Energy Conversion and Management 124 (2016) 104–115

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Effects of structure parameters on flow and cavitation characteristics within control valve of fuel injector for modern diesel engine

Chao Wang [a], Guo-Xiu Li [a,*], Zuo-Yu Sun [a], Lan Wang [a], Shu-Ping Sun [b], Jiao-Jiao Gu [b], Xiao-Jun Wu [b]

[a] School of Mechanical, Electronic and Control Engineering, Beijing Jiaotong University, Beijing 100044, China
[b] China North Engine Research Institute, Tianjin 300400, China

ARTICLE INFO

Article history:
Received 21 January 2016
Received in revised form 17 May 2016
Accepted 3 July 2016
Available online 9 July 2016

Keywords:
The effect of structure parameters
The flow and cavitation characteristics
Control valve
Numerical simulation

ABSTRACT

Cavitation is a common phenomenon in diesel injector and has a strong influence on the internal flow. However, studies so far have focused on cavitation characteristics inside the nozzle. Its influence on the flow during control valve opening remains still unclear. In the paper, a computational study focused on the flow and cavitation phenomena within control valve has been reported and the effects of control valve's structure parameters (including rounded edge, seal cone angle and outflowing control-orifice structure) on the flow and cavitation characteristics have been investigated in detail.

Firstly the 3D model has been validated in terms of single injection quantity and fuel injection duration, showing a good consistency. And then, the development from sheet cavitation to cloud cavitation and the relationship between cavitation, pressure and velocity has been discussed. Based on the numerical results obtained, it is shown that not only the variation of pressure but also the velocity is the important factor which affects cavitation. The increase of the flow velocity reduces the pressure within the flow field which can aggravate the development of cavitation. As cavitation region increases, the fuel flow is hindered and the flow velocity decreases. However, the decrease of flow velocity has suppressed the development of cavitation. All of those variations form a cyclical process.

1. Introduction

Due to high thermal efficiency, wide power scope, low fuel consumption and reliability, diesel engine is widely applied in various fields, such as machinery, marine technology and automobile industry [1–3]. Among the components of diesel engine, fuel injection system is the core [4–6]. Fuel injector is the most complex core part of the fuel injection system, which plays important role in fuel injection and its control process. The structure and flow characteristics of fuel injector can affect the performance of the whole fuel injection system, thereby affecting the impact performance of diesel engine [7–9].

The flow and cavitation characteristics of fuel are the two main factors affecting the performance of fuel injector. Therefore so many investigations pay attention to this field [10–14]. Salvador et al. [7] study the flow and cavitation characteristics of fuel inside the nozzle under different degrees of needle lift using numerical methods. It has been reported that, for different degree of needle lift, cavitation domain inside the nozzle changes and mass flow rate around the outlet of nozzle will be influenced. Molina et al. [9] compared the flow and cavitation characteristics inside the round and the oval nozzles employing numerical modeling method. Its results show that, inside the oval nozzle, cavitation happens rarely and discharge coefficient is lower, whereas inside the round nozzle the phenomenon is completely opposite that cavitation happens easily and discharge coefficient is higher. Wang and Su [10] studied the unstable cavitation process caused by pressure fluctuation inside the nozzle and showed that the occurrence and development of cavitation will be delayed since process depends not only on the local pressure but also on the flow rate caused by pressure difference. Sun et al. [15] employed Quasi-steady method to study the flow and cavitation characteristics of fuel inside the nozzle and pointed out the factors affecting the rate of fuel injection and mass flow rate. Salvador et al. [16,17] compared the flow and cavitation characteristics of fuel around the nozzle of microsac and VCO (different types of diesel engines) and asserted that the influence of various nozzle and nozzle chamber on those characteristics is different.

Studies so far discussed above have focused on the flow and cavitation characteristics around the nozzle. However, few studies have investigated the flow and cavitation characteristics within the control valve. In fact these characteristics inside control valve have

* Corresponding author.
E-mail address: Li_guoxiu@yahoo.com (G.-X. Li).

http://dx.doi.org/10.1016/j.enconman.2016.07.004
0196-8904/

Nomenclature

$\Re_B$	bubble radius	α_v	volume fraction of vapor
P_B	bubble surface pressure	R	net mass source term
P	local far-field pressure	v_l	kinematic viscosity of the surrounding liquid
P_v	saturation vapor pressure	S	surface tension of the bubble
n	bubble number per unit volume of liquid	ρ_v	vapor density
R_c	mass source terms connected to the collapse of the vapor	ρ_l	liquid density
R_e	mass source terms connected to the growth of the vapor	ρ_m	mixture density

a direct influence on response characteristics, fuel injection quantity, fuel injection pulse width and fuel injector rate, eventually influence on the power and emission performance of the diesel engine. Until recent years, the researches of flow and cavitation characteristics inside control valve have got the attention of scholars. It is difficult to perform the experiments on the flow and cavitation characteristics inside the control-valve. Therefore, determining the cavitation and its occurrence area can be done only by observing the component damage in the control-valve due to cavitation.

Ferrari et al. [18] have studied the effects of bypass, pressure fluctuation and different driving electric current on the quantity of fuel injected. Duan et al. [19,20] have observed the damage inside the control-valve caused by cavitation through scanning electron microscopes. As demonstrated in Fig. 1(a), the inner wall of the control-valve is damaged more seriously with many small irregular holes. In order to demonstrate the intensity of this kind of damage, the 1000-h reliable experiment about fuel injector has been conducted and results are shows in Fig. 1(b). The damage area is larger than the tangent area of the ball valve and the wall, and the flow characteristics around the sealing area of control-valve provide the possibilities of cavitation. Therefore, cavitation has an important role in the damage of injector.

Previous works principally focused on the simulation study of control valve of injector or experimental analyses of cavitation phenomenon. But there are no sufficient studies on the demonstration of the structure's effects on flow characteristics and cavitation process within control valve. Therefore, the flow and cavitation characteristics are studied with the numerical simulation method in this paper. Using 3D model, the effect of structure parameters (including rounded edge, seal cone angle and OA structure) on the flow and cavitation characteristics within the control valve are studied systematically.

2. Simulation method and related settings

2.1. General settings

Fig. 2 shows the structure of fuel injector. Based on the functionality, it can be divided into electromagnetic valve component, control valve component and fuel injector nozzle component. Among the three sections, the control valve component (composed of ball valve, inflowing control-orifice (OZ), outflowing control-orifice (OA), control chamber and control-piston) controls the opening and closing of the needle valve by the hydraulic effect.

The simulation calculations have been performed by ANSYS Fluent [21–23]. The internal flow is simulated by the dynamic mesh technique (combination of structured and unstructured grid) during the opening of the control valve. Total number of meshes is 943,131 with a maximum size of 0.08 and a minimum size of 0.008. Some of the specific parameters are presented in Table 1. Duo to the limitation of software, the number of layers of grid between the ball valve and the valve seat are kept as two. Therefore, the ball valve starts to move 0.02 mm ahead the valve seat and lifts upwards 0.06 mm during 0.1 ms (see Fig. 3).

2.2. Cavitation model

In most engineering situations we assume that there are plenty of nuclei for the inception of cavitation. Thus, our primary focus is on proper accounting of bubble growth and collapse. In a flowing liquid with zero velocity slip between the fluid and bubbles, the bubble dynamics equation can be derived from the generalized Rayleigh-Plesset equation as:

$$\Re_B \frac{d^2\Re_B}{dt^2} + \frac{3}{2}\left(\frac{d\Re_B}{dt}\right)^2 = \left(\frac{P_B - P}{\rho_l}\right) - \frac{4v_l}{\Re_B}\frac{d\Re_B}{dt} - \frac{2S}{\rho_l \Re_B} \quad (1)$$

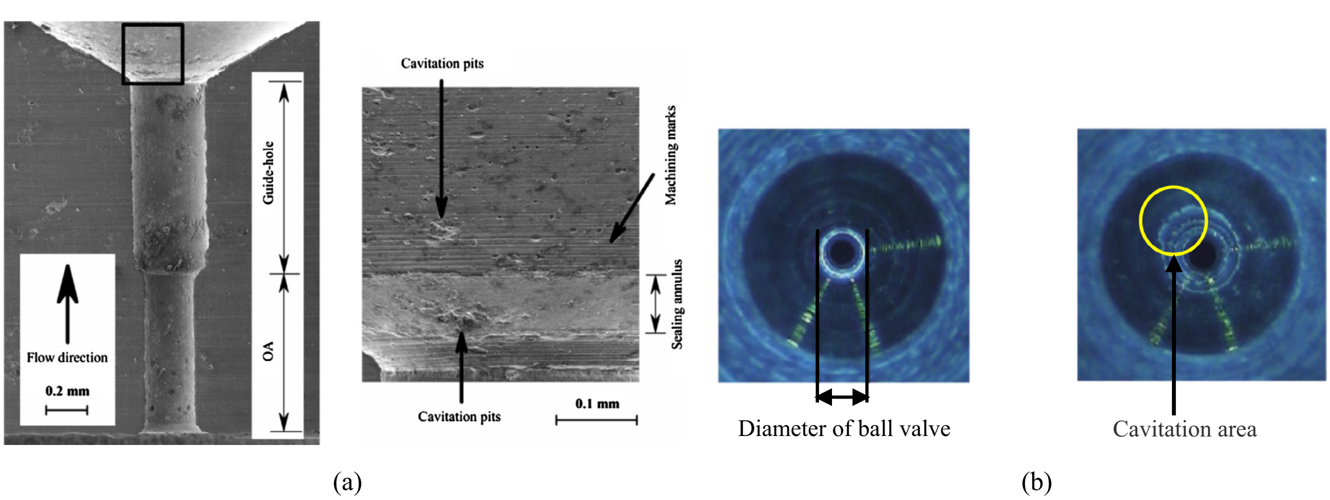

Fig. 1. Photos of cavitation damage on control-valve-seat with scanning electron microscopes (after 1000 h alternating fatigue test) [19,20].