Spray Characteristics of Diesel Fuel Using Numerical Simulation

P. Vengatesh[1], P. Raghu[2], N. Nallusamy[3]

[1,2] Department of Mechanical Engineering, Sri Venkateswara College of Engineering, Sriperumbudur – 602117, Tamilnadu, India
[3]Department of Mechanical Engineering SSN College of Engineering, OMR, Kalavakkam – 603110, Tamilnadu, India

---***---

Abstract - *The primary objective of this study is to improve the spray of internal combustion engines using dynamic mesh refinement. The first part of the study used advanced spray models with a dynamic mesh refinement scheme to simulate atomization of diesel and bio-diesel sprays. Traditionally diesel sprays and bio-diesel sprays were simulated using different models due to their different characteristics. Engine simulations with diesel fuel spray were conducted in a vertical valve engine geometry using ANSYS FLUENT 15.0. The fuel spray characteristics such as Sauter mean diameter, spray cone angle, spray tip penetration and fuel/air mixture were studied under the presence of in-cylinder flow. The present spray model with dynamic mesh refinement algorithm was shown to predict the spray structure and liquid penetration accurately with reasonable computational cost. This can then be used to advance the current computational fluid dynamic models used to model an engine's combustor. This will save the industry time and money, in the design and development stages. In recent years diesel engine has been greatly involved in terms of higher efficiency and pollutant emissions. Due to stringent international regulation and emission restrictions researches on spray simulation will help in developing better fuel injection by analyzing spray characteristics. Spray simulation is simulated by varying fuel Injection Pressure for diesel and bio diesel.*

Key Words: **ANSYS, SMD, Spray cone angle, Eulerian, Mesh, RNG k- ε model.**

1. INTRODUCTION

Due to the growing importance of fuel economy and future emissions restrictions, engine manufactures are continuously forced to improve the combustion process. Despite the quantitative uncertainties of numerical simulation, modeling of fuel spray and combustion processes has significant advantages that make its utilization in current engine development a necessity. Numerical simulation can potentially provide detailed information about the complex in-cylinder process. However, accurate models are required.

The combination of 300 MPa injection pressure and 0.08 mm nozzle-hole diameter reportedly gave the best performance in terms of turbulent mixture rate and droplet size reduction to decrease the mixture process and lean mixture formation [1]. The model also integrated with sub-models includes, spray, droplet collision, wall filmed combustion model with species transport and finite rate chemistry. The bowl-in-piston combustion geometry was used for model construction. In this study RNG k- ε model is implemented to confine in-cylinder turbulence [2].

Numerical investigations are conducted for biodiesel spray under transient engine conditions. The spray tip penetration of biodiesel has been studied in a comparison with diesel fuel in a diesel engine under transient engine conditions. The predicted results are compared with experimental data 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 impacts the spray evaporation rate, the local mixture fraction, and consequently the combustion heat released rate and the predicted emissions. The simulation with no radiation modeling 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. The small-size droplets are not considered in the experiment and where the flow exhibits high unsteadiness. The high rms values at the first three locations confirm the high unsteadiness of the shear layer, where the rms value is around 100% of the corresponding mean value [4].

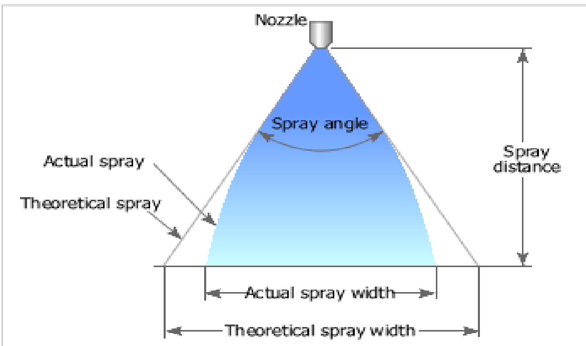

Fig: 1 fuel spray terminology

This to improve the predictive capabilities of spray breakup and combustion models. The first objective is to develop 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 validated by experimental data of both diesel and bio-diesel sprays. Fig1. Show the Fuel Spray Terminology.

2. NUMERICAL SIMULATION

Any physical fluid flow problem can be solved either experimentally or numerically. The Numerical simulation is more suitable for parametric studies and it also gives accurate results by solving governing equations in each and every cell of the fluid domain. In the recent years there were tremendous improvement in the field of numerical technique, which made a great impact 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 available CFD software named ANSYS. The work is focused only on the spray that is injected from the nozzle having different characteristics like size, velocity, density etc.

For this spray analysis the evaporative condition are chosen for free atmospheric spray and flat wall impinging spray and engine spray.

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 numerical simulation of 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. This methodology has been widely used for the spray modeling 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 modeled 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 first important task in the process of simulation is creation of a computational mesh to represent the flow domain geometry. The mesh generalize shown in Table: 1

Table: 1 Mesh dimension

Bore (mm)	80
Stroke (mm)	110
Capacity (cc)	552.92
Injection pressure (bar)	180,200,220
Mesh	Fine Mesh

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

4. MESH

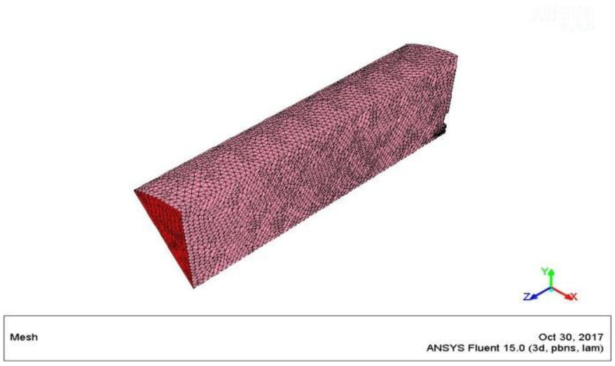

Fig: 2 Mesh only 30° section are model of 360° section

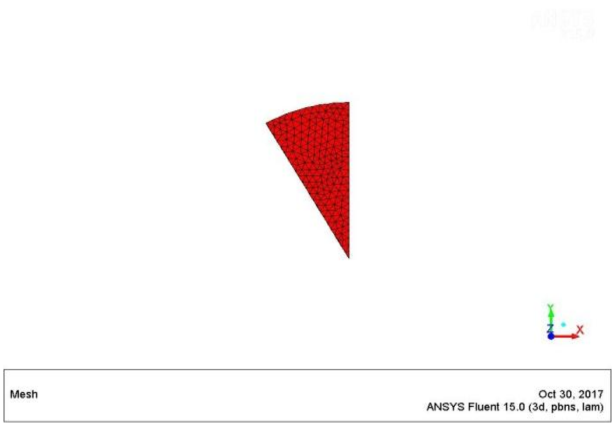

Fig: 3 Top view of Mesh

5. RESULTS AND DISCUSSION

The intention of the present study is to investigate the spray characteristics, fuel droplet atomization behavior and non-dimensional behavior of the test diesel by varying injection pressure (180, 200, 220 bar) IN ANSYS (FLUENT). Fuels with higher density exhibit 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 large extent. The atomization phenomenon happens in millisecond after the start of injection. Fuels with 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 have relatively higher inertia and therefore they travel longer distance in the spray chamber. Higher spray tip penetration and poor atomization behavior 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 for Spray length (9%), cone angle (5.6%), and SMD (2.64%) 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

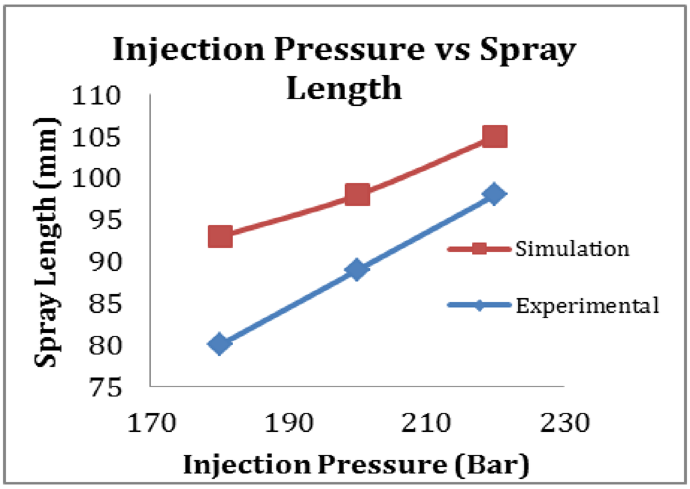

Fig: 4 comparison of Experimental and Simulation value Spray length

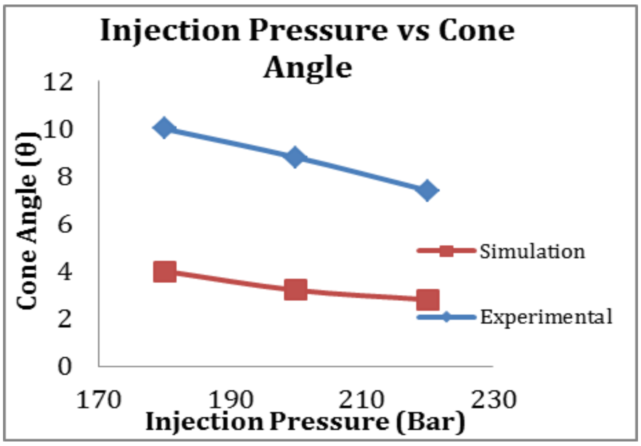

Fig: 5 Comparison of Experimental and Simulation value of Spray cone angle.

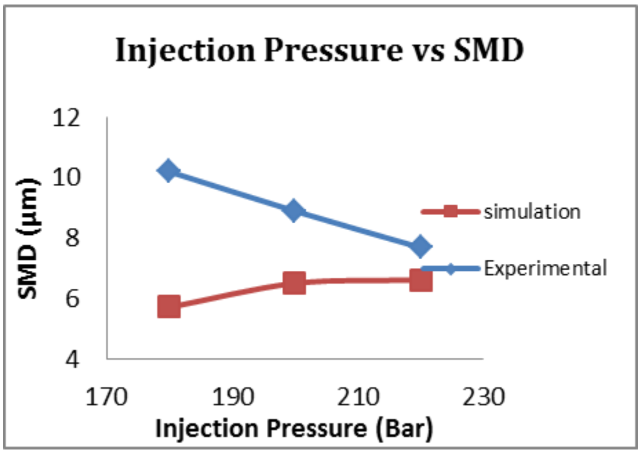

Fig: 6 Comparison of Experimental and simulation value of SMD

5. CONCLUSIONS

Simulation of spray characteristics has been simulated with three various pressures of 180bar, 200bar, 220bar to obtain the spray length, spray cone angle, Sauter mean diameter.

1. 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.

2. 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.

3. 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.

ACKNOWLEDGEMENT

Author thanks the management of Sri venkateswara college of engineering for providing the experimental setup to do this work.

REFERENCES

[1] 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.

[2] 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.

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

[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).