Energy Conversion and Management 154 (2017) 46–55

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Review

An investigation of the validity of a homogeneous equilibrium model for different diesel injector nozzles and flow conditions

Ehsanallah Tahmasebi[a,*], Tommaso Lucchini[a], Gianluca D'Errico[a], Angelo Onorati[a], Gilles Hardy[b]

[a] *Internal Combustion Engine Group, Department of Energy, Politecnico di Milano, Italy*
[b] *FPT Motorenforschung AG, Switzerland*

ARTICLE INFO

Keywords:
Nozzle internal flow
Two-phase flow with cavitation
Homogeneous equilibrium model
Nozzle geometry
Injection pressure

ABSTRACT

In the present work, a methodology for modeling flow behavior inside the fuel injector holes is applied to a number of cases with different geometries and flow conditions. After assessment of the approach results through various experimental studies looking into the flows behavior inside the diesel nozzles, two series of analyses are defined. In the first study, the effect of inlet pressure is investigated by using a series of different rail pressures in both numerical and experimental tests in a single hole industrial injector. Results show a non-cavitating flow and an approximately linear increase of the velocity, turbulence kinetic energy, and turbulence dissipation energy with the increase of pressure difference and linear increase of the mass flow rate with the square root of the pressure difference in this nozzle. The second study is related to the effect of hole geometry on injector performance. The effects of entrance edge rounding and the tube conicity factor are investigated by changing these parameters in a series of geometries from an industrial diesel nozzle. Results show that cavitation occurs in the geometries with a sharper edge and low conicity. The role of the cavitation in emerging flow properties is emphasized in the values of the injector discharge factor and the turbulence properties. The results of this work can be used in the simulation of the primary breakup of fuel spray, and this approach is useful for design and optimization of the injectors for industrial sectors.

1. Introduction

Diesel engines in the modern automotive market are still the best choice for a broad range of applications. Over the past years, advanced injection technologies with higher injection pressure have offered a compromise between emission reduction and fuel consumption which was estimated by Mahr [1] and Baumgarten [2]. A recent report by Morgan et al. [3] has indicated that this approach is still attractive for future diesel engine concepts by using advance techniques as well as injection shaping [4,5]. The effects of injector hole geometry in the primary breakup of high-pressure sprays have been emphasized in several investigations. Schmidt and Corradini [6] have shown that cavitation in an injector causes significant initial disturbances that accelerate the breakup process. Cavitating nozzles are expected to generate the most rapid breakup whereas single phase flow relies mostly on turbulent fluctuations produced in them, creating moderate initial disturbances. Desantes et al. [7] have shown that the presence of cavitation in the nozzle exit improves the turbulence intensity in the spray as well as the atomization and mixing process. Their experimental investigations have shown that in inlet/outlet pressure conditions with cavitation inside the nozzle, the fuel jet cone angle increases. Owing to their test conditions, they could not separate the role of cavitation from the effect of fuel pressure. Also fuel temperature could affect the fuel spray characteristics as it considered in several studies like Salvador et al. [8] and Payri et al. [9] work.

As shown by several studies, e.g. Hulkkonen et al. [10]; Salvador et al. [11–13], Wang et al. [14] nozzle characteristics may affect cavitation development. Experimental tests on the behavior of the flow inside the real size cases are difficult, due to their tiny dimensions (on the order of a 100-micron diameter for injector hole), high-pressure gradients, and fast transient processes, thus imposing limitations on some studies such as Winklhofer et al. [15]. Several kinds of research like Pratama et al. [16] in recent years have attempted to use scaled-up experimental geometries, but despite their valuable achievements, scaling does not seem to be entirely applicable to real size injectors. Finding a fast, economical, and reliable numerical tool for estimation of flow behavior inside the injectors, as reflected by Battistoni et al. [17], Bicer and Sou [18], and Brusiani et al.[19], is desired for both academic and industrial purposes. To this end, a Homogeneous Equilibrium Model implemented in the OpenFOAM package [20] has been selected,

* Corresponding author.
E-mail address: ehsanallah.tahmasebi@polimi.it (E. Tahmasebi).

http://dx.doi.org/10.1016/j.enconman.2017.10.049
Received 17 June 2017; Received in revised form 15 October 2017; Accepted 16 October 2017

and its validity for diesel nozzles has been examined for three different cases. In the first part, experimental tests for a scaled-up two-dimensional geometry are undertaken, and simulations outputs are compared with published experimental results by Pratama et al. [16]. In the second study, this methodology is examined in the simulation of two single hole diesel injectors, with different geometrical characteristics (Spray C and D), from Engine Combustion Network (ECN) [21]. The compatibility of our method with real size injectors in cavitating and non-cavitating regimes is projected in this analysis. The third study focuses on an actual size industrial single nozzle diesel injector. Effects of injection rail pressure on nozzle performance are investigated both numerically and experimentally. Then, the effects of geometrical parameters such as inlet edge rounding and the conicity factor on flow behavior are analyzed.

Nozzle conicity and inlet edge rounding are the two main geometrical parameters on the injector performance. Usually, the amount of conicity of the tube is defined by the conical factor:

$$k = \frac{d_{in}-d_{out}}{10} \tag{1}$$

where d_{in} and d_{out} are inlet and outlet diameters of the nozzle.

Cylindrical tubes produce intense cavitation and also increase spray breakup with a significant spray divergence near the outlet. However, axisymmetric conical geometry suppresses cavitation by gradually reducing the effective cross-sectional area of the nozzle [2]. A comparison between a cylindrical layout ($k = 0$) and conventional diesel injectors ($k = 1.1$ to 2) shows the influence of this parameter on the nozzle performance, e.g. the amounts of injection rate, fuel velocity, cavitation and turbulence at the outlet as presented by Brusiani et al. [19].

Another important parameter in the fluid behavior inside the injector is the effect of the hole entrance edge. In sharp corners, streamlines cannot follow the sudden changes in geometry direction. Subsequently, the extraction of streamlines, the separation of flow close to the wall, and the generation of strong vorticities become possible. Thus, a suitable condition is available for reducing pressure up to the saturation pressure and the cavitation regime developing. Today, in new diesel injectors, nozzle inlets are usually rounded to improve inflow conditions and remove the unwanted erosion during the injectors working life which gradually changes the spray characteristics. As an example, *ks* nozzle technology combines the conical and flow-optimized geometries, where the reduction of the cross section area depends on the distribution of the mass flow suppressing any possible cavitation [2].

In the second case study of the present work, the difference of two geometries, specifically their conicity and edge rounding, produce hugely contrasting results. Furthermore, the third study, into realizing the effect of these two parameters, presents an extensive investigation with customized geometries.

For preparing computational grid, blockMesh mesh generation utility, supplied with OpenFOAM is used. The blockMesh utility creates parametric meshes with grading and curved edges. The principle behind this utility is to decompose the domain geometry into a set of 1 or more three dimensional, hexahedral blocks [20].

2. Theoretical aspects of the method

Two main approaches are used for modeling two-phase cavitating flows: two fluid flow models treating the liquid and vapor separately on the one hand and continuum flow methods containing a homogeneous mixture of liquid and vapor on the other. In continuum flow methods, an equation of state helps to define the phase changing and cavitation growth. In this work, a homogeneous equilibrium model (HEM) which is suggested by previous works, e.g. Salvador et al. [11]; Schmidt and Corradini [6], is used to capture cavitation growth. The liquid and vapor phases are assumed to be mixed perfectly in each cell while also considering the compressibility of both phases. Likewise, pressure and density are related to each other with a barotropic equation of state as:

$$\frac{D\rho}{Dt} = \Psi\frac{Dp}{Dt} \tag{2}$$

in which Ψ is the compressibility of the mixture and is defined as the inverse squared sound speed as $\Psi = 1/a^2$.

The equation of state should be consistent with the liquid and vapor equations of state when only one phase is present and also at intermediate states when there is a mixture of them. Both phases can be defined with a linear equation of state:

$$\rho_v = \Psi_v \cdot p \tag{3}$$

$$\rho_l = \rho_l^0 + \Psi_l \cdot p \tag{4}$$

To compute the amount of vapor in the mixture, γ is defined as:

$$\gamma = \frac{\rho-\rho_{lsat}}{\rho_{vsat}-\rho_{lsat}} \tag{5}$$

where $\rho_{vsat} = \Psi_v \cdot p_{sat}$. It could be observed that in a flow without cavitation $\gamma = 0$, whereas for a fully cavitated flow $\gamma = 1$. The mixtures density is calculated with Eq. (6), taking into the account vapors amount in the fluid (γ) together with a correction term based on the pressure (the mixtures equilibrium equation of state).

$$\rho = \gamma\cdot\rho_v + (1-\gamma)\cdot\rho_l + \Psi(p-p_{sat}) = (1-\gamma)\cdot\rho_l^0 + [(\gamma\cdot\Psi_v + (1-\gamma)\cdot\Psi_l)-\Psi]\cdot p_{sat} + \Psi\cdot p \tag{6}$$

In Eq. (6), the liquid density at a given temperature condition is defined as $\rho_l^0 = \rho_{lsat} - \Psi_l \cdot p_{sat}$. As far as the mixtures compressibility is concerned, it is modeled by a simple linear formula:

$$\Psi = \gamma\cdot\Psi_v + (1-\gamma)\cdot\Psi_l \tag{7}$$

in which Ψ_l is the compressibility of the liquid. Despite this, there are models describing the compressibility of the mixture in a more physical way, as well as studies focusing on compressibility effect [22] but a linear formula is chosen in this work due to the stability and convergence advantages. As for compressibility, it is possible to obtain the viscosity of the mixture through a linear equation:

$$\mu = \gamma\cdot\mu_v + (1-\gamma)\cdot\mu_l \tag{8}$$

The methodology used by the solver starts by solving the continuity equation for ρ:

$$\frac{\partial\rho}{\partial t} + \nabla(\rho.\ u) = 0 \tag{9}$$

The value obtained for ρ is used to determine preliminary values for γ and Ψ by using Eqs. (5) and (7), and also solving the momentum equation (Eq. (10)) from which the matrix is derived which allows the velocity u to be calculated:

$$\frac{\partial(\rho\cdot u)}{\partial t} + \nabla(\rho.\ u.\ u) = -\nabla p + \nabla(\mu_f\cdot\nabla u) \tag{10}$$

Convection terms in both mass and momentum conservation equations are discretized by using the Gauss theorem with an upwind scheme. This selection provides a stable simulation in the presence of large pressure and density gradients, despite the fact that first-order schemes are known to increase numerical diffusion when the mesh resolution is reduced. Concerning the diffusion terms, the non-orthogonal part of the gradient is included due to the relatively low mesh non-orthogonality for the configurations tested in this work, as recommended by Jasak [23] and Salvador et al. [11]. An iterative PISO algorithm is used to solve for p and correct the velocity to achieve continuity. The equation solved within the PISO loop is the continuity equation transformed into a pressure equation by using the equation of state (Eq. (6)):