Numerical Simulation for Spray Characteristics of Diesel and Biodiesel fuel

M.Dyanesh[1], Nishanth Paul.V[2], Girish Ethiraj[3], P Raghu[4]

[1,2,3,4] Department of Mechanical Engineering, Sri Venkateswara College of Engineering, Sriperumbudur – 602117, Tamilnadu, India

---

ABSTRACT: *The main purpose of this study is to enhance the spray of internal combustion engines using dynamic mesh refinement. The first part of the this study uses advanced spray models with a dynamic mesh refinement plot to simulate the atomization of diesel and bio-diesel sprays. Earlier diesel sprays and bio-diesel sprays were simulated using distinct models due to the difference in their properties. Real time Engine simulations with bio-diesel fuel injection were conducted in a vertical valve engine geometry using ANSYS FLUENT 19.0. The injected fuel spray characteristics such as Sauter mean diameter (SMD), spray cone angle (**θ**), spray tip penetration (l) and fuel/air mixture were studied under the presence of in-cylinder flow. The current spray model with dynamic mesh refinement algorithm is exhibited to forecast the spray structure and liquid penetration accurately with sensible computational cost, which can then later be utilized to promote the current computational fluid dynamic models used to model an engine's combustor. This will be more economical as well as time-saving, in the design and development stages. In recent era diesel engine has been greatly involved in terms of higher efficiency and pollutant emissions. Due to uncompromising international regulation and emission restrictions, study on spray simulation will assist in enhancing the fuel injection by examining spray characteristics. Spray characteristics simulation is simulated by varying fuel Injection Pressure for diesel as well as biodiesel.*

Key words: ANSYS, SMD, Spray coneangle

1. Introduction

Due to the expanding significance of environment emissions standards and fuel economy, vehicle research and developers are demanded to refine and boost the combustion procedure. In spite the quantitative uncertainties of simulation through numerical way, modelling of fuel spray and combustion procedure has significant benefit that make its application in on-going engine research and development inevitable. Simulation through Numerical methods can possibly provide comprehensive data about the tedious in-cylinder process. However, specific models is necessary.

The synergy of 300 MPa injection pressure and 0.08 mm nozzle-hole diameter apparently gave the leading performance in words of turbulent mixture rate and droplet size reduction to lessen the mixture process and lean mixture formation [1]. The version is also integrated with sub-models including, spray, droplet collision, wall filmed combustion model along with the species transport and finite rate chemistry. The bowl-in-piston combustion geometry was utilized for construction of model. In this work RNG k- ε model is applied to confine in-cylinder turbulence [2].

Investigations through numerical methods are conducted for biodiesel spray under transient engine state. The spray tip penetration of biodiesel has been analysed in a contrast with diesel fuel in a diesel engine under transient engine state. The projected results are compared with data available experimentally with a chosen case for highly ambient pressure. The study uses the Eulerian-Eulerian approach for the two phase flow simulation [3].

The effect of combustion is found to change the shape and structure of the central recirculation zone to be more compact in length but larger in diameter in the transverse direction. In-addition the results show that the gas phase radiation alters the spray dynamics by changing the local gas-phase temperature distribution. This affects the spray evaporation rate, the local mixture fraction, and accordingly the combustion heat released rate and the forecast emissions. The simulation with no radiation modelling shows over prediction in the temperature distribution, pollutants emissions, higher fuel evaporation rate, and narrower range of droplet size distribution with lower number density for the smaller size particles. Only Large-Size particles are considered and not the small-size droplets in the experiment as the flow exhibits high unsteadiness in small-size. The high unsteadiness of the shear layer is confirmed by root mean square(rms) values at the first three locations, where the rms value is around 100% of the corresponding mean value [4].

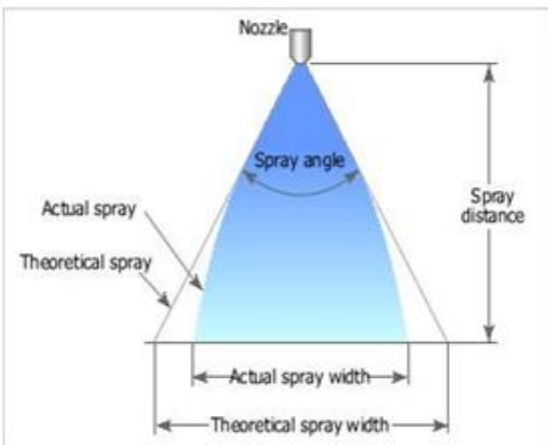

Fig: 1 Fuel spray terminology

This to improve the predictive capabilities of spray breakup and combustion models. The initial aim is to progress a unified spray model to be used with dynamic mesh refinement to simulate spray atomization in diesel and bio-diesel engines. The model will be confirmed by experimental data of both diesel and bio-diesel sprays. Fig1. shows the Fuel spray terminology.

Spray model to be used with dynamic mesh refinement to simulate spray atomization in diesel and bio-diesel engines. The model will be confirmed by experimental data of both diesel and bio-diesel sprays. Fig1. shows the Fuel spray terminology.

2. Numerical simulation

Physical fluid flow problem can be solved both experimentally and numerically. The Numerical simulation is more worthy for parametric studies and it also gives precise results by decoding governing equations in each and every cell of the fluid domain. In the recent years there has been huge development in the field of numerical technique, which has made a high effect on the evaluation of complex flow problems and achieving their solution.

The test cases examined in the present investigation are free atmospheric spray, impinging flat wall diesel spray and engine spray, the numerical simulation is carried out by using commercial accessible CFD software called ANSYS 19.0. The work is focused only on the spray that is injected from the nozzle having different characteristics like size, velocity, density etc.

Model description

A general physical description of the sub model used for the simulation of the gas and liquid phases of the fuel spray is as follows.

2.1 Gas Phase

The simulation through numerical methods for flow and mixture formation is based on an Eulerian description of the gas phase and on a Langrangian description of the droplet phase. The interaction between the both phases is described by source terms for momentum, heat and mass exchange.

4. MESH

This methodology has been widely used for the spray modelling and is also implemented in the CFD-code ANSYS.

The turbulent gas flow is described by a numerical simulation of the complete ensemble averaged equations of the conservation of mass, momentum, energy and species mass fraction in an unstructured numerical mesh. Turbulence is modelled using a standard k-ε. model.

2.2 Dispersed phase

The droplets are considered as disperse phase. The motion of the dispersed phase will be influenced by that of the continuous one and interphase, momentum, mass and heat transfer effects. The strength of the interaction will depend on the dispersed particle's size, density and number density.

When the dispersed phase is volatile soluble or reactive, mass transfer occurs between the phases. This is accompanied by interphase heat transfer, which may also arise due to the interphase temperature difference. Interphase mass transfer cause size changes in the dispersed element.

The size change may also be produced by fluid-dynamics forces acting on the dispersed elements causing them to break up into smaller elements. Inter element collisions processes may also produce the opposite effects (i.e) size increase due to coalescence or agglomeration.

3. NUMERICAL MESH

The initial crucial task in the process of simulation is creating a computational mesh to represent the flow domain geometry. The mesh generalise shown in Table: 1

Table: 1 Mesh dimension

Bore (mm)	20*20
Stroke (mm)	100
Capacity (cc)	40
Injection pressure (bar)	180,200,220
Mesh	Fine Mesh

Numerical mesh is shown in fig 2and3. The Experimental condition way shown in Table: 2

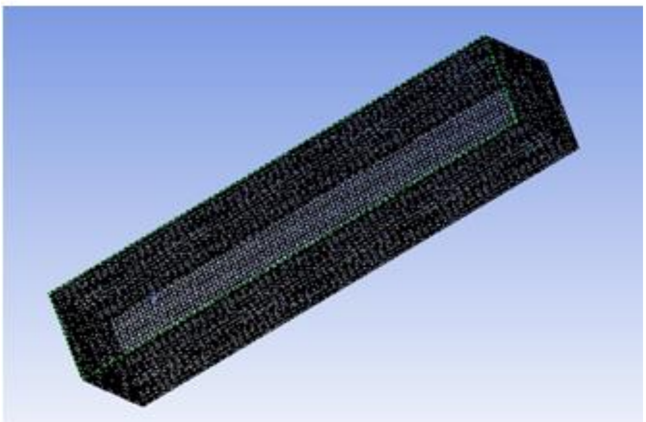

Fig: 2 Mesh side view

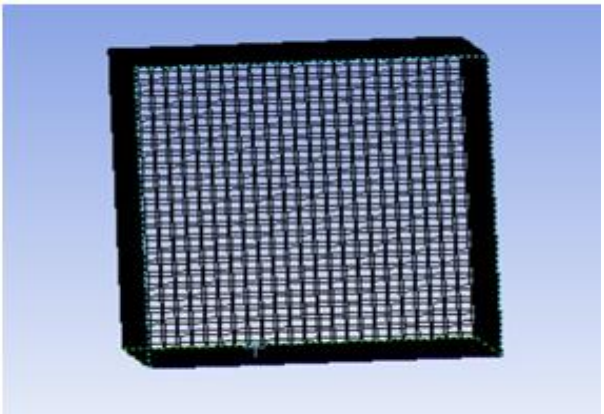

Fig: 3 Top view of Mesh

5. RESULTS AND DISCUSSION

The purpose of the current study is to investigate the spray characteristics, fuel droplet atomization behaviour and non-dimensional behaviour of the test diesel and biodiesel blends by varying injection pressure (180, 200, 220 bar) IN ANSYS 19.0 (FLUENT). Fuels with higher density show inferior spray and atomization characteristics. When fuel injection starts, droplets move fast and tend to break into smaller droplets.

The droplets size distribution depends on the density of the fuel to a high extent. The atomization phenomenon occurs in millisecond after the point of injection. Fuels having higher density lead to higher spray penetration and poor atomization because of their higher inter-molecular forces. This leads to formation of larger-fuel droplets, which has relatively higher inertia and therefore they travel longer distance in the spray chamber. Higher spray tip penetration and poor atomization behaviour of fuels may cause inefficient fuel-air mixing and may consequently lead to formation of higher soot in the engine.

Figure 4,5,6 shows the comparison between the experimental value and Numerical value for spray length, spray cone angle and SMD. It is observed that Experimental values are very close to the simulated value. The percentage error in Diesel for Spray length (9%), cone angle (5.6%), and SMD (2.64%) are respectively. The percentage error in KOME for Spray length (1.29%), cone angle (1.21%), and SMD (1.45%) respectively. The percentage error in KB20 for Spray length (1.14%), cone angle (1.05%), and SMD (1.13%) are respectively. By doing the Numerical simulation the Experimental work was reduced.

Experimental condition Table 2

Chamber Temperature	180ºC
Fuel temperature	50ºC
Chamber Pressure	60 bar

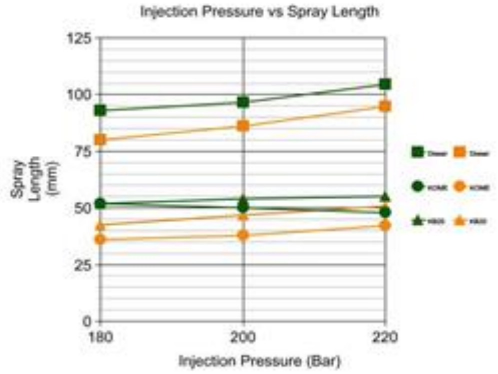

Fig: 4 comparison of Experimental and Simulation value Spray length

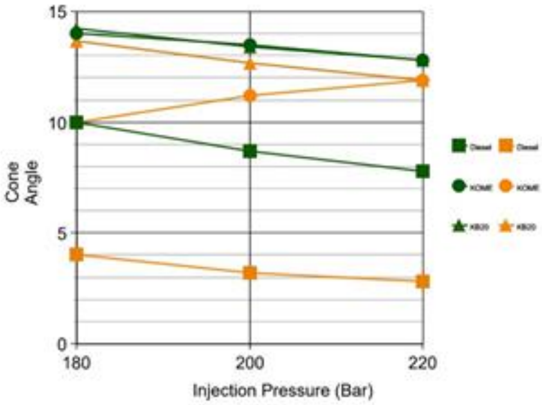

Fig: 5 Comparison of Experimental and Simulation value of Spray cone angle.

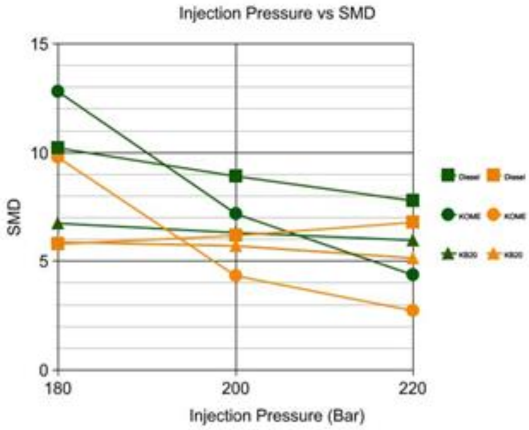

Fig: 6 Comparison of Experimental and simulation value of SMD

6. CONCLUSION

Simulation of spray characteristics has been simulated with three different pressures of 180bar, 200bar, 220bar to derive the spray length, spray cone angle, Sauter mean diameter. Values for KB20 gives less percentage compared to diesel and KOME.

Modelling of Spray characteristics using 180 bar pressure at constant fuel temperature gives 93mm of spray length with 4° cone angle along with 5.73e-3mm of sauter mean diameter. Spray characteristics for 200 bar pressure at constant fuel temperature was observed as 98mm of spray length with 3.2° wide angle and also has an 6.50e-10 of sauter mean diameter. With 220 bar pressure of injection Spray angle was observed as 105mm with cone angle of 2.8° and sauter mean diameter of 6.63e-10.

7. ACKNOWLEDGEMENT

We hereby thank our Institution Sri Vennkateswara College of Engineering, Head of Department ,Mechanical and the Principal.

8. REFERENCES

1) Raghu, P., et al. "Experimental study of mixture formation in biodiesel spray with preheated fuel." International Journal of Applied Engineering Research 10.19: 2015.

2) Nishida, Keiya, et al. "Effects of micro-hole nozzle and ultra-high injection pressure on air entrainment, liquid penetration, flame lift-off and soot formation of diesel spray flame." International Journal of Engine Research 18.1-2 (2017): 51-65.

3) Majhool, Ahmed Abed Al-Kadhem, and Abbas Alwi Sakhir AL Jeebori. "Study Of Modelling Spray Penetration Of Biodisel Fuel Under Transient Engine Conditions." Academic Research International 3.3 (2012): 70.

4) El-Asrag, Hossam A., Anthony C. Iannetti, and Sourabh V. Apte. "Large eddy simulations for radiation-spray coupling for a lean direct injector combustor." Combustion and Flame 161.2 (2014): 510-524.

5) Pei, Yuanjiang, et al. "Large eddy simulation of a reacting spray flame with multiple realizations under compression ignition engine conditions." Combustion and Flame 162.12 (2015): 4442-4455.

6) Kongre, Umakant V., and Vivek K. Sunnapwar. "CFD Modelling and Experimental Validation of Combustion in Direct Ignition Engine Fuelled with Diesel." International Journal of Applied Engineering Research 1.3 (2010): 508.

7) Kolhe, Ajay V., Rajesh E. Shelke, and S. S. Khandare. "Combustion Modeling with CFD in Direct Injection CI Engine Fuelled with Biodiesel." Jordan Journal of Mechanical and Industrial Engineering 9.1 (2015).