Fuel 217 (2018) 193–201

Contents lists available at ScienceDirect

Fuel

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

Full Length Article

Investigation of the effect of nozzle inlet rounding on diesel spray formation and combustion

Ozgur Oguz Taskiran

Design Project Office, Turkish Naval Forces, Pendik, Istanbul

ARTICLE INFO

Keywords:
Nozzle inlet rounding
Discharge coefficient
Diesel spray modeling
Spray ignition
KIVA

ABSTRACT

In this study, the effect of nozzle inlet rounding on diesel spray formation and combustion is investigated. In order to take into account nozzle geometry, two different nozzle geometries (sharp inlet and rounded inlet) were used in the experiments. Diesel spray injected into a constant volume combustion chamber was modeled by using KIVA3V R2 code. One dimensional nozzle flow model was used in numerical simulations. This model calculates nozzle discharge coefficient and updates the initial droplet radius and droplet velocities according to nozzle inlet geometry. Diesel fuel is represented by diesel oil surrogate mechanism which consists of 71 species and 323 reactions. The numerical spray results were validated with macroscopic spray characterization data obtained by constant volume experiments. Results showed that inlet rounding increases discharge coefficient. At high ambient pressure conditions, sharp and rounded nozzles have similar spray internal structure. However, it is observed that sharp inlet nozzle produces smaller droplets that shorten spray tip penetration and autoignition delay period due to low discharge coefficient. Comparing spray combustion of rounded and sharp inlet nozzles, it is showed that rounded inlet nozzle has lower combustion temperature, less NO and soot concentration than sharp inlet nozzle.

1. Introduction

Many researchers are intensively interested in spray characteristics, ignition and combustion process of diesel spray to reduce harmful emissions and fuel consumption of diesel engines. To enable better fuel and air mixing process, the prominent number of experimental and numerical studies focused on new injection strategies and new nozzle designs. Since combustion chamber geometry is affected by spray characteristics, numerous investigations are still carried out to improve nozzle designs. Nozzle inlet rounding is a primary characteristic of nozzle geometry that affects spray structure. In literature, one of the important studies on sharp and rounded inlet nozzles and their relation on discharge coefficient was presented by Nurick [1]. By making parametric analysis, Nurick states that cavitation occurs due to different nozzle geometries.

Fuel injection pressures of diesel engines have increased drastically by the implementation of common rail injection systems. Higher injection pressures made nozzle geometry more important because it has influence on cavitation and fuel-air mixing process [2–4]. Recent studies presents that cylindrical or conical nozzle geometries have different atomization process due to decrease in effective fluid velocity [5,6]. Angle between nozzle holes was investigated by Salvador and coworkers and a correlation between discharge coefficient and injector nozzle hole angle was suggested [7]. A numerical investigation of nozzle inlet rounding and its effect on cavitation, spray characteristics and atomization was made by Shervani-Tabar et al. [8]. Though their study reveals spray atomization and fuel–air mixture behaviour of nozzle inlet rounding, it does not include spray combustion. Despite there are wide range of nozzle geometry investigations in literature, information about the effect of nozzle geometry on turbulent spray combustion is limited. Since, setting experimental facility and carrying out experiments covering nozzle internal flow and spray combustion is a difficult and expensive process, numerical investigation can give valuable information filling this gap.

Developments on numerical simulations indicate that future engine problems and optimizations can be done more reliable by using numerical modeling programs. Though there are new promising developments on modeling fluid flow in nozzle and atomization outside of it, a numerical code giving complete modeling solutions of an internal combustion engine that deals with the all aspects of fluid flow inside the nozzle, atomization and spray combustion has not been developed yet. In this study, KIVA code (KIVA3V R2) was used in numerical calculations. KIVA, which is a non-commercial CFD program for engine research, describes the spray dynamics and combustion process with its related sub-models [9]. Open source code of KIVA enables to improve sub-models for better modeling outcomes of spray and combustion

E-mail addresses: taskiranoz@itu.edu.tr, ootaskiran@hotmail.com.

https://doi.org/10.1016/j.fuel.2017.12.031
Received 27 August 2017; Received in revised form 25 October 2017; Accepted 5 December 2017

modeling.

KIVA code uses Lagrangian framework for the liquid phase while the continuous gaseous phase is modeled by the standard Eulerian conservation equations [9]. One dimensional flow model of Sarre et al. [2] which is based on theoretical analyses and previously reported experimental results describes possible flow types inside injector nozzle. To investigate effect of nozzle geometry, they have implemented one dimensional flow model in two different CFD codes. The reported results showed that the numerical results can be improved by taking into account nozzle geometries and injection conditions [2]. One dimensional nozzle flow model of Sarre uses experimental and theoretical loss coefficients to calculate nozzle discharge coefficient. Flow velocity and pressure at contraction point are calculated by using Nurrick's [1] expression and Bernoulli equation. According to nozzle discharge coefficient, velocity at the nozzle exit and initial droplet Sauter Mean Diameter (SMD) are calculated [2].

In this study, effect of nozzle inlet rounding on nozzle discharge coefficient, spray structure and combustion was investigated by experimentally. In addition, it is also intended to drive effect of nozzle inlet rounding on combustion by numerical study. Numerical results are validated by basic experimental results in terms of spray penetration and autoignition delay period.

2. Theoretical background and experiments

Discharge coefficient is a basic parameter that gives information about nozzle flow conditions. The discharge coefficient, C_d, defines the ratio of actual and theoretical fuel flow. The actual mass flow of a nozzle, $\dot{m}_f$ is different from the theoretical one because of friction and contraction of the flow in the nozzle. In the recent study of Salvador and co-workers it is stated that discharge coefficient reduces as the angle between nozzle holes increases [7]. Their statistical data presents discharge coefficient may change between 0.85 and 0.95 as the angle between nozzle holes change between 180 and 0. Discharge coefficient is defined as;

$$C_d = \frac{\dot{m}_f}{\dot{m}_{th}} = \frac{\dot{m}_f}{A\sqrt{2.\,\rho_f.\,(P_{inj}-P_{back})}} \quad (1)$$

where A is the area of the nozzle, ρ_f is the fuel density, P_{inj} is the injection pressure and P_{back} is the chamber pressure. In this study, injector nozzle with single hole shown in Fig. 1 was used in the experiments.

Two different nozzle was manufactured for the experiments: sharp inlet nozzle (hole diameter:0.265 mm) and rounded inlet nozzle (hole diameter:0.240 mm). Internal geometry measurements of the nozzles are presented in Fig. 2. The silicone mold measurements showed that rounded inlet has 0.12 mm rounding radius which is obtained by hydro-grinding. The details of the nozzle inlet measurements can be found in [10]. Though sharp inlet nozzle has bigger nozzle diameter, it has the same mass flow rate with the rounded inlet one according to manufacturer data which states that both nozzles has 170 cm^3/30 s flow rate at 100 bar injection pressure.

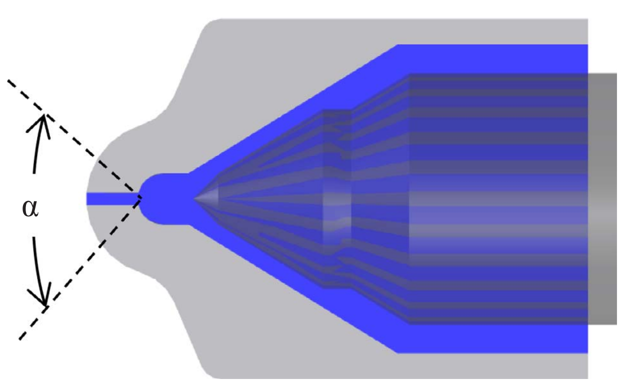

Fig. 1. Shape of injector nozzle.

Discharge coefficient of sharp and rounded inlet nozzles are shown in Fig. 3. Measurements showed that inlet rounding has increasing effect on discharge coefficient. The increase in the discharge coefficient can be attributed to increase in effective nozzle area obtained by inlet rounding. The reason of the obtaining low discharge coefficient than the presented data in the literature can be explained by measurements of the actual mass ratio that are limited by 1.5 ms injection periods. As a result of this, injector opening and closure delays reduced the actual mass flow. However, the nozzles can be compared with each other since the same conditions applied to both of them.

In this study a constant volume combustion chamber 56 mm in diameter, and 85 mm in depth and capable of heating the chamber air up to 850 K was used for spray experiments. Schematic view of the experimental setup is shown in Fig. 4. The injector is mounted on to the center of top cover, spray axis coincide with the chamber axis. The chamber equipped with two opposite 25 mm thick quartz windows which enable a 55 × 25 mm sight view. Compressed air supplied from two high pressure (20 MPa) air bottles via pressure reducers to obtain 2 and 3 MPa test conditions as shown in Fig. 4. A typical common rail system was used for fuel injection. Other details of the experimental setup can be found in [10].

In order to investigate spray internal structures, shadowgraphy images of the spray were divided to 3 grey levels according to grey intensity histogram of the images by using a digital image processing program. By this way, the grey level of the images representing high, medium and low intensity regions were obtained. It is observed that at low ambient pressure, sharp inlet nozzle does not have axi-symmetrical flow structure. This is better visualized at early spray images as presented in Fig. 5. Measurements at high ambient conditions showed that sharp inlet and rounded inlet nozzle has very similar spray structures as presented in Fig. 6. As the ambient pressure increased, the differences between sharp and rounded inlet nozzles vanished and have almost the same area for of high, medium and low regions were obtained.

3. Numerical modeling

One dimensional nozzle flow model was used with KIVA3V R2 code for the numerical calculations presented in this study. In order to compare the nozzle flow model of sharp and rounded nozzle inlet configurations, the same injection duration and injection profile were chosen for both cases. The computed discharge coefficient and effective injection velocity are presented in Fig. 7. One dimensional flow model showed that inlet rounding has significant increasing effect on nozzle discharge coefficient. The results are consistent with the expectations. It is probable that flow is turbulent at the beginning of injection and as the flow velocity increases the discharge coefficient decreases because of the cavitation. Initial SMD of injected droplets varies according to effective nozzle discharge area. Nozzle discharge area reduction has increasing effect on injection velocity. As the flow contraction increases, effective flow area reduces and thus effective velocity increases as presented in Fig. 7.

In numerical calculations, droplet dynamics have to be modeled accurately to obtain correct representation of air-fuel mixing process. Since the droplet diameter distribution, vaporization and penetration are highly effected by droplet break-up model, it is the most effective sub-model among the others. Kelvin Helmholtz and Rayleigh Taylor model (KHRT) was used as droplet break-up model in this study. KHRT, based on Kelvin-Helmholtz instability growing on the droplet surface and instable RT waves growing at the backward direction of the liquid was combined and used as a hybrid model by Beale and Reitz [11]. After that, KHRT breakup model intensively used in modeling diesel spray simulations [12]. Droplet collision modeling is another important research area in spray modeling calculations. Recent experimental studies and numerical models on spray collision are discussed in the