Synopsis

Of the thesis entitled

"Numerical modelling of cavitation and its effect on spray characteristics in fuel injector for bio-diesel"

to be submitted

As a partial fulfilment for the award of the degree of

DOCTOR OF PHILOSOPHY

In

MECHANICAL ENGINEERING

By

Mehul Pravinbhai Bambhania
(FOTE-1066)

Under the supervison of

Dr. Nikul K. Patel
Assistant Professor

सत्यं शिवं सुन्दरम्

Department of Mechanical Engineering
Faculty of Technology & Engineering
The Maharaja Sayajirao University of Baroda
Vadodara- 390001

March 2024

INDEX

List of Figures

List of Table

1. Introduction

The rapid growth in energy demands increases carbon emissions and leads to the risks of the global warming effect. According to the projection, about 40% of energy sources will come from liquid fuels by 2040[1]. A major portion of liquid fuel is used in the transportation sector with the Internal Combustion (IC) engine. Efficient fuel combustion is the prime requirement of both diesel & gasoline engines. Although, both engine technologies are evolving to satisfy two major requirements: Fuel efficiency, & Emission reduction. The emission of the diesel engine can be reduced by efficient fuel atomization i.e. producing smaller and more dispersed fuel droplets. Producing smaller and more dispersed droplets leads to complete combustion of the fuel-injected, resulting in less emission production. Thus the flow of fuel inside the fuel injector nozzle is to be known to have a considerable effect on the spray [2].

2. Need for the study

Modern injectors for passenger cars and heavy-duty engines are working at 2000 bar injection pressure and inject the liquid fuel through the orifice of diameter in the order of 100 to 300 μm[3]. The liquid fuels enter the combustion chamber in the form of a fuel jet with velocities of around 500 m/s. The liquid fuel jets experience aerodynamic resistance in the combustion chamber, which leads to breakup of liquid jet into liquid ligaments, immediately after leaving the nozzle orifice. This phenomenon is called primary break-up. In addition, liquid ligaments break into droplets and further break up forming a dense spray. During high injection pressure, fuel flowing inside the fuel injector nozzle hole observed turbulence and cavitation phenomenon leading to a primary break-up. Cavitation is one of the important phenomena happening inside the fuel injector nozzle leading to primary break-up as shown in Fig.1.

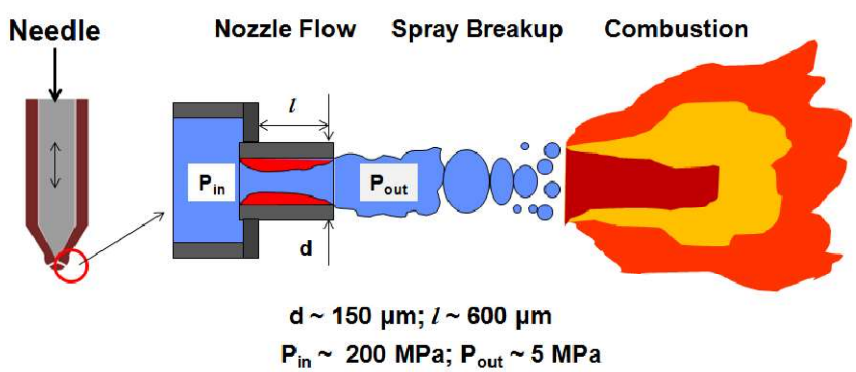
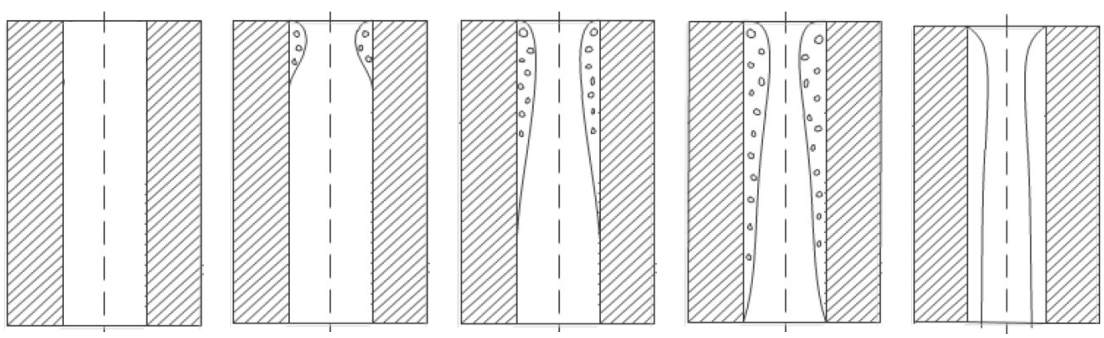


Fig. 1: Illustration of a cavitation nozzle and cavitation affecting spray breakup & combustion[4]

The primary breakup is an important phenomenon for the atomization and combustion of fuel inside the combustion chamber of an IC engine. The extent of work carried out by various researchers shows the potential of the cavitation phenomenon leading to primary break-up.

2.1 Cavitation in the Fuel injector nozzle

The rise in the injection pressure increases the intensity of the cavitation. Based on the injection pressure the inner nozzle flow can be classified into different flow regimes: (a) no cavitation (b) cavitation inception (c) cavitation growth (d) super-cavitation and (e) cavitation flip. Fig.2 shows the graphical representation of the different stages of cavitation flow inside the fuel injector nozzle. A low pressure area forms close to the fuel injector nozzle entrance when high pressure fuel accelerates due to a rapid constriction in the surrounding area. Cavitation bubbles begin to form when the local pressure drops below the vapour pressure; this phenomenon is known as cavitation inception (Fig 2(b)). Increases in injection pressure also result in a rise in the low pressure region, which promotes the formation of more cavitation bubbles and refers to cavitation development (Fig 2 (c)). Super cavitation is the term for the specific stage that occurs when cavitation bubbles form and reach at the nozzle outlet and collapsing there (Fig 2(d)). The further rise in injection pressure leads to gas entrainment into the nozzle creating a thin layer of gas attached to the wall and cavitation disappears immediately. This is known as hydraulic flip (Fig. 2(d)).



(a) No Cavitation | (b) Cavitation inception | (b) cavitation development | (c) Super cavitation | (d) Hydraulic flip

Fig. 2 Different stages of cavitation flow inside the fuel injector nozzle [5]

The cavitation can be used to identify the inner nozzle flow's characteristics. It is widely accepted that there is a close relationship between spray formation and primary breakup. The combination of three mechanisms controls the primary breakup of the liquid jet (a) Aerodynamic forces experienced by the liquid jet (b) Turbulence within the liquid phase (c) Cavitation bubbles shown in Fig.3. The spray characteristics can be understood by the macroscopic parameter like the Spray cone angle and Spray tip penetration as well as the microscopic parameter like droplet size distribution and velocity distribution.

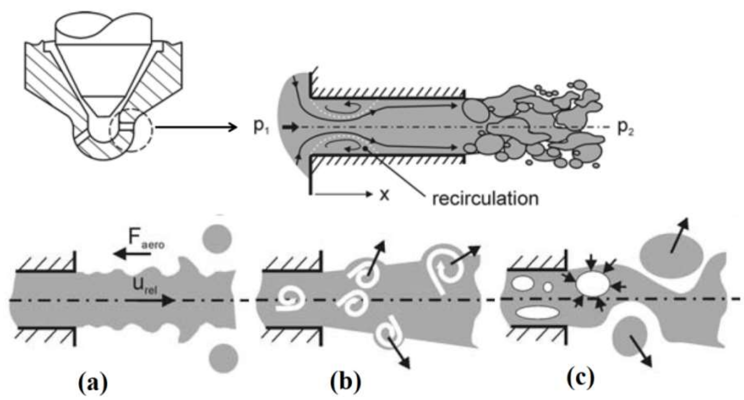


Fig.3 Governing mechanism for primary spray break-up[6]

3. Review of Literature

Several experimental literature works had been published to determine the behaviour of the cavitation flow in the fuel injector nozzle. This experiment helps to understand the structure of cavitation and the two-phase flow pattern. An early pioneer experimental work has been carried out by Begewerk[7] in which he discussed the correlation between fuel injection and cavitation. With the help of the flow visualization technique, the flow through a large size transparent nozzle was observed. The presence of cavitation and hydraulic flip mainly depends upon the Cavitation number (CN) and has little dependence on the Reynolds number (Re). When the liquid flow entirely separates from the nozzle wall and downstream gas enters the nozzle is known as the hydraulic flip condition. Nurik[8]experimented with a scaled-up transparent nozzle (d=8mm) and observed that cavitation and hydraulic flip depend on CN, nozzle radius, length-to-diameter ratio and pressure. Based on his research, he has created an empirical association between the nozzle's cavitation characteristics and discharge coefficient.

Soteriou et al. [9] experimented with the large-scale transparent injector nozzle to explore the different flow regimes and the mechanism of its formation inside the nozzle. He emphasizes the effect of hydraulic flips on spray atomization. The flipped nozzle does not experience any wall shear, which leads to poor spray atomization. This condition reduces the turbulence and smooth unbroken liquid jets that come out from the nozzle outlet. Although hydraulic flip had never been observed in real scale nozzles with realistic operating conditions. Chaves et al. [10] extended Soteriou et al.[9] work with a small-scale nozzle (d=0.2mm) and injection pressure up to 1000 bar. In his work, he reported super cavitation is different from the hydraulic flip. Super cavitation is referred to when cavitation bubbles reach the nozzle's outlet. In this condition, bubbles collapse at the nozzle exit which is a favourable condition for primary spray breakup.

The first real-size nozzle investigation was made by Arcoumanis et al. [11], He found that in real-scale experiments there were clear voids were observed and in scaled-up experiments, cloudy bubbles appeared. He also found that the cavities are initially clear and become more opaque towards the exit of the real scale nozzle. The results indicate that the nature of cavitation changes from a large void to a bubbly mixture in scaled-up experiments. However Winklhofer et al.[12] reported experimental study with real size nozzle & developed optical method to capture cavitation inside fuel injector nozzle. An extensive study for measurements of velocity, as well as, mass flow were reported. Payri et al. [13] used a single-hole transparent cylindrical nozzle made of fused silica (SiO_2) with an outer diameter of 0.51mm and 1mm in length. Four different fuels n-dodecane, n-heptane, n-decane & commercial diesel has been used for visualization and parametric study of cavitation flow. The inception of cavitation appears early with low viscous fuel. He also study the effect of geometry shape on the cavitation flow.

Due to experimental limitation, the use of computation fluid dynamics (CFD) increases recently, which allows to capture inside detail of cavitation phenomena. Various cavitation model has been developed, which can be categorized as: (1) single fluid or continuum model (2) two fluid model. In single fluid or continuum model, the average mixture properties, such as density and viscosity, are determined based on the vapor volume fraction. Schmidt et al. [14] developed model based on single fluid approach. In two-fluid models, the liquid and vapor phases are treated separately using two sets of conservation equations. Martynov [15] studied two-fluid model with Eulerian–Eulerian approach. Giannadakis E. [16] adopt Eulerian–Lagrangian based two-fluid model. The use of commercial CFD code allows to understand hydrodynamic behavior of flow in detail. S.Som et al. [17], Salvador et al. [18], KaushikSaha et al. [19], Michele Battistoni et al.[20], carried out extensive computational study on cavitation and carried out parametric study based on nozzle geometry, fuel properties, pressure difference & needle movement. Following section illustrate summary of literature referred to understand cavitation in fuel nozzle. The macroscopic parameter, i.e. spray tip penetration and spray cone angle are reported by Payri et al. [21] for biodiesel. Wang et al. [22] carried out detail experimental study for macroscopic parameter of spray for palm oil and used cooking oil at high injection pressure of 300 MPa. Reitz et al. [23] introduced blob injection model, in which liquid drops with a diameter equal to the nozzle diameter injected and droplet break up followed by KH-RT instability.

3.1 Cavitation coupled spray model

Very few literature available for cavitation coupled spray simulation. There are two different approaches researchers are following. The first one is the two-step approach, where two separate calculations using the Eulerian model to simulate cavitation inside nozzle flow and the Lagrangian approach for the outside injector for the spray region. In this method, discrete particles are superimposed on the continuous gas phase. Berge et al. [24,25], Som et al. [26,27,28] and Wang et al. [29] work to predict the near nozzle flow by considering the effect of nozzle turbulence & cavitation on the primary breakup. Berge et al. [24] developed a methodology to couple spray and internal nozzle flow at Anstaltfur Verbrennung List (AVL) and applied it within the framework of AVL-FIRE CFD code. They adopt a two-fluid model for cavitation flow and a discrete droplet model (DDM) for spray simulation. Som et al. [28] implemented kelvin helmholtz-aerodynamic cavitation turbulence (KH-ACT) model in CONVERGE®[30] CFD code to simulate spray characteristics of diesel and bio-diesel and compared it with data from Sandia National Laboratory.

As summarized in literature review number of researcher had tried to understand cavitation phenomenon in fuel injector nozzles using experimental as well as numerical techniques using diesel as a fuel. A large body of work exists in the literature regarding the cavitation phenomenon in fuel injector nozzles. However very few work is published on cavitation induced primary breakup. From the experimental studies a lot has been learned but the confusion of the pattern of the two phase mixture still persists, current accepted perception is that in real scale injectors there is cavity or film of vapor and then a diffused mixture of bubbles and liquid. It has been concluded that the bubbles/cavities do not exactly scale with the geometry rather they have their own length scale depending on the flow field. It is also learnt that the fuel jet break-up process during initial stages of droplet formation in injectors is not yet fully understood. In addition to the aerodynamic effects, turbulence and cavitation play an important role in break-up mechanisms. The cavitation phenomenon is still under research phase for diesel as fuel in fuel injector leading to the scope of understanding the phenomena using biodiesel as fuel in IC engine.

4. Objective of the study

- Detail assessment of an existing two-phase numerical model to study the cavitation phenomena inside fuel injector nozzle by using biodiesel.
- To validate the numerical results with existing experimental data available in the literature for diesel as fuel and explore the limitations of numerical model.
- The hydrodynamics behavior and the size of cavitation bubbles will be investigate with different operating conditions to develop a cavitation map.
- To calculate the flow parameter at the outlet of the nozzle in both cavitation and non-cavitation condition. These results are used to assign the initial condition for spray simulations as a part of the coupling method.
- To investigate the spray characteristics for diesel and biodiesel, by incorporating the effect of cavitation bubble.
- To proposed numerical model for development of a fuel injector nozzle using biodiesel as fuel with a cavitation phenomenon, this could provide a reasonable estimation with different boundary condition, for better prediction of spray breakup.

5. Parameters of the study

The cavitation phenomena mainly govern by operating parameter and geometrical parameter. It is very important to identify non-dimensional number which includes the effect of geometrical and operating parameters. The two important non-dimensional numbers that are used to characterize cavitation nozzle flows are: Cavitation Number (CN) and Discharge coefficient ($\mathrm{C_d}$). The cavitation number is one of the criteria frequently used to determine the appearance of the cavitation and is given by:

$$\mathrm{CN} = \frac{p_b - p_v}{\frac{1}{2}\rho V^2}$$

Wherep_b, and p_v are the back pressure and vapour pressure respectively, V is the velocity at the nozzle outlet. The critical cavitation number (CN_{crit}) corresponds to the instant when cavitation bubbles begin to form at the nozzle entry, or the inception of cavitation. Cavitation flows, characterized by highly fluctuating spatial and temporal parameter. In the fuel injection process the nozzle geometry, needle lift, injection pressure, and back pressure considerably influence the mass flow rate of the fuel and the cavitation phenomenon. The efficiency of the fuel injector nozzle can be represented in terms of a non-dimensional parameter called the discharge coefficient (C_d). The discharge coefficient

(C_d) is the ratio between the actual mass flow rate and the ideal mass flow rate based on loss-free conditions and can be calculated based on Bernoulli's equation as below:

$$C_d = \frac{\dot{m}_{actual}}{\dot{m}_{ideal}}$$

Where $\dot{m}_{actual}$ is the actual mass flow rate through the nozzle, and $\dot{m}_{ideal}$ is the ideal mass flow rate calculated by combining the Continuity and Bernoulli's equation as given by :

$$\dot{m}_{ideal} = A_0\sqrt{2\rho(P_{inj} - P_{back})} \text{ (for non-cavitating flow)}$$

$$\dot{m}_{actual} = CcA_0\sqrt{2\rho(P_{inj} - P_v)} \text{ (for cavitating flow)}$$

Where A_0 and ρ are the outlet area of the nozzle and fluid density respectively. And C_c is the coefficient of contraction. The C_c can be defined as the ratio of the effective area of the liquid jet emanating from the nozzle to the actual exit area of the nozzle. Using the above equation coefficient of discharge can be written as:

$$C_d = \frac{\dot{m}_{actual}}{\dot{m}_{ideal}} = C_c\sqrt{\left(\frac{P_{inj} - P_v}{P_{inj} - P_{back}}\right)} = C_c\sqrt{K}$$

Where K is defined as the Cavitation parameter, which is different from CN and used by Nurik [9]. When the injection pressure of fuel is very high as compared to back pressure K tends to become unity, which leads to discharge coefficient equals to contraction coefficient. Therefore the discharge coefficient at high injection pressure will become equal to the coefficient of contraction. The shape (cylindrical, conical) and size (length, diameter) of the fuel injector nozzle has a considerable effect on C_d. Moreover, the sharp edge inlet, rough edge inlet and rounded inlet show more influence on the discharge coefficient.

6. Hypothesis

During high-pressure injection, fuel flowing inside the fuel injector nozzle holes observed the cavitation phenomenon. The cavitation flow has a predominant effect on the near nozzle flow. It is very important to understand the characteristics of the cavitation flow and its effect on spray formation.

It is assumed that the spray characteristics change dramatically and the atomization process is improved in the presence of cavitation from the inlet to the exit of a nozzle. The collapse of the cavitation bubble promotes the liquid jet breakup. It is concluded that the spray angle improved during developing cavitation and super-cavitation regimes but dropped significantly during the cavitation flip.

Comprehensive multidimensional cavitation coupled spray model has been required to validate above hypothesis. This research work aim to developed two-way coupling method to assess the effect of super cavitation on spray characteristics. The method has been implemented on ECN spray-C type fuel injector with using of bio-diesel as a fuel.

7. Research Methodology

7.1 Cavitation modelling

In the present work single fluid approach will be used with cavitation models of Schnerr-Sauer (SS)[31] and Zwart-Gerber-Belamri (ZGB)[32] in the framework of ANSYS-Fluent platform. In the single-fluid model, the two-phase flow is governed by a set of conservation equations of mass and momentum as given below:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (u_j \rho)}{\partial x_j} = 0$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_i}{\partial x_i}\right)\right]$$

The volume fraction of the liquid phase is used to calculate the mixture density and mixture viscosity.

$$\rho = (1 - \alpha_l)\rho_v + \alpha_l \rho_l$$

$$\mu = (1 - \alpha_l)\mu_v + \alpha_l \mu_l$$

Where ρ_v and ρ_l are the vapor phase and liquid phase density, and μ_v and μ_l are the vapor phase and liquid phase viscosity, respectively. The effective viscosity (μ_{eff}) is the summation of the molecular viscosity (μ_m) and turbulent viscosity (μ_t). The turbulent viscosity can be calculated by:

k-ε model:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$$

k − ω model:

$$\mu_t = \alpha\, \rho \frac{k}{\omega}$$

A source term is required in the mass transport equation to evaluate the phase change between liquid and vapor with this model.

$$\frac{\partial (\alpha_l \rho_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l U) = R_c + R_e$$

Where R_c and R_e are the source term to calculate the rate of mass transfer for condensation and evaporation, respectively. U, α_l, and ρ_l are mixture velocity, the volume fraction of liquid and density of liquid respectively. If there is no mass transfer between phases, right hand side zero. The transport equation proposed by Hirt & Nichols[33] is known as the Volume of Fluid (VOF) model. In the VOF model the volume fraction of the vapor phase (α_v) is calculated by:

$$\alpha_v = 1 - \alpha_l = \frac{\frac{4}{3}\pi R_b^3 n_0}{1 + \frac{4}{3}\pi R_b^3 n_0}$$

Where R_b and n_0 denote the bubble radius and bubble nuclei number density (bubble concentration per unit volume). The growth & collapse of the bubbles can be calculated by using the Rayleigh[34]-Plesset[35] bubble dynamic equation (RP Equation).

$$R\frac{d^2R}{dt^2} + \frac{3}{2}\left(\frac{dR}{dt}\right)^2 + \frac{4\mu}{R}\frac{dR}{dt} + \frac{2\sigma}{\rho R} = \frac{P_v - P_\infty}{\rho}$$

Where R is the bubble radius, $\frac{dR}{dt}$ is the bubble wall velocity, σ is the surface tension, and P_v is the vapor pressure. The RP equation considers vapor pressure (P_v) as a threshold for evaporation and condensation. The mass transfer rate is given below in the table-1 using the RP equation.

Table-1 Different mass transfer source terms for condensation and evaporation

Evaporation ($P_L < P_V$)	
Schnerr-Sauer [29]	$R_e = -C_v \frac{3\rho_l \rho_v}{\rho_m} \frac{\alpha_l(1-\alpha_l)}{R_b} sgn(P_l - P_v)\sqrt{\frac{2\lvert P_l - P_v\rvert}{3\rho_l}}$
Zwart *et al.*[30]	$R_e = F_{vap} \frac{3\alpha_{nuc}\rho_v(1-\alpha_v)}{R_b}\sqrt{\frac{2\lvert P_l - P_v\rvert}{3\rho_l}}$ (F_{vap}=50)
Condensation ($P_V < P_L$)	
Schnerr-Sauer [29]	$R_c = C_c \frac{3\rho_l \rho_v}{\rho_m} \frac{\alpha_l(1-\alpha_l)}{R_b} sgn(P_v - P_l)\sqrt{\frac{2\lvert P_l - P_v\rvert}{3\rho_l}}$
Zwart *et al.*[30]	$R_c = F_{con} \frac{3n_0\rho_v}{R_b}\sqrt{\frac{2\lvert P_l - P_v\rvert}{3\rho_l}}$, ($F_{con}$=0.01)

7.2 Spray Modelling

The atomization of fuel involves the liquid break up in to the droplets through primary and secondary breakup. Assessment of primary and secondary breakup is important in non-evaporating spray modelling. The break up process governs by KH (Kelvin-Helmholtz) and RT (Rayleigh-Taylor) instability at the interface of two fluids. The KH instability generates due to high shear force at the interface and RT instability is due to acceleration in the normal direction of the plane. Present work propose the study of non-evaporating spray characteristics by using discrete phase model (DPM), which is based on the Lagrangian drop Eulerian fluid method. The transport equation is solving for continuous flow in the Euler frame of reference, while discrete phase particles are used to simulate spray formation in a Lagrangian frame of reference. The spherical particle introduced in to the continuous phase & trajectories are calculated by two way coupling. Spray breakup can resolve by the blob injection model. In which liquid droplet, diameter equal to the nozzle diameter, is experience the KH (Kelvin-Helmholtz) and RT (Rayleigh-Taylor) instability. Kelvin-Helmholtz instability exhibited due to relative velocity between the gas and liquid phases create shearing action on the droplet. As a result the parent droplet breaks in to small child droplets. This child droplets experience secondary breakup due to combine effect of KH and RT. Rayleigh-Taylor instability generate due to the inertia of the denser fluid. The droplet breakup process is similar in both KH and HT. As Weber number increases, faster aerodynamic effects led to increases the formation of the smaller droplets. The Wave model used to calculate the aerodynamic effects on droplet breakup.

8. Validation Procedure

8.1 Validation of Cavitation modelling

Qualitative information in a real-size nozzle is essential to understanding cavitation flow and its complete behavior. However, direct observation of cavitation flow in the real condition is difficult due to very small space and time parameters. To observe cavitation in a real-size nozzle, it must be transparent and capable of withstanding high injection pressure and choking condition. A good quality cavitation image will be convenient for the reader and model validation. Winklhofer et al. [12] experimented with a real-sized two-dimensional throttle (transparent rectangular cross-section) working with European diesel fuel, as shown in Fig.4. In the present study, identical geometry & boundary condition has been used to analyze the cavitation flow inside the nozzle. The density of diesel's liquid and vapor phases are considered 830 kg/m^3 and 9.4 kg/m^3, respectively. The viscosity of liquid and vapor phases are considered as 0.00336 kg/ms and 7.006 x 10^{-6} kg/ms, respectively. The grid is sufficiently refined based on grid-independence study and shown in Fig 5.

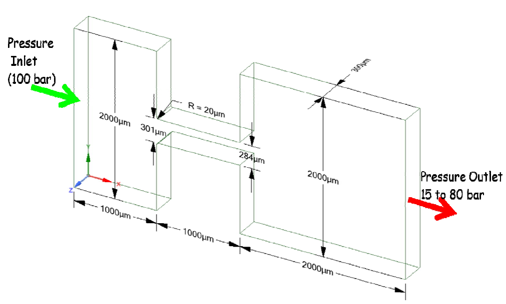


Fig. 4 Computational domain with its dimension

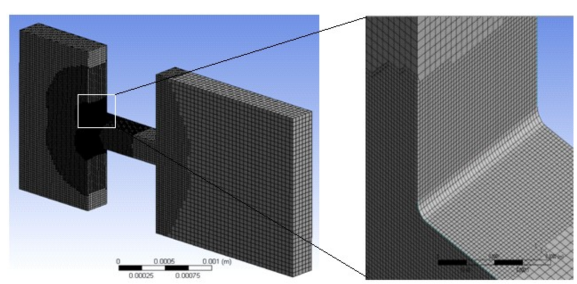


Fig. 5 Grid generation with refinement at throttle inlet

To validate this numerical setup, comparisons were made between the results of simulations and that of an actual experiment by Winklhofer et al. [12] Quantitative & Qualitative validation have been shown in Fig. 6, 7 & Fig. 8. A range of pressure differences has been used to calculate the mass flow rate of fuel. Present results show good agreement with the experimental data of Whinklhfer et al. [12] with a maximum error of 7%. It can be attributed to the assumptions taken in simulations and material properties variations. But simulation results hold a good agreement with respect to experimental results.

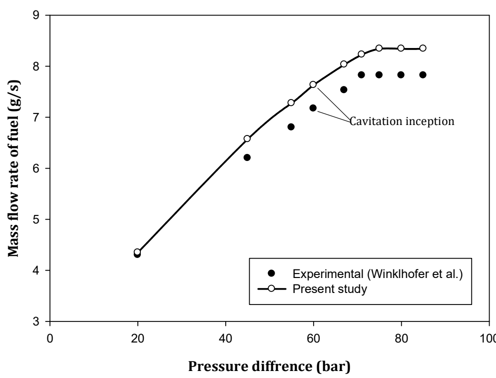


Fig. 6 Quantitative comparison of fuel mass flow rate with experimental results

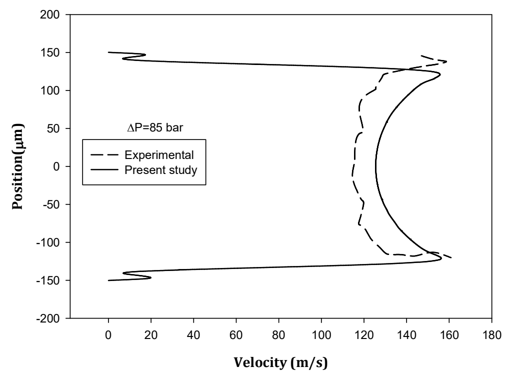


Fig. 7 Comparison of the velocity profiles at 53 μm from the nozzle inlet

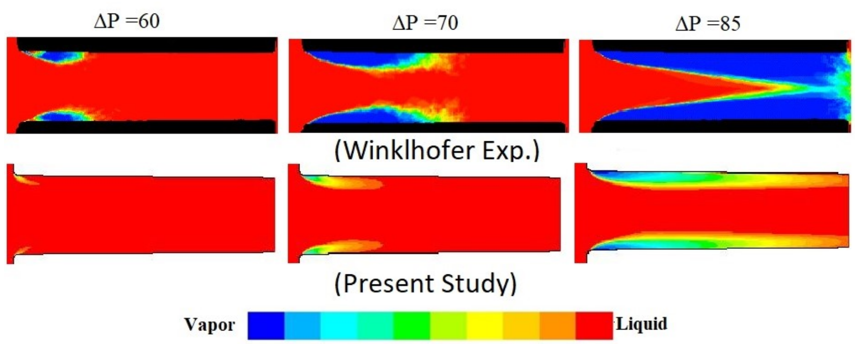


Fig.8 Qualitative comparison of Liquid volume fraction with Winklhofer et al. [12] experimental results

Fig. 7 shows the velocity profile at a 53 μm distance from the inlet of the nozzle for the pressure difference of 85 bar, which are compared with experimental data. This study observes mass flow rate calculated is higher than the experimental claim, which results in the velocity profile being over-predict for both cases. In the experimental images, the red color indicates the liquid fraction and blue color used for the vapor; a similar color map is used for validation purposes, as shown in Fig.8. It is observed that the numerical model accurately predicts the inception of cavitation, although, with a large pressure difference, it under predicts the extent of the cavitation zone. The results obtained in the experimental work are the integrated effect of the light transmitted through the two-phase flow. However, the current results belong to a single plane in the fluid domain.

The accuracy of the numerical solution depends on the appropriate section model and sub-model. The main idea behind this work is to identify the best-suited numerical model for the cavitation flow. The proposed model must be accurate & computationally less costly. The cavitation phenomenon involved gas-liquid flow which is turbulent. A systematic study has been carried out with three different turbulence models, two cavitation models and two multiphase models. The results of the mass flow rate of fuel have been compared with experimental data. The maximum error with different combinations of the model has been calculated and reported in Fig.9. The combination of the VOF+ZGB+ k-ω SST exhibits the least error among all cases. Moreover, LES based turbulence model is widely adopted to capture the detail of bubble and ligament formation. In the present work, the VOF is tuned with a sharp interface capturing method using the k-ω SST turbulence model. The present method required substantially low computational time as compared to the similar work reported by S. Rojas et al. [36] using large eddy simulation (LES).

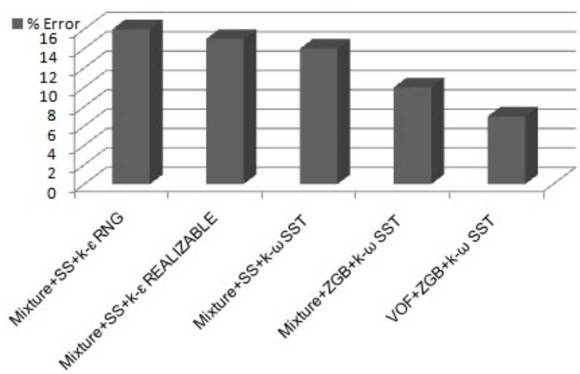


Fig.9 Percentage error in mass flow rate of the fuel with different combinations of numerical models

8.2 Validation of Spray modelling

In the present study, the spray breakup model has been validated with the experimental work carried out by Wang et al. [37]. For validation purposes nozzle with a 0.16 mm diameter is used to inject the diesel at the rate of 2.225 g/s at 300 MPa injection pressure. The fuel injection chamber is symmetrical in shape; hence simulation is carried out with an axi-symmetric computational domain which is created in the ANSYS-Design modeller. The discretization of the domain is done with the use of ANSYS-ICEM, which allows the creation of fine structural mesh in the centre core of the chamber shown in Fig. 10.

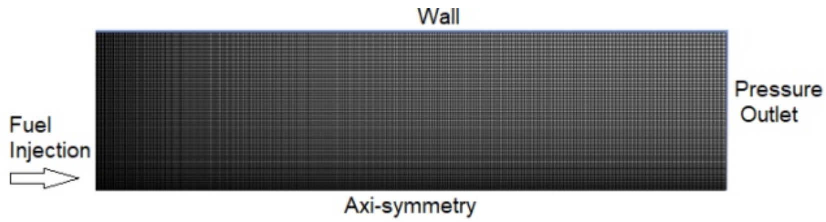


Fig. 10 Axi-symmetric computational domain with detail mesh

The spray tip penetration (STP) of the diesel spray at the different injection times has been calculated. The development of spray has been captured from 0 ms to 1 ms with the time interval of 0.2 ms shown in Fig. 11. To compare the capability of different spray models, the SSD, the Wave and the KHRT models have been implemented. STP grows more quickly in the beginning of the injection and less rapidly in the closing stages. The results obtained at the beginning of the fuel injection are quite reasonable with the experimental trend. The KHRT and the Wave model predict the penetration length reasonably good as compared to the SSD model as shown in Fig. 12. Moreover, the spray shape obtained in the WAVE model is closer to the experimental work of Wang et al. [37].

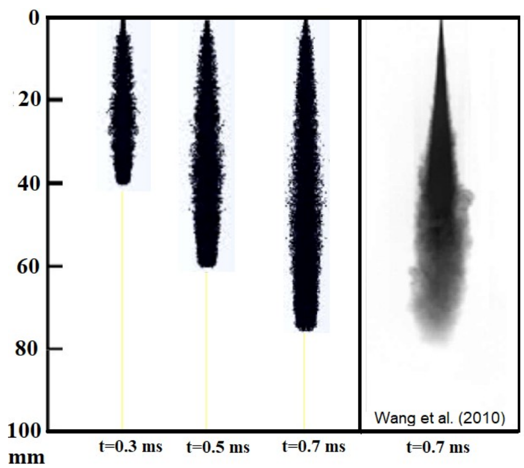


Fig. 11 Spray plum development compared with the experimental data [37]

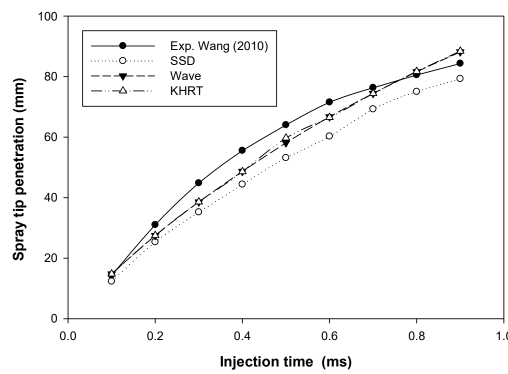


Fig. 12 Comparison of Spray tip penetration of different spray breakup models with experimental data [37]

9. Experimentation Details

9.1 Experimental Test-rig to visualize the Cavitation flow

To visualise the cavitation flow within a transparent nozzle, an experimental setup is developed and manufactured. Fig.13 (a) depicts a schematic diagram and Fig 13 (b) shows a actual image of the experimental setup. The Plunger pump (3 piston, 3HP) has been used to pump the filtered tap water through the nozzle. Utilizing a ball valve and needle valve, the water flow rate can be regulated. The rotameter (0 to 1000LPM) was used to measure the flow rate and the calculated mean exit velocity was based on this result. Before the nozzle, a pressure sensor with a digital display was installed. The visualisation of the internal flow was performed in front of the nozzle using a high-speed camera to obtain still photos. The LED lamp was utilised to illuminate the cavitation bubble in order to capture accurate images. The nozzle is made of a aluminium block and is fabricated through the laser cutting operation

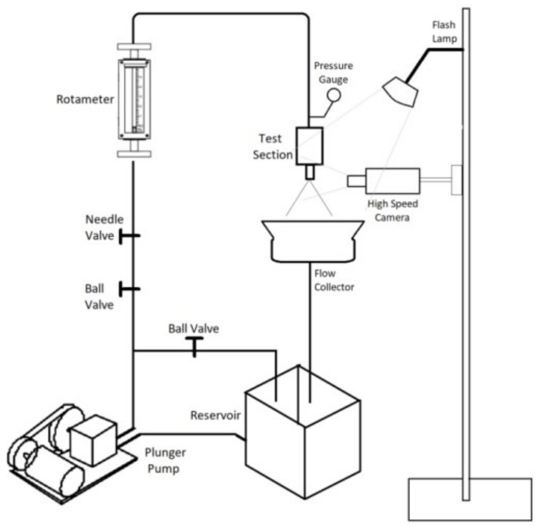


Fig 13(a) Schematic diagram of Test-rig for visualize the cavitational flow

Fig 13 (b) Actual image of the experimental test facility to visualize the cavitational flow

Fig 14 Different Nozzles with its dimension and stages of preparation of nozzle

9.2 Nozzle Design

The cavitation phenomenon occurs inside the nozzle hence it is crucial to design nozzles that permit visual access within the nozzle so that the cavitation bubble may be observed. Furthermore, due to small dimensions of the nozzle, extremely precise machining is needed. In light of all these factors, aluminium nozzle-1 has been designed and covered with an acrylic sheet to visualize the cavitation flow. The specific purpose nozzle-2 design form aluminium with different nozzle length in right & left side. Finally nozzle-3 has been manufacture by using acrylic as shown in Fig. 14.

9.3 Experimental Test Rig to visualize the spray development

In the present work, a single-hole injector with a nozzle size of 0.186 mm was used. The injector operated at a maximum working pressure of 1500 bar. To control the injection pressure, a motor-driven fuel pump was employed. The spray characteristics of both diesel and Tallow-biodiesel fuels were studied, including the penetration length and spray cone angle. The development process of the spray was captured using a high speed video grapy. Subsequently, the Sauter Mean Diameter (SMD) was determined using CFD. The motor operated fuel pump is used to transferring fuel from the fuel tank. Two injectors were used in the setup. Injector 1 functioned as a nozzle, while Injector 2 collected the fuel supplied, allowing for the measurement of flow rate. The fuel pump operated as a distribution type, ensuring both injectors received an equal amount of flow and pressure. To monitor the injection pressure, a pressure gauge was placed at Injector 2. After each run, the fuel collected from Injector 2 was measured, while the injection spray video was recorded from Injector 1. This data facilitated the calculation of the collected fuel amount and time, thereby enabling the determination of the flow rate at specific pressures. For the evaluation of cone angle and spray penetration, different blend of Tallow bio-diesel were used. Four different blend are B00 (100% Diesel), B10 (10% Biodiesel, 90% Diesel), B20 (20% Biodiesel, 80% Diesel) and B100 (100 % Biodiesel).

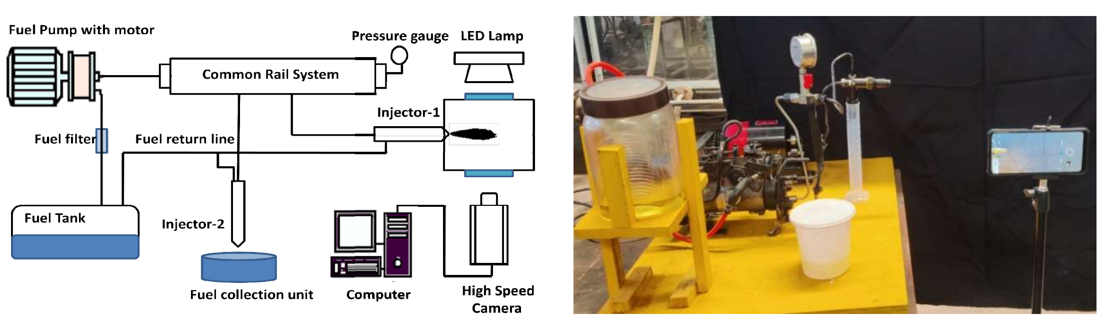


Fig 15 Schematic diagram and actual image of the experimental test facility to visualize the spray development

10.Results and Discussion

10.1 Cavitation coupled spray breakup

(a) ECN Spray C Fuel Injector nozzle and boundary condition

The spray C fuel injector nozzle, developed as part of the 4th generation fuel injectors by the Engine Combustion Network (ECN), is specifically designed to explore cavitation flow dynamics. In our study, we employed a detailed geometrical model of the spray C fuel injector to implement and validate two-step coupling method. Obtaining dimensional specifics from the Engine Combustion Network (ECN), a two-dimensional fluid domain was constructed, and simulations were conducted using the ANSYS-Fluent commercial CFD software. The outcomes, presented in terms of the Cavitation number, unequivocally indicate that cavitation intensity escalates with a decrease in the cavitation number. This underscores the crucial role of cavitation dynamics in the performance of the spray C fuel injector.

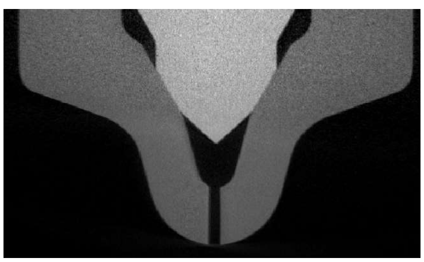


Fig 16 Three dimensional X-ray CT image at the injector's central cross-section

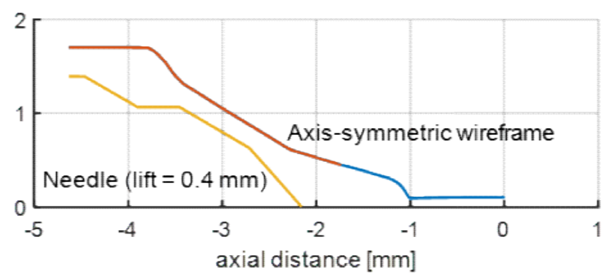


Fig 17 Two-dimentional axis-symmetric wireframe geometry

In this study, we employed identical boundary conditions and fuel to ensure a direct comparison between our obtained results and the experimental data available from the Engine Combustion Network. This approach enhances the reliability and relevance of our findings by establishing a consistent and comparable basis for evaluation.

Table- 2 Boundary conditions for nozzle flow and spray simulation

Type of Nozzle	Fuel	Internal nozzle flow condition (k-ω+ VOF + ZGB)			Chamber conditions for Spray simulation (DPM + SST k-ω)	
		Inlet Pressure (MPa)	Outlet Pressure (MPa)	Turbulence Model	Density (kg/m^3)	Temp. (K)
ECN Spray C	n-Dodecane	160	6	SST k-ω	22.8	303

(b) Two-step coupling method

Coupling the internal flow with spray atomization, particularly in ANSYS-Fluent, poses considerable challenges. As mentioned in previous work, despite the activation of the shared memory option, concurrent utilization of the DPM model with the VOF model for particle tracking is not feasible. This necessitates exploring alternative coupling approaches. These include orifice coordinate position, geometry size, outlet pressure, flow rates, turbulent kinetic energy, turbulent dissipation rate, and gas-liquid two-phase volume fraction, along with temperature. These parameters are calculated in each grid cell of the outlet cross-section and then stored in a *.inj file. To create the *.inj file we create the txt file in format as: (($x\ y\ z\ u\ v\ w\ Diameter\ Temperature\ mass-flow)\ name$). Where x, y, and z are the position coordinates and u, v, and w is the velocity in respective dimensions. The conceptual diagram for proposed method is shown in Fig.18.

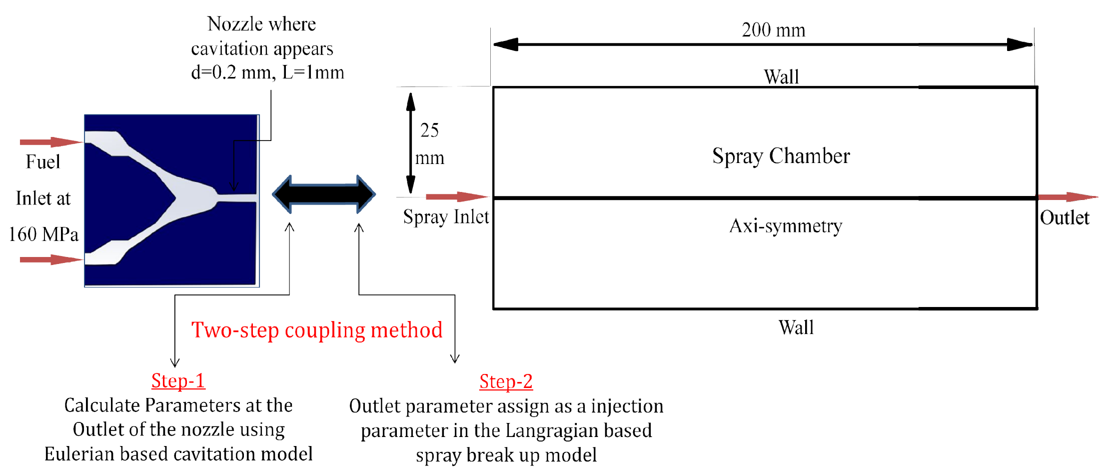


Fig. 18 Conceptual diagram of Two-step Cavitation coupled spray breakup method

(c) Validation of the coupled method

In collaboration with the Engine Combustion Network (ECN), Payri et al. [38] conducted experimental investigations on a ECN spray fuel injector nozzle. Their focus was on determining the spray cone angle for n-Dodecane fuel at an injection pressure of 1600 bar. Subsequently, in another study, Payri et al. [39] developed cavitation induced Eulerian-Eulerian single-step method and implemented on ECN spray C fuel injector. However, this method required grid refinement down to the sub-micron level to detect droplets without introducing discrete particles, making its application limited due to its high computational cost. In current research, we introduced a novel methodology—the Eulerian-Lagrangian two-step approach. This method was implemented in the ECN spray c injector nozzle and cavitation coupled spray calculation has been obtained. Notably, this approach offers enhanced accuracy, and the results align more consistently with the experimental data on spray cone angles provided by ECN shown in Fig.19. The developed spray patterns with our proposed method exhibit a higher level of realism.

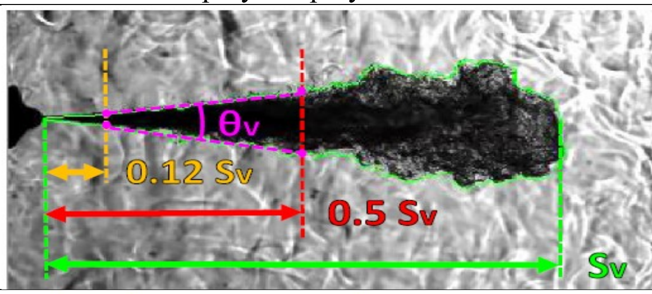
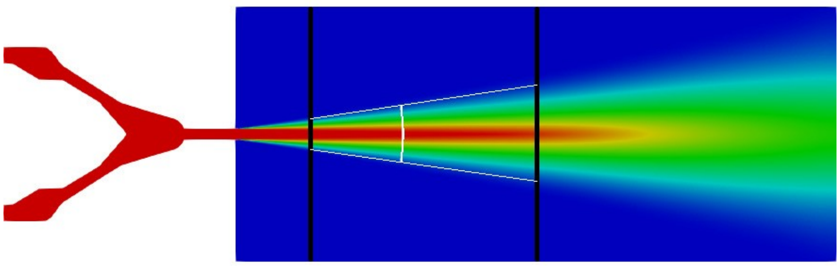
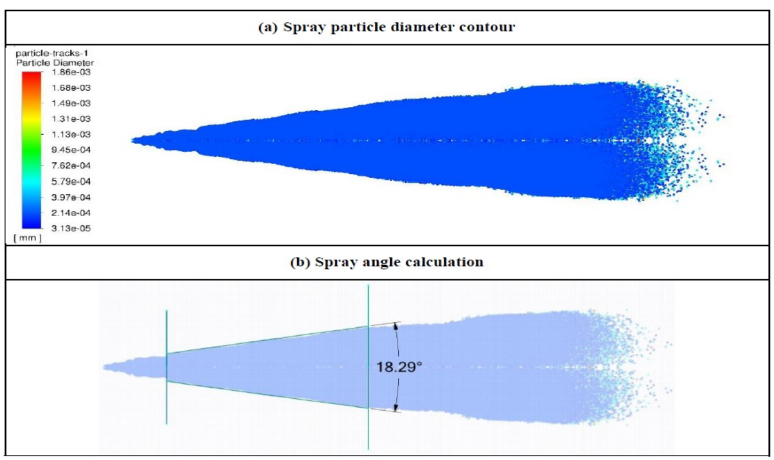
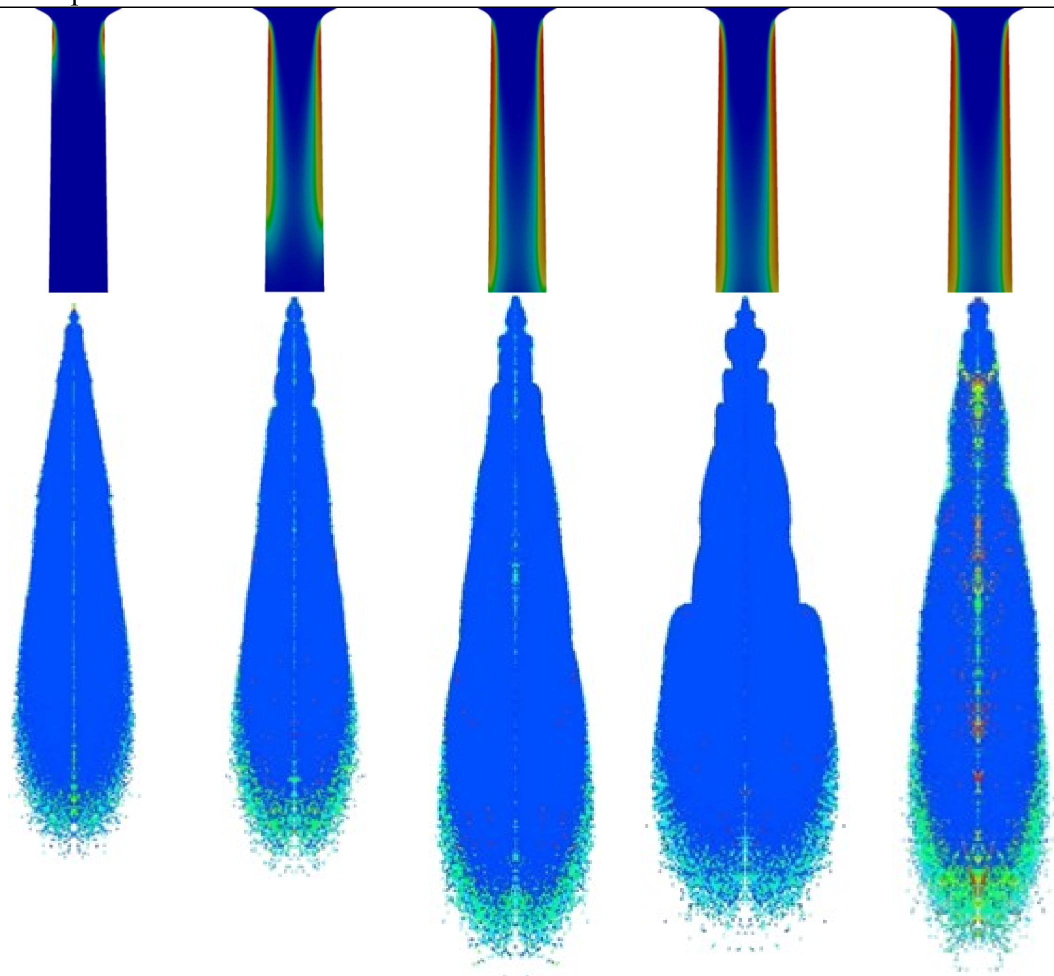
ECN Spray-C Spray	Cone Angle (θ) in Degree
	**17.78°** (Experimental results of R. Payri et al. [38] & ECN)
 (Calculated by using Eulerian-Eulerian coupling approach)	**18.76°** (Single-step Coupled method R. Payri et al. [39])
 (Calculated by using Eulerian-Langrangian coupling approach)	**18.29°** Two-step coupled method **(Present Method)**

Fig. 19 Validation of proposed two-step coupling method with ECN data

(d) Results of cavitation flow and spray for ECN Spray C fuel injector

It is found that, cavitation moves downstream as the pressure differential increases. The intensity of cavitation increases as the pressure difference between the input and output expands. The various stages of cavitation have been identified, as Fig. 20 illustrates. These stages include cavitation inception, cavitation development, super cavitation, and cavitation flip. The corresponding spray images were acquired using a two-step coupling method, revealing a noticeable increase in the cone angle during the super cavitation stage.

ΔP= 36 MPa	ΔP= 40 MPa	ΔP= 86 MPa	ΔP= 140 MPa	ΔP= 186 MPa
Cavitation Inception	Development of Cavitation	Development of Cavitation	Super cavitation	Hydraulic flip



θ = 15.6˚	θ =15.8˚	θ =16.3˚	θ =17.4˚	θ =17.3˚

Fig. 20 Different stages of Cavitation coupled with spray development for ECN spray-C fuel injector

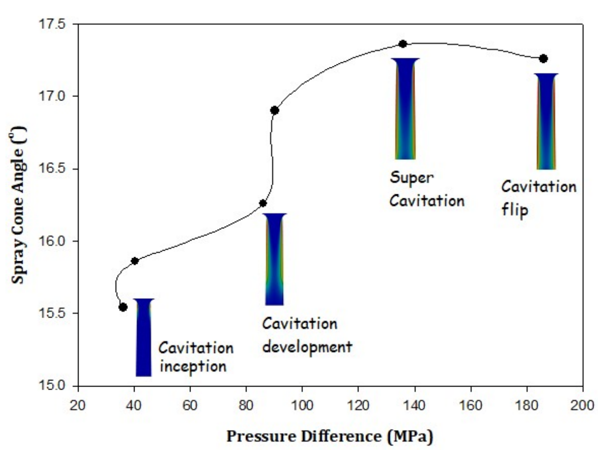


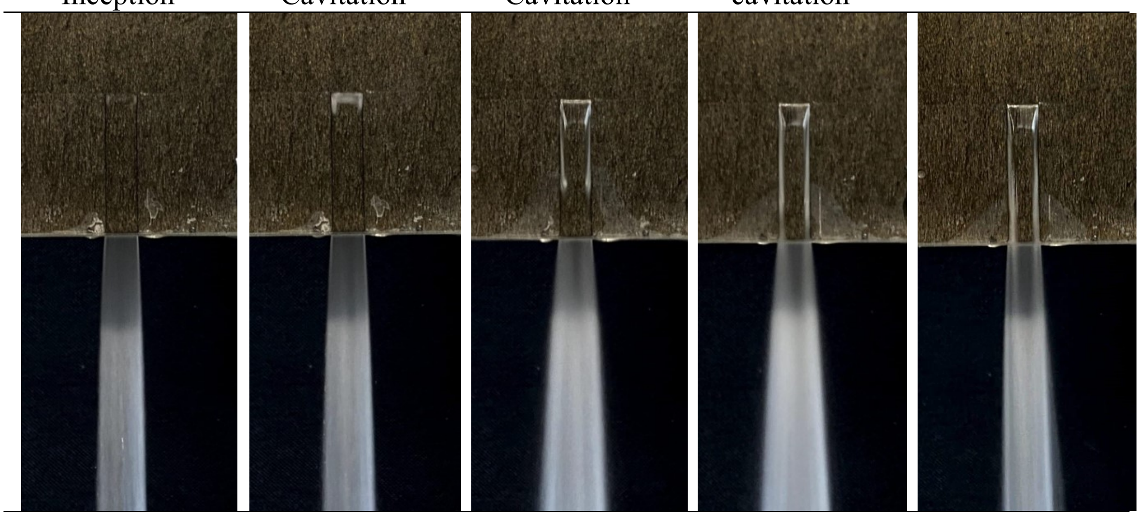
Fig. 21 Spray cone angle at various pressure difference and stages of cavitation

Fig 21 shows the spray cone angle at various stages of cavitation. The highest cone angle obtained with super cavitation stage. This observation aligns with the hypothesis of the current research.

10.2 Experimental results with scaled up nozzle

The experimental investigation involved a rectangular cross-section nozzle [Width(W_n) = 5mm and Length (L) = 1 mm]. The water supplies through the nozzle at the injection pressures ranged from 0.1 MPa to 0.5 MPa. High-speed cameras (Up to 1000 frame per second) were employed to capture still images at various injection pressures. Notably, cavitation bubbles were seen to begin at 0.2 MPa injection pressure, and the magnitude of the bubbles increases with injection pressure. The experimental investigation also examines the many stages of cavitation flow, including its inception; development, super cavitation, and cavitation flip stages. The different stages of cavitation and the related spray cone angle are shown in Fig. 22.

ΔP= 0.15 MPa	ΔP= 0.2 MPa	ΔP= 0.3 MPa	ΔP= 0.35 MPa	ΔP= 0.45 MPa
Cavitation Inception	Development of Cavitation	Development of Cavitation	Super cavitation	Cavitation Flip



$\theta = 5^{\circ}$	$\theta = 6.7^{\circ}$	$\theta = 10.7^{\circ}$	$\theta = 12.9$	$\theta = 7.7^{\circ}$

Fig 22 Different cavitation stages coupled with spray results with scaled up nozzle

It is evident that the highest spray cone angle was recorded during super cavitation shown in Fig. 23. Vapour bubbles reach the nozzle outlet during super cavitation and collapse there. The liquid jet will experience turbulence as a result, improving the breakup mechanism. Hydraulic flip occurs when there is an additional increase in injection pressure, which causes air entrainment inside the nozzle and flow separation from the nozzle surface. Hydraulic flip reduces the spray cone angle. To characterise the cavitation flow and spray breakup, experimental data are used to calculate non-dimensional numbers such as the Reynolds number, discharge coefficient, and cavitation number. The cavitation number indicates the cavitation intensity within the nozzle. It is noted that the Cavitation number decreases with injection pressure. Experiment shows that when Cavitation number drops below unity the bubbles start to form as shown in Fig.24.

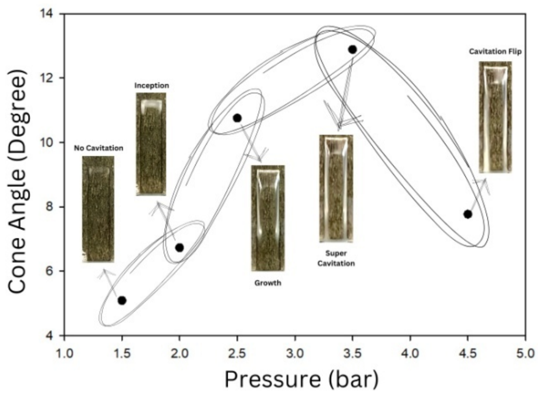


Fig. 23 Spray cone angle vs. Injection pressure

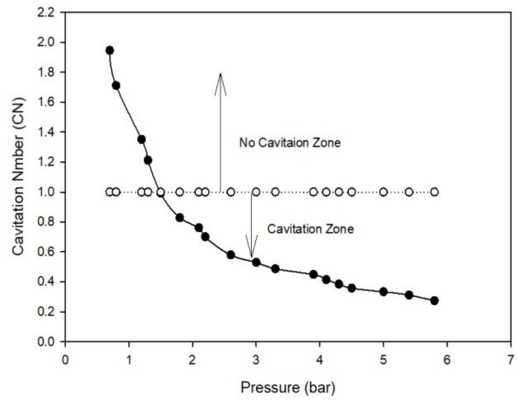


Fig. 24 Injection pressure vs. Cavitation number

The goal of the nozzle-2 is to confirm how supercavitation affects the spray angle. In order to determine the nozzle length, numerical simulation has been used. This ensures that only one side of the nozzle exhibits supercavitation flow, while the other side must meet non-cavitation conditions. Figure 25 illustrates how the nozzle's left side shows supercativation, while the nozzle's right side experiences non-cavitation flow due to a variation in nozzle height. The diesel fuel spray image clearly demonstrates a bigger spray on the left side where super cavitation is present. This demonstrates that super cavitation alone is the cause of the cone angle increases. Ultimately, a cavitation map was developed using nozzle-3 and other fluids. The various phases of cavitation using diesel fuel are depicted in Fig. 26. Spray brake up is much enhanced in the supercavitation stage. These results validate the research premise and the outcomes of the two-step cavitation-coupled spray break-up methods. Fig.27 shows comparision of cone angle for three different fluids at different injection pressure. The cavitation number shows a linear decline with Reynolds number prior to cavitation start. The relationship is no longer linear once cavitation flow has formed inside the nozzle, as seen in Fig. 28. This is because the cavitation number and the Reynolds number (Re) are inversely related to the square of the velocity. The inception of the cavitaion delayed as increasing in L/W ratio.

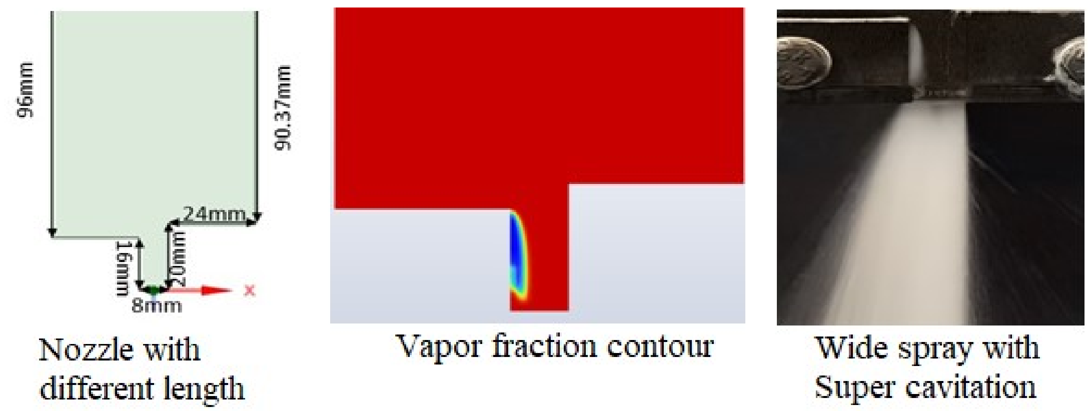


Fig. 25 Nozle-2 with wider diesel spray in super cavitation side

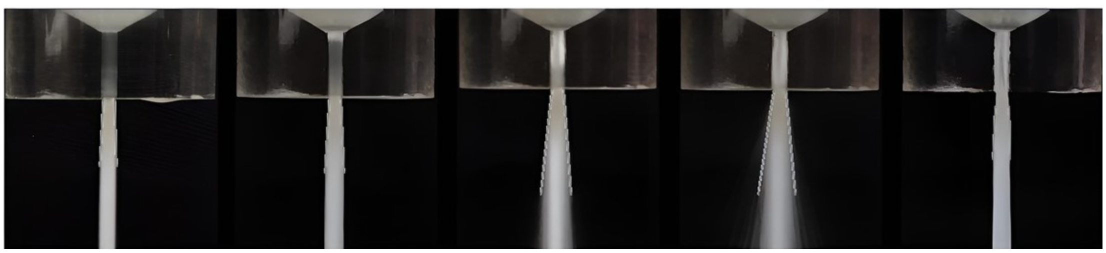


Fig. 26 Nozle-3 with wider diesel spray in super cavitation stage

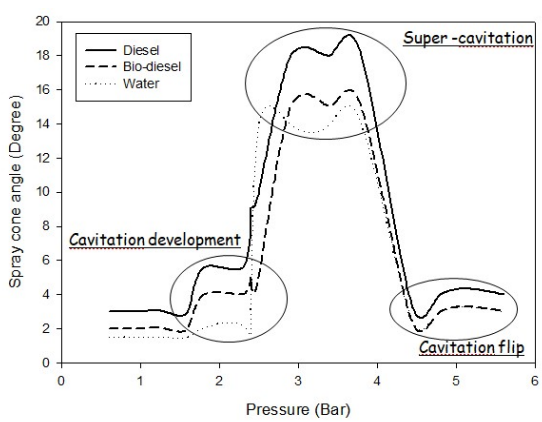


Fig. 27 Spray cone angle vs pressure for different fluids

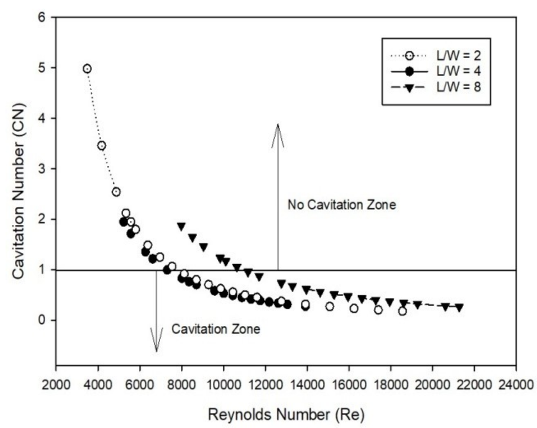
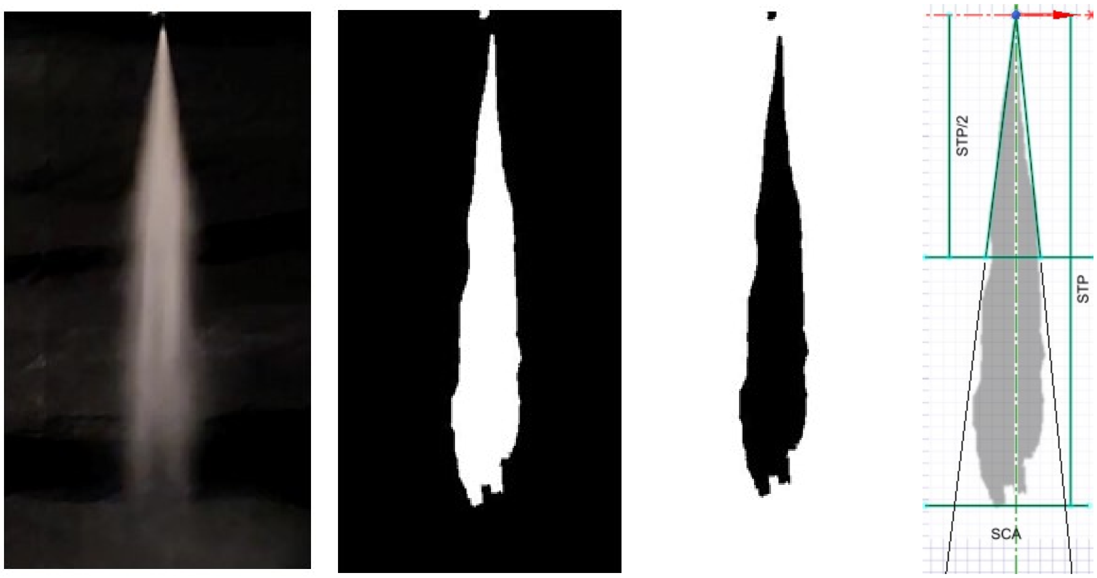


Fig.28 Reynolds number vs Cavitation number for different L/W ratio

10.3 Spray characteristics of Diesel and Tallow-Biodiesel

(a) Image processing method for spray characteristics.

To process a photographic image of diesel and biodiesel spray into a binary format, the initial step involves converting the image to greyscale. This conversion can be efficiently carried out using image editing software such as Adobe Pro. Greyscale images possess a single channel, and the intensity of each pixel is represented by a singular value ranging from 0 to 255. Subsequently, a threshold needs to be applied to transform the greyscale image into a binary image shown in Fig 29. Otsu's method, a thresholding technique, is employed to automatically determine the optimal threshold value for this conversion. Otsu's method operates on the principle of maximizing the between-class variance of the image. This variance is calculated based on the greyscale intensity values between the two classes: the foreground and the background. The threshold value that maximizes this between-class variance is then selected as the optimal threshold value.



Original Image Processed Image

Fig 29 Image processing steps and cone angle calculation

(b) Spray development of diesel and Tallow bio-diesel fuel

The spray characteristics can be defines with spray tip penetration and cone angle. The development of spray has been capture by using high speed camera (1000 FPS) and processed images of the spray have been shown in Fig. 30 for diesel and tallow bio-diesel. The experiments have been conducted with diesel and different blends of tallow bio-diesel (B20, B40, and B100). The penetration length of the diesel and tallow bio-diesel fuel spray as a function of time after the start of injection is shown in Fig. 31. At the start of injection, both fuels have similar penetration lengths, but as time progresses it can be observed that the diesel spray has a shorter penetration length compared to tallow bio-diesel fuel. This is likely to the differences in the physical properties of the fuel, such as viscosity and surface tension, which affect the spray behaviour.

The fuel density, kinetic viscosity, and surface tension of Tallow bio-diesel are higher than the diesel fuel. This indicates that the tallow biodiesel spray has a more stable and consistent behaviour compared to diesel fuel. The viscous nature of tallow biodiesel create resistance in the disintegration process of spray, it also results in narrow spray angle as compared to diesel fuel shown in Fig.32. Overall, the graph suggests that the tallow biodiesel spray has a more stable penetration length and narrow cone angle as compared to diesel fuel.

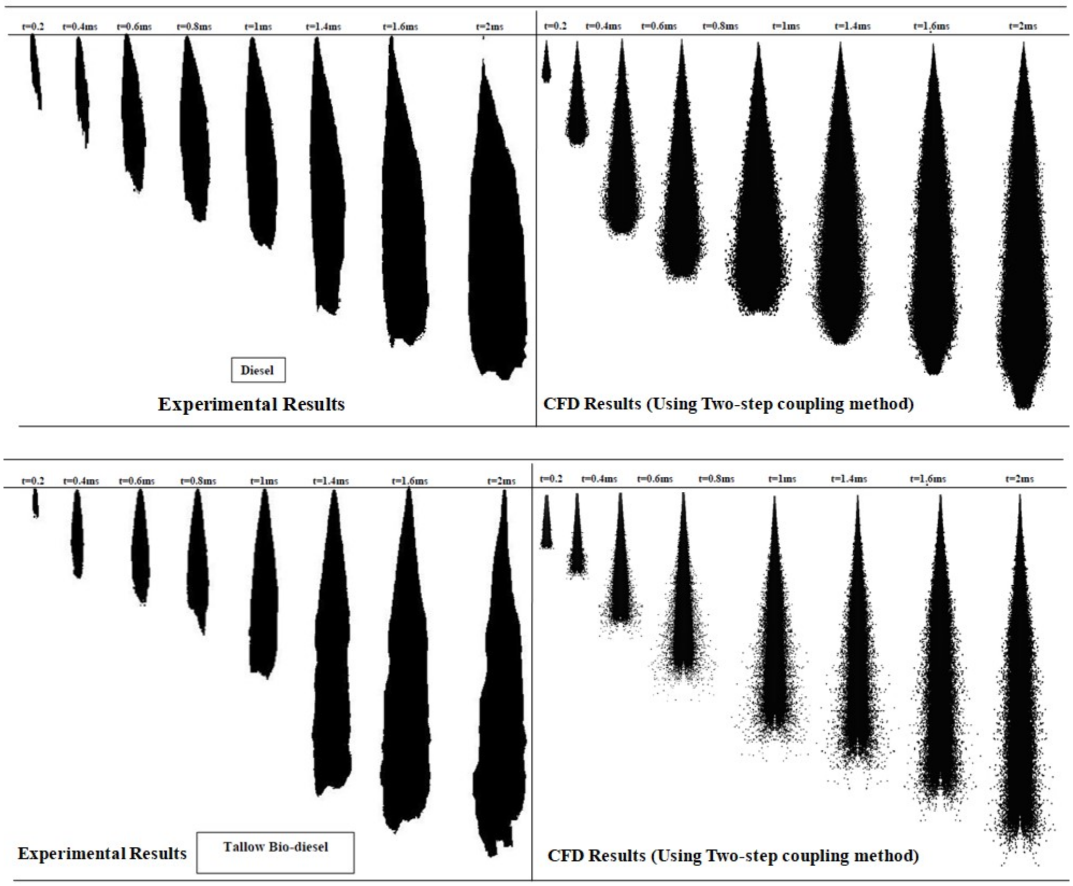


Fig 30 Spray Development images for diesel and Tallow bio-diesel using experiment and two-step methodology

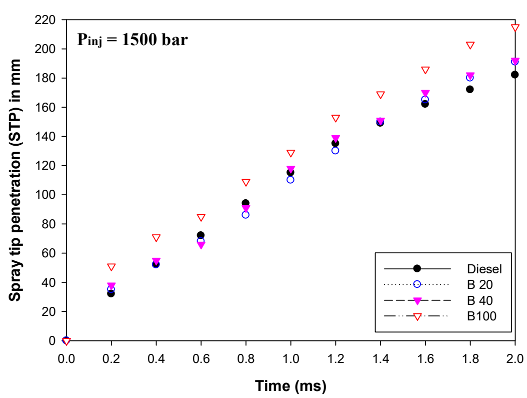


Fig. 31 Spray tip penetration for Diesel and different blend of Tallow bio-diesel

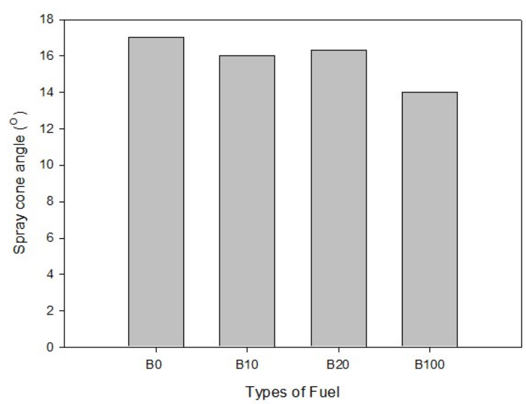


Fig. 32 Spray cone angle for Diesel and different blend of Tallow bio-diesel

One of the key parameter to consider when addressing spray characteristics is the sauter mean diameter (SDM). Using spray modelling, the SMD for diesel and tallow biodiesel was determined and is shown in Fig. 33. Because of its high viscosity, tallow biodiesel exhibits higher SMD values lead to poor spray breakup.

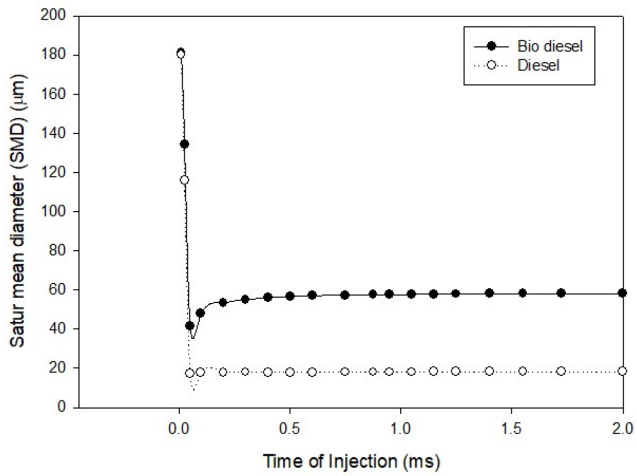


Fig. 33 Sauter mean diameter for Diesel and Tallow bio-diesel fuel

11. Limitations of the study

- While the present study reports the effect of cavitation on the spray, it is essential to acknowledge that cavitation has the potential to cause erosion within the nozzle. This aspect requires dedicated investigation to comprehensively understand its implications.
- In a real fuel injector, the injection process involves the dynamic behavior of the mesh, particularly when fuel is injected by lifting the needle. The present study, however, does not include the effect of needle lift.
- The relationship between high injection pressure and the resulting high injection velocity of the fuel may induce compressible behavior in the fluid. This phenomenon demands further investigation to unravel its implications and understand the fluid dynamics under such conditions.
- The injection process involves the presence of air in the cylinder, considered a non-condensable gas. Consequently, it becomes imperative to develop a three-phase model rather than relying on the two-phase modeling approach employed in the present study.

12.Conclusions

- The combination of the k-ω SST turbulence model, the ZGB cavitation model, and the VOF sharp interface model proves to be the most accurate in capturing cavitational flow within fuel injector nozzles. Calculated mass flow rates using this combination exhibit a maximum error of 7% compared to experimental results, indicating reliable performance.
- The KHRT and Wave models provide reasonable predictions of penetration length, surpassing the SSD model with a maximum error of 14% against experimental data. Particularly, the Wave model yields more realistic spray shapes.
- The newly developed Eulerian-Lagrangian two-step coupling method has been successfully validated using data from the Engine Combustion Network's Spray-C type fuel injector. This method effectively predicts the impact of inner nozzle cavitation on the spray breakup mechanism.
- Both numerical simulations and experimental observations have successfully identified distinct stages of cavitation, encompassing cavitation inception, cavitation development, supercavitation, and cavitation flip. It is evident that supercavitation notably impacts droplet breakup, leading to an increase in the spray cone angle. Conversely, cavitation flip results in a reduction of the spray cone angle, showcasing its influence on spray behavior.
- The intensity of cavitation exhibits a direct correlation with the decrease in cavitation number, approaching unity. During non-cavitational flow, the discharge coefficient is primarily influenced by the Reynolds number. However, with the onset of cavitation within the nozzle, the discharge coefficient ceases to be governed by the Reynolds number and instead becomes dependent on the cavitation number.
- A comparative analysis of spray characteristics between diesel and tallow biodiesel was conducted, revealing noteworthy distinctions. The investigation revealed that tallow biodiesel exhibits a greater spray penetration length compared to diesel fuel. This phenomenon can be attributed to the viscous nature of biodiesel, which impedes fuel disintegration, resulting in a narrower spray cone angle compared to diesel fuel. Interestingly, super cavitation mechanisms were observed to aid in increasing the spray cone angle of tallow biodiesel, mitigating the effects of its viscosity.

13. References

[1] ExxonMobil. (2014). The Outlook for Energy: A view to 2040. Technical report.

[2] Scitesch, G. (2003). Modeling engine spray and combustion process. New York: Springer.

[3] Baumgarten, C. (2006). Mixture formation in internal combustion engines. Springer Science & Business Media.

[4] Saha Kaushik (2006). Modelling of cavitation in nozzles for diesel injection applications, 2014, PhD thesis, Waterloo, Ontario, Canada.

[5] Schmidt, D. P., & Corradini, M. L. (2001). The internal flow of diesel fuel injector nozzles: a review. International Journal of Engine Research, 2(1), 1-22.

[6] Cui J, Lai H, Feng K, Ma Y. (2018). Quantitative analysis of the minor deviations in nozzle internal geometry effect on the cavitating flow. Experimental Thermal and Fluid Science;94:89-98.

[7] Bergwerk, W. (1959). Flow pattern in diesel nozzle spray holes. Proceedings of the Institution of Mechanical Engineers, 173(1), 655-660.

[8] Nurick, W. H. (1976). Orifice cavitation and its effect on spray mixing. ASME Journal of Fluids Engineering, 98(4), 681-687.

[9] Soteriou, C., Andrews, R., & Smith, M. (1995). Direct injection diesel sprays and the effect of cavitation and hydraulic flip on atomization. SAE transactions, (950083).

[10] Chaves, H., Knapp, M., Kubitzek, A., Obermeier, F., & Schneider, T. (1995). Experimental study of cavitation in the nozzle hole of diesel injectors using transparent nozzles. SAE transactions, (950504).

[11] C. Arcoumanis, M. Gavaises, B. French (1997). Effect of fuel injection process on the structure of diesel sprays, SAE transactions, (1997-01-1246).

[12] Winklhofer E, Kull E, Kelz E, Morozov A. (2001). Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions, ILASS-EUROPE, Zurich, Switzerland.

[13] Payri, R., Salvador, F. J., Gimeno, J., & Venegas, O. (2013). Study of cavitation phenomenon using different fuels in a transparent nozzle by hydraulic characterization and visualization. Experimental Thermal and Fluid Science, 44, 235-244.

[14] D Schmidt, P. Rutland, C. J. Corrandini, P. Roosen, P., and O. Genge (1999). Cavitation in 2-D Asymmetric Nozzles," SAE Paper No. 1999-01- 0518.

[15] S Martynov (2005). Numerical Simulation of the Cavitation Process in Diesel Fuel Injectors, Ph.D. thesis, The University of Brighton.

[16] E. Giannadakis, M. Gavaises, H. Roth, and C. Arcoumanis (2004). Cavitation Modeling in Single-Hole Diesel Injector Based on Eulerian-Lagrangian Approach, Proceedings of the THIESEL International Conference on Thermo- and Fluid Dynamic Process in Diesel Engines, Valencia, Spain.

[17] S. Som, S. K. Aggarwal, E.M. El-Hannouny, and D.E. Longman, (2010). Investigation of nozzle flow and cavitation characteristics in a diesel injector, ASME Journal of Engineering for Gas Turbine and Power, 132:1-12.

[18] Salvador, F.J, Martínez-Lopez, J, Caballer, M, De Alfonso (2013). Study of the influence of the needle lift on the internal flow and cavitation phenomenon in diesel injector nozzles by CFD using RANS methods, Energy Conversion and Management, 246–256.

[19] SahaKaushik, Ehab Abu-Ramadan, Xianguo Li (2013). Modified Single-Fluid Cavitation Model for Pure Diesel and Biodiesel Fuels in Direct Injection Fuel Injectors, Proceedings of the ASME 2012 Internal Combustion Engine Division Fall Technical Conference, University, Canada.

[20]Michele Battistoni, SibenduSom, and Douglas E Longman (2014). Comparison of mixture and multifluid models for in-nozzle cavitation prediction. Journal of Engineering for Gas Turbines and Power, 136(6):061506.

[21] F. Payri, V. Bermudez, R. Payri, and F. J. Salvador (2004). The influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzles. Fuel, 83:419-431.

[22] Wang X, Huang Z, Kuti OA (2010). Experimental and analytical study on biodiesel and diesel spray characteristics under ultra-high injection pressure, Int J Heat Fluid Flow, 31:659–666.

[23] Reitz R.D., Diwakar J. (1986). Effect of drop break-up on fuel sprays, SAE Technical Paper No.860469.

[24] Berg EV., Edelbauer W., Tatschl R., Volmajer M., Kegl B., Alajbegovic A., Ganippa L., (2003). Validation of a CFD model for coupled simulation of nozzle flow, primary fuel jet break-up and spray formation. InInternal Combustion Engine Division Spring Technical Conference 2003 Jan 1, Vol. 36789, (pp. 171-180).

[25] Berg EV, Edelbauer W, Alajbegovic A, Tatschl R., (2003). Coupled calculation of cavitating nozzle flow, primary diesel fuel break-up and spray formation with an Eulerian

multi-fluid-model. InProc. 9th International Conference on Liquid Atomisation and Spray Systems, pp. 12-02.

[26] Som S., Ramirez AI., Aggarwal SK., Kastengren AL., El-Hannouny E., Longman DE., Powell CF., Senecal PK., (2009). Development and validation of a primary breakup model for diesel engine applications. SAE Technical Paper, pp 20.

[27] Som S., Aggarwal S., (2009). Assessment of atomization models for diesel engine simulations. Atomization and Sprays; 19(9).

[28] Som S., Aggarwal SK., (2010). Effects of primary breakup modeling on spray and combustion characteristics of compression ignition engines. Combustion and flame, 1;157(6):1179-93.

[29] Wang Y, Ge HW, Reitz RD., (2010). Validation of mesh-and timestep-independent spray models for multi-dimensional engine CFD simulation. SAE International Journal of Fuels and Lubricants;3(1):277-302.

[30] Richards KJ.,Senecal PK., (2008).Pomraning E. CONVERGE™ (Version 1.2) manual. Middleton, WI: Convergent Science, Inc.

[31] Schnerr G., and Sauer J. (2001). Physical and numerical modeling of unsteady Cavitation dynamics, Fourth international conference on Multiphase Flow(ICMF-2001), New Orleans, LA.

[32] Zwart P., Gerber A., and Belamri T. (2004). A Two-Phase Flow Model for predicting Cavitation dynamics, Fifth international conference on Multiphase Flow, Yokohama, Paper No. 152.

[33] Hirt, C. W., & Nichols, B. D., (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics, 39(1), 201-225.

[34] Rayleigh, L., (1917). On the pressure developed in a liquid during the collapse of a spherical cavity. The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science, 34(200), 94-98.

[35] Plesset, M. S., (1949). The dynamics of cavitation bubbles. Trans. ASME, J. Appl. Mechanics, 16, 228–231.

[36] Sanmiguel-Rojas E, Gutierrez-Castillo P, del Pino C, Aunon-Hidalgo JA. (2019). Cavitation in transient flows through a micro-nozzle. Journal of Fluids Engineering. 1;141(9).

[37] Wang X, Huang Z, Kuti OA (2010). Experimental and analytical study on biodiesel and diesel spray characteristics under ultra-high injection pressure, Int J Heat Fluid Flow, 31:659–666.

[38] R. Payri, J. Gimeno, J. Cuisano, and J. Arco, (2016). Hydraulic characterization of diesel engine single-hole injectors, Fuel, vol. 180, pp. 357–366,

[39] R. Payri, J. Gimeno, P. Martí-Aldaraví, and M. Martínez, (2021). Validation of a three-phase Eulerian CFD model to account for cavitation and spray atomization phenomena, Journal of the Brazilian Society of Mechanical Sciences and Engineering, vol. 43, no. 4,

14. List of Publication

Journal Paper

(1) Mehul Bambhania, Nikul Patel “Numerical modelling of the cavitation flow in throttle geometry”, Journal of Applied Fluid Mechanics, Vol. 16, No. 2, pp. 257-267, 2023 (SCI & Scopus) (H index-34)

Journal of Applied Fluid Mechanics, Vol. 16, No. 2, pp. 257-267, 2023.
Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
https://doi.org/10.47176/jafm.16.02.1348

Numerical Modeling of the Cavitation Flow in Throttle Geometry

M. Bambhania† and N. K. Patel

Mechanical Engineering Department, Faculty of Technology & Engineering, The Maharaja Sayajirao University of Baroda, Vadodara, Gujarat, 390002, India

†Corresponding Author Email: mehul.bambhania-med@msubaroda.ac.in

(Received June 14, 2022; accepted September 29, 2022)

ABSTRACT

(2) Mehul Bambhania, Nikul Patel “ Hydrodynamic Cavitation in the Fuel Injector Nozzle and its Effect on Spray Characteristics: A Review”, Journal of Heat and Mass Transfer Research, Vol. 10, pp. 1-20, 2023 (Scopus)

Journal of Heat and Mass Transfer Research 10 (2023) 1 - 20

Semnan University
Journal of Heat and Mass Transfer Research
Journal homepage: https://jhmtr.semnan.ac.ir

Review Article

Hydrodynamic Cavitation in the Fuel Injector Nozzle and its Effect on Spray Characteristics: A Review

Mehul P. Bambhania*, Nikul K. Patel

Mechanical Engineering Department,Faculty of Technology & Engineering,The Maharaja Sayajirao University of Baroda, Vadodara, India.

PAPER INFO

Paper history:

Received: 2022-12-02
Revised: 2023-06-20
Accepted: 2023-06-24

ABSTRACT

The performance of internal combustion engines can be improved by optimizing fuel spray characteristics. However, high injection pressures and small nozzle diameters in modern fuel injectors result in cavitation flows inside the nozzle, making it difficult to accurately characterize vapor bubble formation and growth. In this review, we explore the influence of cavitation flow on spray formation and examine the effects of geometric and operational factors. We discuss the experimental techniques used to generate a cavitation map and the

Conference Paper

(1) Presented paper entitled “Investigation of spray behaviour simulation to predict spray tip penetration under ultra-high injection pressure” in the International conference in Fluid, Thermal and Energy System (ICFTES’22) at NIT-Calicut on 9th June 2022.

Proceedings of the 1st International Conference in Fluid Thermal and Energy Systems
June 9-11, 2022, NIT Calicut, Kerala, India
ICFTES2022–FF–062

Investigation of spray behaviour simulation to predict spray tip penetration under ultra-high injection pressure

Mehul Bambhania[1*], Nikul K Patel[2]

[1,2]Department of Mechanical Engineering, Faculty of Technology & Engineering, The M S University of Baroda, Vadodara, Gujarat, India

(2) Presented paper entitled, “Experimental study for characterization of cavitation phenomenon in the optical nozzle” in International Conference on Recent Advances in (Applied) Sciences & Engineering (RAISE)12-13April,2023 organized by Faculty of Technology & Engineering, The Maharaja Sayajirao University of Baroda, (ISBN: 978-81-962938-1-9, pp 1-8)

International Conference on Recent Advances in (Applied) Sciences &Engineering (Raise) 12-13April,2023
organized by Faculty Of Technology & Engineering, The Maharaja Sayajirao University Of Baroda

Experimental study for characterization of cavitation phenomenon in the optical nozzle

Mehul P.Bambhania[1], Sandeep Singh[2] and Nikul K.Patel[3]
[1]*PhD Research Scholar, Department of Mechanical Engineering, Faculty of Technology & Engineering, The Maharaja Sayajirao University of Baroda, India – 390002;*
Email: mehul.bambhania-med@msubaroda.ac.in
[2]*PG Student, Department of Mechanical Mechanical, Faculty of Technology & Engineering, The Maharaja Sayajirao University of Baroda, India – 390002;*
Email: rsandeepsingh918@gmail.com
[3]*Assistant Professor, Department of Mechanical Engineering, Faculty of Technology & Engineering, The Maharaja Sayajirao University of Baroda, India – 390002;*
E-mail: n.k.patel-med@msubaroda.ac.in

Book Chapter

Entitled “Investigation of Spray Behaviour Simulation to Predict Spray Tip Penetration Under Ultra-High Injection Pressure”, Chapter-20, ISBN 978-981-99-5989-1, Lecture Notes in Mechanical Engineering, Springer Nature., Pages 215-227 DOI: https://doi.org/10.1007/978-981-99-5990-7_20

International Conference in Fluid, Thermal and Energy Systems
↳ ICFTES 2022: **Proceedings of the 1st International Conference on Fluid, Thermal and Energy Systems** pp 215–227 | Cite as

Home › Proceedings of the 1st International Conference on Fluid, Thermal and Energy Systems › Conference paper

Investigation of Spray Behaviour Simulation to Predict Spray Tip Penetration Under Ultra-High Injection Pressure

Mehul P. Bambhania & Nikul K. Patel

Conference paper | First Online: 18 January 2024

50 Accesses

Part of the Lecture Notes in Mechanical Engineering book series (LNME)

Access via your institution →

Chapter EUR 29.95
Price includes VAT (India)

- Available as PDF
- Read on any device
- Instant download
- Own it forever

Buy Chapter