21st Australasian Fluid Mechanics Conference
Adelaide, Australia
10-13 December 2018

Modelling Primary Atomization and its effects on spray Characteristics under Heavy Duty Diesel Engine Condition

Muteeb ul haq[1], Rizwan Latif[2] , Imran Shafi[3] and Ali Javaid[4]

[1,3,4] Department of Mechanical Engineering
National University of Science and Technology, Islamabad 44000, Pakistan

[2]Department of Mechanical Engineering
University of Engineering and Technology, Taxila 47070, Pakistan

Abstract

Diesel engines are most widely used all over the world because of their efficiency, reliability and adaptability. In heavy duty engines we are more interested in the torque produced rather than the speed of an engine, such engines that are being used in the industries have low efficiency and high soot emissions due to inefficient combustion unlike high speed diesel engines that have wide commercial use. Fuel spray atomization is the key process that defines combustion efficiency, which further affects fuel economy and soot emissions. This research is focused on modelling primary atomization under several injection conditions and its effects on spray characteristics and fuel breakup under Heavy duty diesel engine conditions. The computational fluid dynamics (CFD) tools available in Ansys in which basic conservation equations of mass, momentum and energy equations are used. Fluid breakup is modelled using the new VOF-DPM Hybrid approach along with the KH breakup model. Turbulence is incorporated using Realizable kε model. Spray characteristics including penetration length, souter mean diameter (SMD),drop size distribution and drop velocity were studied , all these properties affects engine efficiency ,results showed their strong dependence on injection parameters and the back pressure. The new Volume of fluid-Discrete phase model (VOF-DPM) hybrid approach proved much more effective in capturing the spray characteristics and fuel breakup.

Introduction

Diesel fuel based internal combustion engine are widely used for industrial, commercial, marine and defence applications all over the world. Diesel engines that were invented decades ago are bound to face issues regarding dynamics, metallurgy, combustion, heat transfer and lubrication etc. These engine has able to attract the attention at global level in terms of economical operation, case of maintenance, relatively long life and variety of applications [1].Diesel Engines are extensively used in industries as they offer high efficiency and have longer life than Otto engines and also the diesel fuel is cheaper than the gasoline, due to these advantages production of diesel engines has increased in past few decades. Manufacturers of diesel engine are facing many challenges today, one of the major is to reduce soot and NO_x generated in result of burning the fuel. These gases are injurious to humans and animals [2,3],due to which concerned authorities all around the world are imposing many restrictions to the manufacturers to minimize the soots of their vehicles and to make them environment friendly as well, P.Kaesong [3].
Performance of a diesel engine is dependent upon 6 basic characteristics, which are fuel injection, fuel spray, combustion, ecology characteristics and fuel economy. The most critical of above is the fuel spray characteristics which has a direct effect on, fuel atomization, fuel spray development, fuel-air mixture formation and combustion and these parameters further effect the engine performance and fuel economy [4]. The spray dynamics is a very complex process, which includes fuel atomization, breakup of liquid in to droplets, evaporation, collision, coalescence, moreover micro level time and length scales involved in the process, makes it very difficult in controlling accuracy and the relevant boundary conditions. Atomization of fuel and breakup of droplets are the most critical of all physical processes among these [2].This research is focused on studying the effect of injection and back pressure on penetration length, souter mean diameter (SMD) and drop velocity using VOF-DPM hybrid approach along with Wave breakup model.

Primary Atomisation

As the fuel exits the nozzle a competition is induced between the cohesive and disrupting forces acting on the liquid surface, which gives rise to oscillations and agitations. These oscillations are amplified under suitable conditions and the intact liquid breaks up into drops and ligaments. This process is known as primary atomization as shown in Figure 1.These drops undergo secondary atomization if their size is greater than the critical size, this further decreases the droplets size and make them favourable to evaporate. Fuel spray atomization and the breakup process are critical factors that governs the formation of fuel air mixture and thus controls the combustion efficiency and soot emissions. Atomization depends upon various factors such as internal flow characteristics, the jet velocity profile, cavitation, turbulence, thermodynamic states of an engine cylinder and the nozzle geometry. Efficient fuel atomization increases the contact area between fuel-air and also improves the mixture formation in an engine cylinder, which in turn increases the fuel economy, moreover a better atomization quality yields smaller droplet size, which reduces soot emissions, thus a fuel atomization process must be studied in detail for optimizing the engine performance.

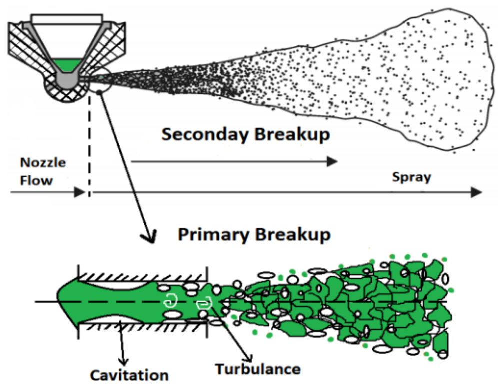


Figure 1: Schematic of full spray structure and primary breakup [5]

Spray Characteristics

It is essential to study the spray characteristics to access the quality of spray, there are two types of spray characteristics macroscopic and microscopic. Breakup length, penetration length and spray cone angle are macroscopic features while the drop diameter, drop velocity and size distribution are microscopic features. These parameters are studied with respect to various operating conditions of the diesel engine.

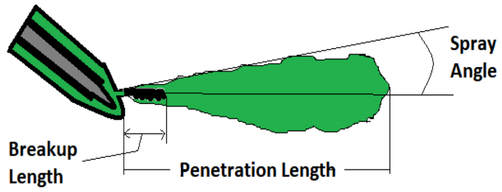


Figure 2: Schematic of various spray parameters

Wave Model

Wave model also known as Kelvin-Helmholtz (KH) model introduces KH instabilities due to aerodynamic forces on the liquid surface [6, 7] .The model assumes that these instabilities grows on liquid surface that causes the droplets to shear off from the liquid surface. Figure 3 shows the schematic view of the wave model.

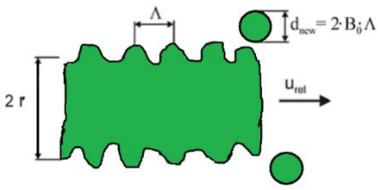


Figure 3: Illustration of Wave model

Rate of decrease for parent droplet and the resulting new child droplet size are related to wavelength (Λ_{KH}) and frequency (Ω_{KH}) of the wave with fastest growth rate that grows on the liquid surface.

$$\Omega_{KH}(\text{growth rate}) = \frac{0.34 + 0.38 \mathrm{Weg}^{1.5}}{(1+Z)(1+1.4T^{0.6})}\sqrt{\frac{\sigma}{\rho_l r^3}} \quad (1)$$

$$\Lambda_{KH}\ (\text{Max Wavelength}) = \frac{9.02 r (1 + 0.45\sqrt{Z})(1 + 1.4T^{0.7})}{(1 + 0.865 \mathrm{We}_g^{1.67})^{0.6}} \quad (2)$$

Where parent drop radius(r), fuel density (ρ_l), fuel surface tension (σ) and the air Weber number (Weg), Ohnesorge number (Z) and Taylor number (T) are dimensionless parameters.

$$We_l = \frac{\rho_l U_r{}^2 D}{\sigma}, \mathrm{Re} = \frac{\rho_l U_r d}{\mu_l}, \mathrm{Oh} = \frac{\sqrt{We_l}}{\mathrm{Re}}, T = Z\sqrt{We_g}$$

Values of these constants depends upon jet velocity, fuel properties and nozzle geometry.

THE VOF-DPM HYBRID APPROACH

It is an Euler-Lagrangian approach in which the continues liquid phase is treated as a continuum where the Naiver-Stokes and other governing equations are solved, while the droplets (dispersed phase) are solved by tracking several number of droplets through the fluid domain its accuracy depends upon how well the interface is captured by VOF model and also on the accuracy of the droplet identification algorithm. The Algo identifies and isolate liquid structure on the basis of their Size range by calculating the volume equivalent sphere diameter of the lump and on the basis of sphericity, which is the measure of how close is the shape of the drop to a perfect sphere. The droplet identification algorithm is available in Ansys Fluent which caters the transfer of a lump from Eularian to Langrangian (DPM) phase .Similar approach has been used by [8,9] for spray simulations.

Governing Equations

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \bar{u} = 0 \quad (3)$$

Reynolds Average Navier Stokes Equation:

$$\frac{\partial u_i}{\partial t} + \mathrm{div}(u_i \mathbf{U}) = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \mathrm{div}(\mu\ (\mathrm{grad}\ u_i)) + \frac{1}{\rho}\left[\frac{\partial(-\rho \overline{u_i' u_j'})}{\partial x_j}\right] \quad (4)$$

Realizable k ε Equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k \mathbf{u_j})}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon - Y_M + S_k \quad (5)$$

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k \mathbf{u_j})}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial(k)}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + (\varepsilon\nu)^{0.5}} + S_\varepsilon \quad (6)$$

G_k is the generation of turbulence kinetic energy, C_1 and C_2 are constants,σ_k and σ_ε are the turbulent Prandtl numbers for k and ε ,S_k and S_ε are source terms.

Simulation Settings

A cylindrical meshed body with 40 mm diameter and 80 mm length is shown in Figure 4.

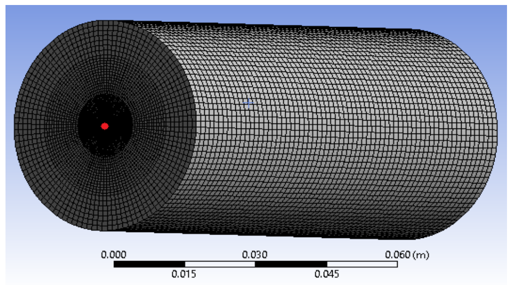


Figure 4: Numerical grid for Spray Simulation

Structured meshing with hexahedral cells is used, mesh becomes finer as we move from periphery to the centre of the cylinder. The total number of cells are 1327410 with average cell volume of 0.096 mm^3.The injection take place from the centre of the left face marked by red dot, only one jet of the spray is simulated which is enough to study the spray characteristics and also simplifies the modelling process.

Parameter	Quantity
Injection Pressure	100,200,300bar
Injection Temperature	300K
Back pressure	1&2 bar
Nozzle diameter	0.290 mm
Nozzle Length	1.4 mm
Mass flow rate	0.003g/s
Injection Duration	3ms
Chamber length	80mm

Chamber diameter	40 mm
Chamber density	1.225 & 2.3 kg/m^3
Chamber Temperature	300K
Diesel density	$830kg/m^3$

Table1: Settings for Spray Simulations

Results

Spray characteristics are observed for different Engine operating conditions using Ansys Fluent .Hybrid VOF-DPM approach is used to model primary breakup along with the KH breakup model to include the aerodynamics effect and realizable kε model for turbulence effects, results shows that the new VOF-DPM hybrid approach is very effective in capturing the breakup of the liquid core as compared to DPM model and the spray properties are in accordance to the results obtained by Jing[10].

Penetration Length

Penetration length is studied under three different injection pressures at different ambient conditions. Spray penetration length increases gradually with the increase in injection pressure from 100 to 300 bar as expected, because greater pressure difference yields higher velocities and momentum at the nozzle exit that helps the drops to penetrate further and have more contact area with the surrounding air however penetration should be such that it do not cause wall impingent, which leads to incomplete combustion and reduce engine efficiency. Increase of back pressure has slight effect on penetration length at the initial stages, as the spray is very dense, as the spray penetrates further aerodynamic and drag force becomes significant ambient conditions starts affecting the spray properties, spray is much diluted, and the contact probability with surrounding air increases. Higher back pressure yields higher surrounding density in the chamber that enhances drag force and causes the reduction in penetration length after the initial stage.

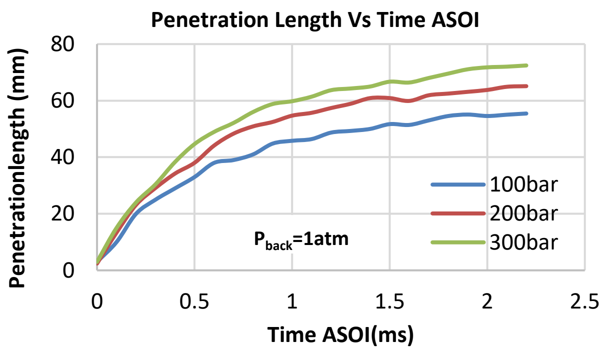


Figure 5: Comparison of penetration length at different injection pressures w.r.t Time after start of injection (ASOI) (atmospheric back pressure).

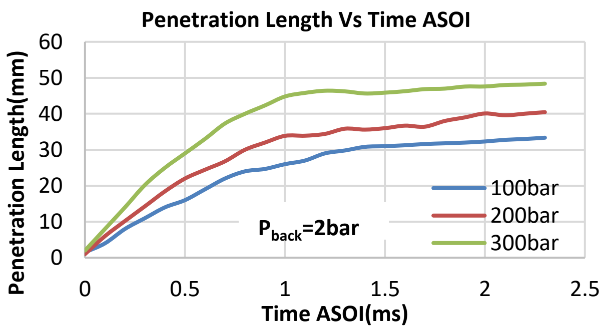


Figure 6: Comparison of penetration length at various injection pressures w.r.t Time ASOI (back pressure 2bar).

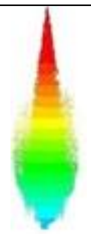
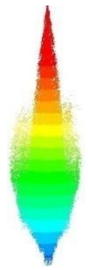
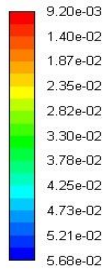
Penetration Length (m) w.r.t Time ASOI (ms)			
0.5ms	**1.5ms**	**2.5ms**	**P Length (m)**
			

Table 2: Comparison of penetration length at different time steps (injection pressure 300bar, back pressure 2bar)

Souter Mean Diameter (SMD)

It is the ratio of the drop volume to its surface area [11], Increasing the injection pressure decreases SMD gradually, and narrows the size difference between the droplets which leads to uniform size distribution, greater number of droplets are generated at higher injection pressure so the greater surface area of the fuel is produced that yields smaller SMD, which enhances the atomization and evaporation process that increases combustion efficiency. The decrease in SMD is much more prominent at the start, because the effect of drag on smaller droplets is much more significant as compared to large drops.

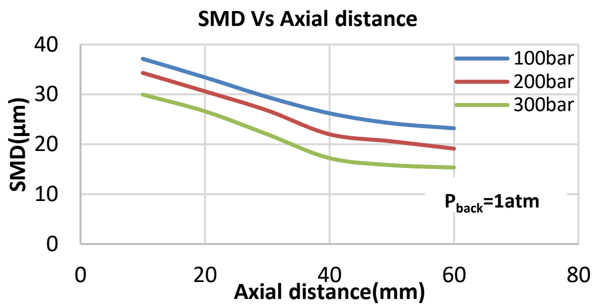


Figure 7: Comparison of SMD at various Injection Pressures w.r.t axial distance (back pressure 1atm)

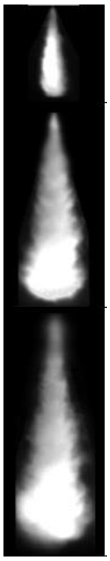
Time ASOI	Jing Experimet	Drop size distribution	Drop size (mm)
0.2ms			
0.4ms			
0.5ms			

Table 3: Comparison between simulated drop size distributions with Jing Experiment [10].

Velocity

At higher injection pressure of 200bar, velocity increases slightly as it enters the chamber because of higher pressure difference however this increase was not observed at the injection pressure of 100 bar and also the velocities for 200 bar are greater than that of 100 bar at the same axial positions, velocity decreases rapidly at the start and then an increase is observed and after this a gradual decrease till the end, This is because the droplets at the start of spray experience much more drag force from the surrounding air, the droplets coming after the initial drops are less affected by the surrounding atmosphere which results in higher velocities. Deceleration rate at the start is much higher as compared to downstream because the spray is much dense at the start which causes more collisions, and also greater drag is induced at higher velocities.

Velocity(m/s) w.r.t Time ASOI (ms)			
0.5ms	1.5ms	2.5ms	Velocity m/s

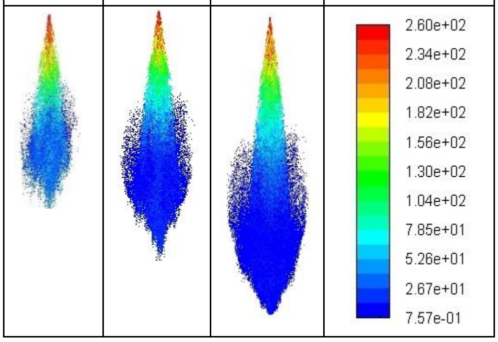


Table 4: Comparison of axial velocity at different Time ASOI (injection pressure 300bar, back pressure 2bar)

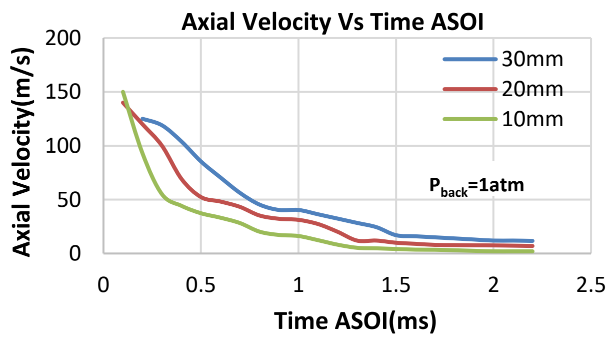


Figure 8: Axial velocity at different sampling position (injection pressure 100 bar & atmospheric back pressure)

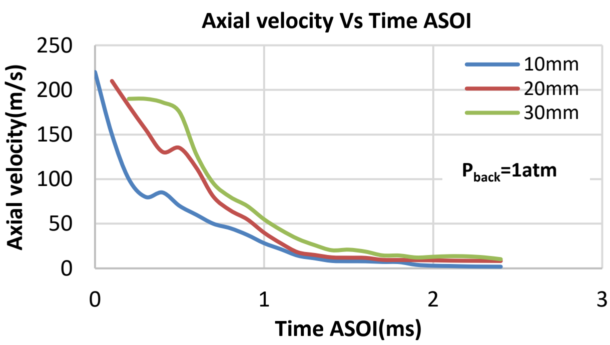


Figure 9: Axial velocity at different sampling position (injection pressure 200 bar & atmospheric back pressure)

Conclusion

Analysis of spray characteristics revealed that, penetration length of the fuel is dependent on initial injection pressure and the back pressure in an engine cylinder, higher injection velocities are induced because of higher injection pressures, however at higher back pressure deceleration rate is higher and the fuel penetration is reduced. Increasing injection pressure decreases SMD that in turn favours early evaporation and enhance combustion efficiency. Decrease of velocity at the start of injection is much higher due to the higher drag experienced by the initial drops. Drop size decreases drastically from the start of injection, with only few drops comparable to the size of nozzle diameter. Mean drop size is about 40µm that makes it favourable to evaporate. VOF-DPM hybrid approach is very effective in capturing the fuel breakup and spray characteristics.

Acknowledgments

Many thanks to my supervisors for continues guidance and to my fellow colleagues Mr.Rizwan Latif and Mr.Mohsin Raza for many use full discussions that helped me to grasp the key concepts regarding spray dynamics and also kept me motivated.

References

[1] Fabian Peng Karrholm, Numerical Modelling of Diesel Spray Injection, Turbulence Interaction and Combustion, Department of Applied Mechanics Chalmers University of Technology Goteborg, Sweden, 2008.

[2] Yuan,W.Sauer,J.and G.H.,S.“Modeling and Computation of Unsteady Cavitation Flows in Injection Nozzles”. Mechanique & Industries, 2, pp. 383–394. 2001.

[3] Masuda, R., Fuyuto, T., Nagaoka, M., Berg, E., and Tatschl, R., 2005.Validation of diesel fuel spray and mixture formation from nozzle internal flow calculation”. SAE Paper 2005-01-2098.

[4] Merker, G.P, Schwarz, C., Stiesch, G, Otto, F. (2005). Simulating combustion. Springer.

[5] Baumgartner, C., Mixture formation in internal combustion engines. Springer, 2006.

[6] Reitz, R.D., Atomization and other breakup regimes of a liquid jet. PhD thesis, 1978.

[7] Reitz, R. and F. Bracco, Mechanism of atomization of a liquid jet. Physics of Fluids, p.1730, 1982.

[8] Rohitkumar Sonawane,Vivek Kumar,Pravin Nakod, A comparison of different mesh topology for numerical analysis of a turbulent liquid jet, ASME 2017 Gas Turbine India Conference Gtindia,2017.

[9] Li, x., Arienti, M.Soteriou, and Sussman.m, “Towards an efficient, high-fidelity methodology for liquid jet atomization computations”, 48th aiaa aerospace sci. meet ,aiaa, Orlando, Florida, USA, january 4-7 (2010-210).

[10] Daliang Jing, Experimental and Numerical Studies of Fuel Spray, The University of Birmingham School of Mechanical Engineering, March 2015.

[11] András Kadocsa,Modeling of spray formation in diesel engines, department of fluid mechanics, Budapest university of technology and economics,2007.