Fuel 222 (2018) 485–495

Contents lists available at ScienceDirect

Fuel

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

Full Length Article

A simplified model for a diesel spray

Simon Marčič[a,*], Milan Marčič[a], Michael Wensing[b], Thomas Vogel[b], Zdravko Praunseis[c]

[a] Faculty of Energy Technology, University of Maribor, Slomškov trg 15, 2000 Maribor, Slovenia
[b] Erlangen Graduate School in Advanced Optical Technologies (SAOT) and Institute of Engineering Thermodynamics (LTT), Friedrich-Alexander-Universität Erlangen-Nürnberg (FAU), Am Weichselgarten 8, 91058 Erlangen, Germany
[c] Faculty of Energy Technology, University of Maribor, Slomškov trg 15, 2000, Slovenia

ARTICLE INFO

Keywords:
Diesel fuel spray
Diesel engines
Thermodynamics

ABSTRACT

This paper describes a computer simulation and measurements of the liquid phase of Diesel spray formation.

The spray is divided into small elementary volumes in which the amounts of liquid and gaseous fuel, air, mean, maximum and minimum fuel droplet diameter are calculated, as well as their number. The total air–fuel and air–fuel vapour ratios are calculated for each elementary volume. The mathematical model of the Diesel spray is based on continuity and momentum equations in Eulerian-Lagrangian formulation.

The measurements of the sprays were carried out in the combustion chamber at the LTT, which is equipped with a high pressure 250 MPa fuel injection system. During the experiments, ambient pressure varied from 3 MPa to 7 MPa, temperature from 673 K to 973 K, injection pressure from 40 MPa to 200 MPa and nozzle temperature from 243 K to 363 K. The injection quantity was 13.5 mm^3 per cycle in all cases. The combustion chamber was filled with pure nitrogen instead of air. Thermodynamic properties of nitrogen are similar to those of air. That way, self-ignition was prevented. We measured only the liquid phase of the fuel in the spray, and to this end, Mie scattering measuring technique was applied.

The comparison between the simulated and actual liquid spray penetration depth and spray cone angle showed good matching.

1. Introduction

Combustion is still the most important contemporary source of energy. The most important process influencing consumption and emissions of diesel engines is combustion itself. In contrast to Otto engines, where the ignition of the petrol vapour-air mixture is effected by a spark, the mixture is self ignited in diesel engines. For efficient combustion, being a pre-condition for low consumption and reduced pollution of the environment, it is very important to understand the spray formation process, many details of which are still not fully understood.

The problem of spray atomization is of considerable practical importance in many areas from agricultural sprays to sprays in Diesel and GDI (gasoline direct injection) engines [1,2]. Computational Fluid Dynamics modelling of reacting diesel sprays is an important factor in recent day advancement of diesel engines. With the future regulations on NOx and soot emissions in diesel engines, it is critical to accurately model the diesel engine spray and the combustion process. The soot emissions depends largely on the spray and flame structure in the engine [3].

Understanding liquid spray atomization is an important part of analysing the combustion process of many propulsion-related applications. Modern computational approaches developed for flows with sprays are often based on Eulerian-Lagrangian formulation, since the Lagrangian model of spray dynamics can be easily coupled with Eulerian continuum equations that describe the gaseous phase. In this approach, the droplet distribution function obeys the spray equation written in [4] for liquid droplets by analogy to statistical Boltzmann formalism. Some expectation was prescribed to the value $f(b_z)db_z$ for the number of droplets in the interval between b_z and $b_z + db_z$, but the probabilities relative to this expectation had not been considered. The Lagrangian Monte Carlo procedure in [5] was proposed to solve the spray equation. The right side of the spray equation was supplied later by models of drops collision [6] and their breakup [7–10]. All of these models are viewed as sub-grid-scale models (see [8], for example), since the mesh spacing typically is appreciably bigger than the drops. The most widely accepted model among [7–11] is the wave model [8]. The breakup is modelled by forming new drops with one scale size proportional to the fastest wave instability growth on the surface of parent fluid "blob." This model shows the presence of relatively large drops near the nozzle and the production of small droplets as spray

* Corresponding author.
E-mail addresses: milan.marcic@um.si (S. Marčič), michael.wensing@fau.de (M. Wensing), thomas.vogel@fau.de (T. Vogel).

https://doi.org/10.1016/j.fuel.2018.02.152
Received 5 April 2017; Received in revised form 14 February 2018; Accepted 22 February 2018

progresses. Nevertheless, since atomization is recognized from experiments as a complex nonlinear phenomenon, representing it in the framework of stochastic processes could be an interesting approach. Analysis of the spray formation and penetration, evaporation of the mono-component and multi-component droplets are well described in the book of Sazhin [12].

There are a lot of mono-component and multi-component Diesel fuel droplet evaporation model. One of the most sophisticated is the quantum chemical model which estimates evaporation of the Diesel fuel components using the results of the quantum chemical and classical kinetic gas theory calculation, and temperature dependence of the density and evaporation enthalpy of these components [13].

The mathematical model of Diesel spray is obtained from the continuity and momentum equation. The spray is divided into small elementary volumes in which the amounts of fuel, liquid fuel and fuel vapours, air, mean, maximum and minimum fuel droplet diameters are calculated, as well as their number. The total air-fuel ratio, air-liquid fuel ratio and air-fuel vapour ratio are calculated in each elementary volume. Identical data of total concentration, liquid fuel concentration, total air-fuel ratio, air-fuel vapour ratio and air-liquid fuel ratio are joined together into curves.

One of the key elements affecting the combustion process is the fuel injection system, where the injection nozzle plays a decisive role in dispersing the fuel in the droplet-vapour-air mixture throughout the combustion chamber. Therefore, we developed two computer programs: first, a simulation of the common rail accumulator fuel-injection system [15], which itself is not presented in this paper, but which provides the starting condition for diesel spray simulation, as presented in this paper. A combination of the two computer programs enables computation of the injection [16,17] and spray formation process.

Both programs are mutually connected.

The measurements of the sprays were carried out in the combustion chamber, which is equipped with a high pressure 250 MPa fuel injection system. During the experiments pressure varied from 3 MPa to 7 MPa, temperature from 673 K to 973 K, injection pressure from 40 MPa to 200 MPa and nozzle temperature from 243 K to 363 K. The combustion chamber was filled with pure nitrogen instead of air. Thermodynamic properties of nitrogen are similar to those of air. That way, self-ignition was prevented. We measured only the liquid phase of the fuel in the spray, and to this end, Mie scattering measuring technique was applied. The usually Shell Diesel fuel was applied for experiment.

2. Mathematical model of diesel spray

The mathematical model of the Diesel spray is based on continuity and momentum equations in Eulerian-Lagrangian formulation [18–20]. In the paper presented, the spray model enables calculations of all spray parameters that can be measured by the detailed laser diagnostic technique. In this way, we can compare the measured spray parameters with the calculated ones. The proposed two-dimensional (radial and axial) spray model is simple and can be solved very quickly using a PC. A three-dimensional CFD method requires expensive CFD programs and a much faster computer, which is much more expensive than the proposed spray computer program. The spray calculation covers the interval from the fuel outlet of the nozzle to the mixture formation, ideal for self-ignition in the combustion chamber of a Diesel engine. The fuel leaves the nozzle sac volume at a high velocity because of the pressure difference between sac volume and cylinder pressures. The fuel spray model is derived in two limiting cases: the initial stage and the two-phase flow regime. At the initial stage, the effects of droplet drag and entrainment of the air are accounted for. In the case of the two-phase flow, it is assumed that the spray droplets have the same velocities as the entrained air [21].

The exit velocity of the fuel u(t) is calculated on the basis of injection characteristics Q(t), which can be measured or calculated by

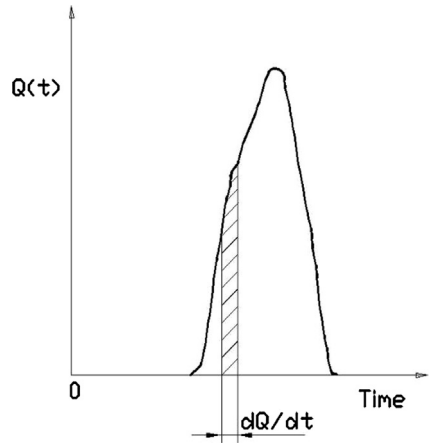

Fig. 1. Injection rate.

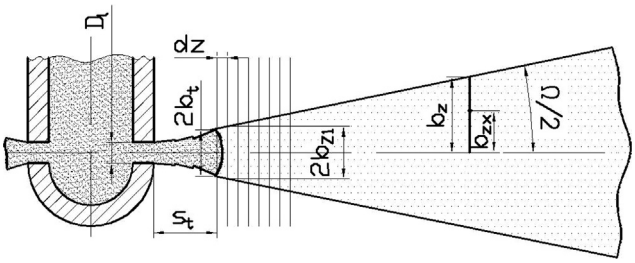

Fig. 2. Spray of the injected fuel.

computer simulation [15–17] of the fuel injection system (Figs. 1 and 2). The computer simulation enables losses calculation in all section of fuel injection system. Injection characteristic is measured at the outlet area of the nozzle hole, therefore all losses are taken into account. The injection characteristic Q(t) is input data for spray simulation. The equation for exit velocity is

$$u(t) = Q(t)/A, \quad (1)$$

where A is the nozzle hole area with diameter D_l.

The basic assumption of the model is that the transient liquid fuel spray can be analysed as a turbulent gas jet. This approach was first taken by Abramovich [20] for a fuel spray. Abramovich reasoned that the fuel droplets in a spray could be treated as a very dense admixture in air. Although the fuel spray is initially pure liquid, it both entrains air and vaporizes as it moves downstream of the nozzle. The liquid fuel droplets soon become a small fraction of the jet volume. The geometry of the spray model is shown in Fig. 2. The fuel leaves an orifice of diameter D_l with velocity $u(t) = u_0$ and cone angle Ω. At the nozzle exit, the fuel jet hits the air and starts pushing it. At the beginning, the jet is in the liquid phase; however, after a distance s_t (Fig. 2), it is affected by external forces (aerodynamic force being the most vital) and breaks up into small droplets. The latter then break up further into smaller droplets until a balance is achieved between the internal forces within the droplets and the external forces [11,21–28].

Three conservation equations comprise the jet model. These include conservation of the fuel mass, conservation of the total mass, and conservation of momentum. The equations that are differential in form are written for a cylindrical element of thickness dz normal to the jet axis, as shown in Fig. 2. The equations are written for the spray axis direction.

The following equations are applied in the mathematical model of the spray:

1. Continuity equation for fuel