IMPACT: International Journal of Research in Engineering & Technology (IMPACT: IJRET)
ISSN(P): 2347-4599; ISSN(E): 2321-8843
Vol. 5, Issue 4, Apr 2017, 11-22

DESIGN AND DEVELOPMENT OF HIGH PRESSURE SPRAY CHAMBER IN IC ENGINE USING FLOW VISUALIZATION TECHNIQUES

ARUL MANIJ[1], NAGARAJAN P[2], SWAMINATHAN G[3], VIGNESH K[4] & SARAVANA BHAVAN P[5]

[1]Assistant Professor, Department of Mechanical Engineering,
Vel Tech (owned by RS trust), Chennai, Tamil Nadu, India

[2,3,4,5]Research Scholar, Department of Mechanical Engineering,
Vel Tech (owned by RS trust), Chennai, Tamil Nadu, India

ABSTRACT

Diesel engines are one of the most important prime movers for mobility and reorganized power generation and are categorized by heterogeneous combustion. Heterogeneous combustion leads to formation of soot and NOx in the combustion chamber. The gasoline injection system, which is the heart of an engine, plays a pivotal role in the fuel–air mixing processes inside the combustion chamber. Fuel atomization activities are mainly contingent on constraints such as nozzle diameter, injection pressure, ambient conditions, and nozzle geometry. Fuel atomization also depends on fuel properties such as viscosity, density, surface tension, and vapor pressure. The main objective of our research work is to design and development of fuel injector holder in high pressure spray chamber, which is used to develop spray formation and understand the behavior of various fuel spray characteristics like diesel and bio-diesel. An important parameters are used to find in the characteristics are fuel spray, fuel spray length, fuel regions, droplet size. In our research also concentrate to arrest the leak formation in high pressure spray chamber.

KEYWORDS: High Pressure Spray Chamber, Fuel Injector Holder, Flow Visualization Technique, Combustion Process, Emission Formation, Spray Modelling

INTRODUCTION

The fuel injector is a significant component of fuel injection equipment (FIE) and delivers gasoline in atomized form to the combustion chamber at very high injection pressures. The main role of the injector is to inject and successively atomize the fuel in the engine combustion chamber. Loftier atomization leads to formation of adequate fuel droplets (lm range) inside the engine combustion chamber. The large number of fuel droplets in the combustion chamber results in higher fuel surface area obtainability for air–fuel interactions, leading to superior air–fuel mixing and combustion. With increasing concerns about air pollution, especially the particulate matter (PM) emissions of diesel engine, more stringent regulations to limit engine emissions were legislated. In recent years, as a result, many methods have been proposed to simultaneously reduce PM and nitrogen oxides (NOx) emissions in diesel engine. Using alternative fuels and gaseous fuels in existing diesel engine has been proved to be one of the most practical ways to reduce the harmful emissions [1].

ERC n-heptane mechanism with 29 species and 52 reactions combined with NOx mechanism (4 species and 12 reactions). They used a sector mesh with one nozzle and dual injection for diesel liquid which showed improvement in fuel efficiency and reduction in particulate matter emission compared to diesel only operation. The authors also found that the rate of pressure rise is too sensitive to injection timing [2]. The comprehensive two-zone phenomenological model has been examined the effects of pilot fuel quantity and pilot injection timing on the dual-fuel engine performance and emissions, and then validated the model with experiment in a premixed dual-fuel engine. They reported that simultaneously increase of the pilot fuel quantity accompanied with an increase of its injection timing results in an improvement of the engine efficiency (increase) and CO emissions (decrease) while it has a negative effect (increase) of NOx emissions [3]. The self-ignition of diesel fuel which is governed by the Stringer relationships for the ignition delay and the diesel combustion model employed for a full engine cycle calculation consists of the classical one step kinetic mechanism of fuel oxidation within a finite rate eddy dissipation approach [4].

The computational fluid dynamics simulation of both DI dual-fuel compression ignition (CI) and SI CNG engines that a combined approach of using both chemical kinetics and G-equation formulations in RANS is capable of capturing most of the physical-chemical processes in both the dual-fuel CI and SI engine combustions [5]. The combustion regime is divided into 3 zones, flame front, volume inside the flame-front and volume outside the flame-front. Flame is always modelled at chemical equilibrium. Species like NO, N, N2O and NO2 are not included in the in the flame modelling because of their relatively short resident time within the flame front and relative slow chemical reaction rate of NOx producing reactions [6] [7] [8].

OEMs design the injector for mineral diesel, which has a moderately lower viscosity and density compared to biodiesel. Different fuel properties of biodiesel. There is a need to more completely understand these effects [9] [10]. The injection rates for two nozzle hole geometries (cylindrical and conical). They describe the behavior of injector nozzles by measuring non-dimensional parameters such as Reynolds number (Re), cavitation number (K), and discharge coefficient (Cd). They detected that cavitation occurs in both nozzle types; however, the conical nozzle had a lower inclination to cavitate compared to the cylindrical nozzle. They also experimented that higher fuel injection pressure and Re leads to confused flow patterns in the cylindrical nozzle, while this is not the case in the conical nozzle[11] [12]. A computational study to determine the effects of cavitation on the nozzle hole internal flows and spray characteristics. They concluded that cavitation increases the spray cone angle and nozzle exit velocity. In a cylindrical nozzle, cavitation starts at the inner sharp corner of the nozzle [13].

As soon as the injection pressure builds up, cavitation extends towards the nozzle exit, which consistently results in increased spray angle [14]. Once cavitation reaches the nozzle exit, this singularity is called super-cavitation. They characterized two injector nozzles; cylindrical and conical, using two sprays parameters, such mass flux and momentum flux. They reported that cavitation occurs when mass flow flops in the cylindrical nozzle. They also described that there was no cavitation in the conical nozzle, even under risky pressure condition. The atomization characteristics of diesel, which is affected by in-nozzle cavitation. A flow visualization system was used for spotting the cavitation, while atomization characteristics were estimated using a droplet analysis system [15] [16]. In order to understand the essentials of spray in IC engines, methods such as visualization, PIV (particle image velocimetry), LIF (laser induced fluorescence) have been used as diagnostics tools for in-cylinder measurements some measurements have been carried out in improved engines with optical access and some have been accomplished in high pressure and temperature constant volume vessels.

In the present study, a broad survey has been carried out in the area of spray and combustion using optical measuring techniques. The main objective of our research work is to design and development of fuel injector holder in high pressure spray chamber, which is used to develop spray formation and understand the behavior of various fuel spray characteristics like diesel and bio-diesel. An important parameter is used to find in the characteristics are fuel spray, fuel spray length, fuel regions, droplet size. Furthermore, our research concentrated to arrest the leak formation in high pressure spray chamber.

MAJOR COMPONENTS OF THE HIGH PRESSURE SPRAY CHAMBER

The major components of the high pressure spray chamber, Constant volume combustion chamber, High speed camera and Common rail.

Constant Volume Combustion Chamber

The design of Constant volume combustion chamber is shown in Figure 1.

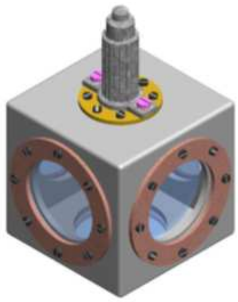

Figure 1: Design of Constant Volume Combustion Chamber

The CVCC system with the chamber is utilizing to investigate the spray combustion. The CVCC is manufacturing with S45C to endure high pressure and temperature. The maximum pressure and temperature are 15 MPa and 2000 K, respectively. The chamber volume is 1.4 L. The CVCC has five 96 mm diameter quartz windows. The high pressure and high temperature ambient condition of a diesel engine has been simulated by performing pre-ignition before the fuel injection. Four gases (acetylene: C_2H_2, hydrogen: H_2, Oxygen: O_2, and nitrogen: N_2) were mixed in a pre-mixing chamber, and then supplied to the CVCC through the intake valve. Before the mixture was supplied to the chamber, the chamber was evacuated by a vacuum pump. The composition of the mixture is modulated by the partial pressure of each gas. The initial chamber pressure before the spark ignition is around 3 MPa. The partial pressures of each gas could be controlled to $\pm 10^4$ MPa. The chamber body is heated up to 473 K using an electric heater to ensure the complete combustion of the pre-mixture. Two spark plugs are utilized to fully ignite the pre-mixture. The ratios of the combustion products, such as oxygen and carbon dioxide, are controlled by adjusting the composition of the reactant.

High Speed Camera

The high Speed Camera is shown in Figure 2.

Figure 2: High Speed Camera

This high-speed camera is a device capable of image exposures in excess of 1/1,000 or frame rates in excess of 250 frames per second. It is used for recording fast-moving objects as a photographic image(s) onto a storage medium. After recording, the images stored on the medium can be played back in slow-motion. Early high-speed cameras used film to record the high-speed events, but today high-speed cameras are entirely electronic using either a charge-coupled device(CCD) or a CMOS active pixel sensor, recording typically over 1,000 frames per second into DRAM and playing images back slowly to study the motion for scientific study of transient phenomena. A high-speed camera can be classified as: a high-speed film camera which records to film, a high-speed video camera which records to electronic memory, a high-speed framing camera which records images on multiple image planes or multiple locations on the same image plane (generally film or a network of CCD cameras) and a high-speed streak camera which records a series of line-sized images to film or electronic memory.

A normal motion picture film is played back at 24 frames per second, while television uses 25frames/s (PAL) or 29.97 frames/s (NTSC). High-speed film cameras can film up to a quarter of a million frames per second by running the film over a rotating prism or mirror instead of using a shutter, thus reducing the need for stopping and starting the film behind a shutter which would tear the film stock at such speeds. Using this technique one can stretch one second to more than ten minutes of playback time (super slow motion). High-speed video cameras are widely used for scientific research, military test and evaluation, and industry. Examples of industrial applications are filming a manufacturing line to better tune the machine, or in the car industry the crash testing to better document the crash and what happens to the automobile and passengers during a crash. Today, the digital high-speed camera has replaced the film camera used for Vehicle Impact Testing.

Uses in Science

High-speed cameras are frequently used in science in order to characterize events which happen too fast for traditional film speeds. Biomechanics employs such cameras to capture high-speed animal movements, such as jumping in frogs and insects, suction feeding in fish, the strikes of mantis shrimp, or the aerodynamic study of pigeon's helicopter like movements using motion analysis of the resulting sequences from one or more cameras to characterize the motion in either 2-D or 3-D.

The move from film to digital technology has greatly reduced the difficulty in use of these technologies with unpredictable behaviours, specifically via the use of continuous recording and post-triggering. Most software allows saving a subset of recorded frames, minimizing file size issues by eliminating useless frames before or after the sequence of interest. Such triggering can also be used to synchronize recording across multiple cameras.

Common Rail

The common rail direct diesel injector is shown in Figure 3.

Figure 3: Common Rail Direct Diesel Injector

Common rail direct fuel injection is a direct fuel injection system for petrol and diesel engines. On diesel engines, it features a high-pressure (over 1,000 bars or100 MPa or 15,000 psi) fuel rail feeding individual solenoid valves, as opposed to a low-pressure fuel pump feeding unit injectors (or pump nozzles). Third-generation common rail diesels now feature piezoelectric injectors for increased precision, with fuel pressures up to 3,000 bars (300 MPa; 44,000 psi). The merits of the common rail fuel injection system architecture have been recognized since the development of the diesel engine.

DESCRIPTIONS

It is essential to characterize the fuel before spray investigations for its important physical, chemical and thermal properties. Kinematic viscosity of mineral diesel and biodiesels has been measured using Kinematic viscometer (Stanhope-Seta: Setavis 83541-3) at 40. ASTM 445 test method has followed for viscosity measurement. Fuel density has measured using portable Density meter (Kyoto Electronics: DA-130N) following ASTM D4052 test method. ASTM D613 test method has used for cetane number determination. Cetane number tests are carried out at Indian Oil Corporation (R&D centre), Faridabad. Flash point test fuels are determined by ASTM 3828 test method using Automatic Flash Point Apparatus (Stanhope-Seta: 33000-0). Carbon residue of test fuels has determined using Carbon Residue Tester. Carbon residue indicates the tendency of the fuel to form carbonaceous deposits and ASTM D189 test method has used for its determination. Bomb calorimeter (Parr, USA; 6200) has used for determination of the calorific value of the test fuels. The constant volume spray chamber consists of four diametrically opposite glass windows, which enable optical access for spray illumination and visualization.

SPRAY MODELLING

The organization of fuel-air mixture is of great importance in the subsequent combustion process and emission formation. With the increasing concern of the environmental aspects, novel fuels and updated injection systems require deep understanding of spray characteristics. The accuracy of spray modelling directly influences the performance of spray strategies and injection system improvements. Thus, the significance of spray modelling is increasingly highlighted and an intensive development has been achieved in the last few decades.

However, sprays are characterized by a broad range of size and time scales; the interaction between sprays and the surrounding gas is a complex two-phase flow. These make spray numerical simulation a challenging task. An applicable method is to introduce sub-models to describe the sub-grid scale physical phenomenon related to the spray process. In the IC engine simulation, spray sub-models generally include atomization, drop drag and deformation, drop breakup and evaporation, drop collision/coalescence, and spray/wall interaction. Developments in these sub-models will be discussed in the following content. The spray regions adopted is shown in figure 4.

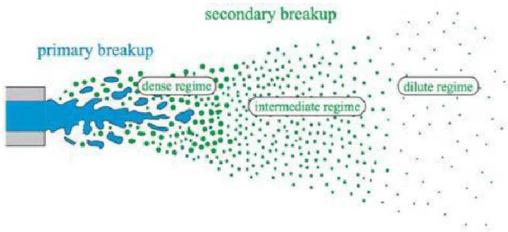

Figure 4: Spray Regimes Adopted

Firstly, it is necessary to describe the spray regimes. As presented in a dense region exists near the nozzle exit, where spray atomization, drop collisions and coalescence are the dominant forms of the two-phase dynamics. Far from the nozzle, a dilute region can be seen where spray drops become wide-spaced and their masses and volumes can be neglected compared to those of the surrounding gas. Between the two regions is an intermediate region, where the drop/gas interactions such as drop wake disturbances and deformation become significant.

To describe the two-phase flow in the atomization process, the Volume of Fluid (VOF) method, Eulerian multi-fluid treatment and Discrete Droplet Model (DDM) Lagrangian method are currently the three major approaches. Using the VOF method which is an interface tracking approach, an accurate evolution of the gas-liquid interfaces and the shapes and sizes of all the particles can be obtained, but very grids are required to track every interface, which limits its application in IC engines. In the Eulerian multi-fluid approach, the liquid-gas phases are treated as continuous phases and are solved using the Eulerian description of an ensemble averaging method. However, to fully capture the droplet size distribution, an enormous number of grids are necessary to denote each size group and thus this method becomes computationally expensive. The Lagrangian method is a particle tracking approach, in which the gas phase is solved using an Eulerian scheme and the parcels of droplets are tracked in a Lagrangian framework. By using the DDM, the coherent liquid core can be efficiently discretized into groups of equally sized droplets and the numerical diffusion in the liquid phase solution is greatly enhanced. Due to its simplicity and steadiness, it is widely used in current spray simulations.

Primary Breakup Models

To employ the DDM Largrangian model, the initial conditions such as the drop size and velocity distributions at the nozzle exit are essential to the accuracy of spray simulation. These conditions are provided by the primary breakup model. For high-speed atomizing sprays, the liquid column is disintegrated into drops and ligaments with the combination of inertia force, surface tension and aerodynamics shear. Gosman summarized that the five major theories about the mechanism of the jet atomization included the disturbance of aerodynamic shear stresses (Kelvin-Helmholtz instabilities). Thus, spray atomization models were established by the single theory or hybrid theories as mentioned above. The MPI primary breakup model is shown in Figure 5.

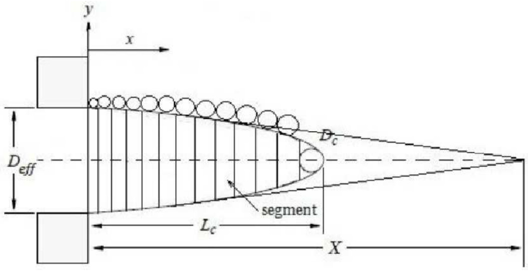

Figure 5: MPI Primary Breakup Model

Among the atomization models based on the hypothesis of the disturbance of the aerodynamic forces, the Kelvin-Helmholtz (KH) model and the Taylor Analogy Breakup (TAB) model are widely used in research and industry. The former model, proposed by Reitz, was derived from the Kelvin-Helmholtz instabilities. This model assumed that the jet disintegration was caused by the surface disturbance growth. While in the TAB model proposed by O'Rourke et al., it was assumed that jet atomization was considered as an analogy to the spring-mass system, which was determined by the comprehensive effects of aerodynamic forces, liquid surface tension and viscosity. The Blob-injection method of jet atomization is shown in Figure 6.

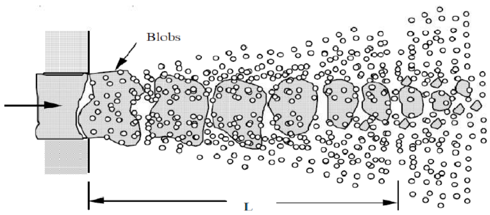

Figure 6: Blob-Injection Method of Jet Atomization

A concept of "blobs" injection was the basis of these models, which was introduced by Reitz and Diwakar. For the blob-injection method, the intact liquid was represented by a train of spherical blobs with the same size of the nozzle hole diameter at the nozzle exit and the liquid was injected as large discrete parcels within the intact core region near the nozzle exit as presented in Additionally, it was assumed to be indistinguishable for the liquid jet atomization and the subsequent drop breakup process. However, without consideration of effects of nozzle flow on the drop size distribution, the initial diameter of these blobs was under adjustment by reference to experimental data.

Secondary Breakup Models

The Schematic of different styles of secondary breakup is shown in Figure 7.

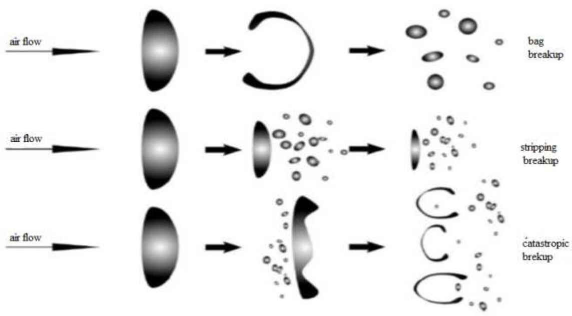

Figure 7: Schematic of Different Styles of Secondary Breakup

The drops detached from the continuous liquid jet undergo the secondary breakup process and are further disintegrated into even smaller drops. Ohnesorge number and Weber number (We) are widely used for the criteria of the secondary breakup mechanism. Ohnesorge number is a dimensionless number that relates the viscous forces to inertial and surface tension forces. For the general liquid, the Ohnesorge number is below one.

Under this circumstance, according to the Weber number, three main mechanisms were proposed by Liu [108]: bag breakup (6≤We<80), stripping breakup (80≤We<350) and catastrophic breakup (We>350). Based on the TAB model, an Enhanced TAB (ETAB) model was addressed by Tanner. In the ETAB model, according to the specific breakup mechanism (bag or stripping breakup), the droplets experienced a cascade breakup by an exponential law until the product droplets reached a stable condition. Thus, the spectrum of droplet size was extended compared to the TAB model.

For a better prediction in the average velocity of the product droplets, the radial component of the product droplet was determined by the energy conservation. Further, the Cascade Atomization and drop Breakup (CAB) model was proposed by Tanner for extending the applicable range to the catastrophic breakup mechanism.

In addition, with the introduction of continuity between different regimes, the number of model constants reduced. The Dynamic Drop Breakup (DDB) model was proposed by Ibrahim ET. Al. and considered as a nonlinear formation of the TAB model. In the DDB model, the droplet breakup resulted from its excessive elliptic deformation. In contrast with the TAB model, the deformation was definedas the motion of the mass centre of the half-drop instead of the droplet equator. However, the strong grid dependency of this model limited its application.

The Rayleigh-Taylor (RT) model was addressed by Su et al. on the basis of the RT instability theory. The disintegration of the droplet was a catastrophic breakup due to the deceleration of the droplets caused by the aerodynamic force. The KH-RT hybrid model was a combination of the KH and RT models, where the KH model alone was used for the primary breakup simulation and the both models were used for the secondary breakup simulation. This model was widely used for diesel sprays.

EXPERIMENTATION

The schematic diagram of Spray visualization experiment is shown in Figure 8.

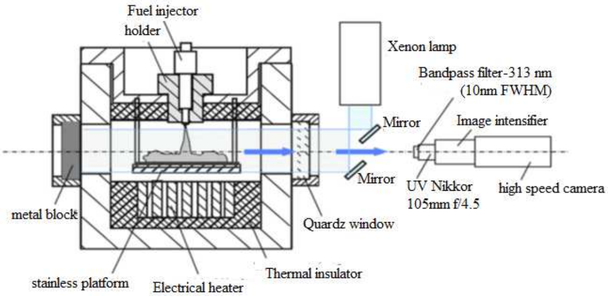

Figure 8: Schematic Diagram of Spray Visualization Experiment

The Injector holder and head of the combustion chamber are shown in Figure 9 (a) and (b) respectively.

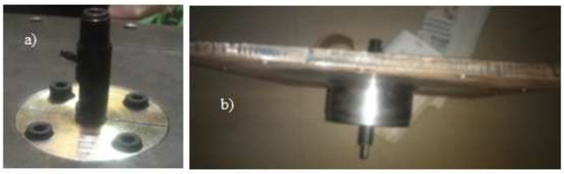

Figure 9: (a) Injector Holder (b) Head Combustion Chamber

A six-hole solenoid fuel injector (with 169 lm nozzle orifice diameter) has installed on the side flange of the chamber, in order to facilitate fuel injection into the chamber. The fuel injection equipment consists of a high pressure pneumatic pump (Maximator; G300 LVE), a common rail (DELPHI-TVS; Rail – 2.2 TML), high pressure fuel lines and an injector driver system (NI; CompactRIO-9022). Nozzle opening pressure of the solenoid injector has maintained at 1000 bar using high pressure pneumatic pump. The test fuels are supplied to the solenoid injector using a high pressure common-rail system. Two white light sources (NaBaGreen; RDL 24W) are used to illuminate the fuel droplets in the spray chamber. A high speed camera (Photron; Fastcam SA1.1) has used to capture the light scattered by the fuel spray droplets during injection. The Constant Volume Combustion Chamber (CVCC) assembly and top view are shown in Figure 10 a) and b) respectively.

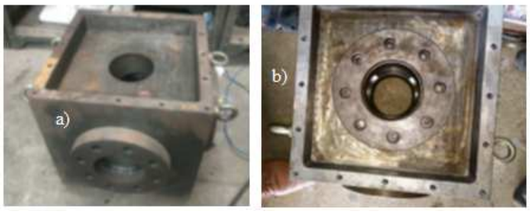

Figure 10: Constant Volume Combustion Chamber (CVCC) a) Assembly b) Top View

RESULT & DISCUSSIONS

Triggering pulse of fuel injector has synchronized with the camera to eliminate the time lag between fuel injection and images capture. During the experiments, temperature and pressure in the constant volume spray chamber are maintained at 25 C and 10 bar respectively. The images are captured using high speed camera, and then analysed using Matlab to determine the spray penetration length, spray cone angle and spray area. The injection duration pulse is kept constant at 3 ms for all experiments. One fixed frame (at 0.9 ms) is considered for the macroscopic analysis. The high speed camera has set at frame speed of 10,000 fps, 1/10,000 shutter speed (fp1) with 768×768 resolution.

However, experiments are performed using a mechanical injector with 220 bar fuel injection pressure. This study included comparison of spray results obtained from these experiments with the injector nozzle hole simulation results for mineral diesel, Karanja biodiesel and blends, and Jatropha biodiesel and blends. Simulation results are obtained for fluid velocity and vapour fraction profiles inside the injector nozzle hole. Experimental work is carried out at an ambient pressure of 10 bars. This is done for the sake of comparison with simulation study, which is also done at 10 bar ambient pressure.

The images captured are used to evaluate macroscopic spray parameters. Spray visualization images provide qualitative information about the spray evolution pattern for this test fuels. There are various methods that can be used to capture the spray and combustion images within the combustion chamber.

Different methods provide different information. Using visualization methods, macroscopic parameters could be determined. However, for combustion characteristics studies, visualization technique is very useful in gathering the information from the combustion chamber.

CONCLUSIONS

In order to study macroscopic spray characteristics and auto-ignition properties of different type of nozzles, a constant volume combustion chamber has been manufactured and diesel engine conditions are obtained in the experimental setup. A CCD camera has been used to obtain spray images and a digital imaging program has been used to obtain corrected images. Digitalization allows objectively determining spray characteristics. Experimental results are compared with well-known spray correlationsin the literature.

REFERENCES

1. Wagemakers, A. and Leermakers, C., "Review on the Effects of Dual-Fuel Operation, Using Diesel and Gaseous Fuels, on Emissions and Performance," SAE Technical Paper 2012-01-0869, 2012, doi:10.4271/2012-01-0869
2. Zoldak, P., Sobiesiak, A., Bergin, M., and Wickman, D., "Computational Study of Reactivity Controlled Compression Ignition (RCCI) Combustion in a Heavy-Duty Diesel Engine Using Natural Gas," SAE Technical Paper 2014-01-1321, 2014, doi:10.4271/2014-01-1321
3. Papagiannakis, R.G., "Study of air inlet preheating and EGR impacts for improving the operation of compression ignition engine running under dual fuel mode," Energy Convers. Manag.68:40-53, 2013, doi:10.1016/j.enconman.2012.12.019
4. Abagnale, C., Cameretti, M.C., Simio, L. De, Gambino, M., Iannaccone, S., and Tuccillo, R., "Numerical simulation and experimental test of dual fuel operated diesel engines," Appl. Therm. Eng. 65(1-2):403-417, 2014, doi:10.1016/j.applthermaleng.2014.01.040
5. Talekar, A., Lai, M., Zeng, K., Yang, B. et al., "Simulation of Dual-Fuel-CI and Single-Fuel-SI Engine Combustion Fueled with CNG," SAE Technical Paper 2016-01-0789, 2016, doi: 10.4271/2016-01-0789.
6. Liang, L., Reitz, R.D., Yi, J., and Iyer, C.O., "A G -equation Combustion Model Incorporating Detailed Chemical Kinetics for PFI / DI SI Engine Simulations G -equation description of turbulent," 2006.
7. Azimov, U., Tomita, E., and Kawahara, N., "Ignition, Combustion and Exhaust Emission Characteristics of Micro-pilot Ignited Dual-fuel Engine Operated under PREMIER Combustion Mode," 2011.
8. Aksu, C., Kawahara, N., Tsuboi, K., and Nanba, S., "Effect of Hydrogen Concentration on Engine Performance, Exhaust Emissions and Operation Range of PREMIER Combustion in a Dual Fuel Gas Engine Using Methane-Hydrogen Mixtures," JSAE 20159, 2015.
9. Agarwal AK, Chaudhury VH. Spray characteristics of biodiesel/blends in a high pressure constant volume spray chamber. Exp Thermal Fluid Sci 2012; 42:212–8.
10. Jung H, Kittelson DB, Zachariah MR. Characteristics of SME biodiesel-fueled diesel particle emissions and the kinetics of oxidation. Environ Sci Technol 2006; 40:4949–55.

11. Carlucci AP, Risi DA, Laforgia D, Naccarato F. Experimental investigation and combustion analysis of a direct injection dual-fuel diesel-natural gas engine. Energy 2008; 33:256e63.

12. Xingcai L, Xiaoxin Z, Libin J, Zheng Y, Dong H, Chen H, et al. Experimentalstudies on the dual-fuel sequential combustion and emission simulation. Energy 2013; 51:358e73.

13. Bergwerk W. Flow pattern in diesel nozzle spray holes. Proc Inst Mech Eng 1959; 173:655–60.

14. Sou A, Masaki Y, Nakajima T. Numerical analysis of cavitating flow in a spray nozzle. In: Proceedings of 6th annual conference on liquid atomization and spray systems 2001. ILASS-Asia; 2001. p. 283–8.

15. M. Gavaises, A. Andriotis, Cavitation inside multi-hole injectors for large diesel engines and its effect on the near-nozzle spray structure, SAE Paper 2006-1-11142006.

16. A. Osman, Failure of a diesel engine injector nozzle by cavitation damage, Eng. Fail Anal. 13 (2006) 1126–1133