves with different structures were studied with respect to the cavitation phenomenon, investigations for the flapper–nozzle structure are still limited.

According to the structure of flapper and nozzles, the flow field in the pilot stage of servo-valves is mostly governed by the orifice flow, impinging jet flow and squeezed flow. In the survey of those interests, from the experimental research of Sun et al. [11] on the cavitation erosion of a water-jet flow, it was concluded that the distance to targeted object, nozzle diameter and pressure of water jet played key roles in cavitation. Lim et al. [1] observed that impact velocity, contact surface area and squeeze flow velocity between two surfaces were important for the appearance of cavitation. Some recent numerical works also have paid attention to the flow phenomenon in servo-valves. Mchenya et al. [12] studied the flow field distribution and characteristics of the discharge coefficient in the pilot stage of servo-valves. They could find a linear relationship between inlet pressure and flow rate. Zhang et al. [13] also numerically investigated the flow filed in a flapper–nozzle valve. He explained that low pressure around the flapper would cause cavitation. All the information above provided either by numerical study or experimental investigations has partially fulfilled the understanding and imagination of flow related phenomenon in servo-valves. However, identifying and understanding of cavitation characteristics in the flapper–nozzle pilot stage through a peer investigation are still critically demanded in developing reliable designs of servo-valves.

Thus, experimental and numerical investigations of cavitation phenomenon in the flapper–nozzle pilot stage of an electrohydraulic servo-valve are conducted in this work. Cavitation phenomenon in a flapper–nozzle pilot valve with various inlet velocity and outlet pressure settings are investigated to understand and characterize the cavitation phenomenon with respect to the given flow conditions.

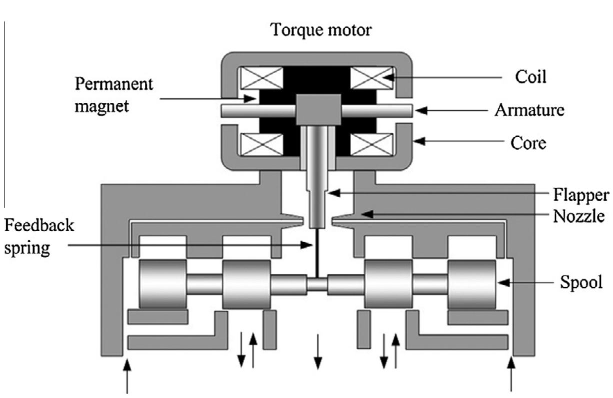

Fig. 1. Schematic of flapper–nozzle pilot stage in an electrohydraulic servo-valve.

2. Working principle of flapper–nozzle pilot stage

Two-stage electrohydraulic servo-valves are most common in practical applications where accurate position control is required. In such servo-valves, flapper–nozzle or jet pipe acts as the pilot (the first) stage to control the pressure acting on both sides of the main spool. This pilot stage is controlled by a torque motor through electrical signals. A close-up schematic illustration of the flapper–nozzle pilot stage is shown in Fig. 1.

The flapper–nozzle pilot stage consists of a toque motor, a flapper, two nozzles and a feedback spring, as shown in Fig. 1. The flapper is connected with the armature and controlled by the output torque proportional to the applied electric current of the torque motor. When the torque is applied, the flapper moves closer to one nozzle decreasing the flow area through this nozzle and lets that of the other nozzle increase. Hydraulic oil is jetted out from both of the nozzles to the flapper and goes back to the return pipe whose pressure is nearly atmospheric pressure through the outlet of the pilot stage. The inlet pressures of the nozzles working on both sides of the main spool are built up due to the jet flow. The pressure and velocity of the jet flow change with the flow area between the flapper and nozzle. Therefore, the movement of the flapper creates a pressure difference between two ends of the main spool which attains a movement to the lower pressure side.

The pressure and velocity distributions in the flow field of the jet flow influence the pressure difference and the movement of the main spool significantly. As the return pressure of the jet flow is nearly the atmospheric pressure, the pressure inside the flow field may become lower than the vapor saturation pressure of hydraulic oil and cavitation will appear inside the flow field. The appearance of cavitation in the flow field between the flapper and nozzle may bring noise and damages to the valve. And thus, the flow efficiency and the performance of the pilot stage will be deteriorated. Therefore mechanism of the cavitation phenomenon should be investigated to improve the performance of servo-valves.

3. Numerical methods

In recently years, Computational Fluid Dynamics (CFD) has been combined with experiments in many works to analyze cavitation in hydraulically operated devices and acceptable results have been found [14–16]. The development of powerful CFD tools also enhances the possibility of achieving unlimited level of details about the behavior of the flow [17]. In this study, aiming to have a