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

# Investigation of Secondary Breakup in DI High Torque Diesel Engine at Different Injection Pressure

**Mohsin Raza[1], Rizwan Latif[2], Ali Raza[3], Imran Shafi[4]**

[1, 3, 4]Department of Mechanical Engineering
National University of Sciences and Technology, Islamabad 44000, Pakistan
[2]Department of Mechanical Engineering
University of Engineering and Technology, Taxila 47070, Pakistan

## Abstract

The high torque and medium speed direct injection (DI) diesel engine has tremendous commercial and industrial utilizations. The ambition of fuel spray atomization process in DI diesel engine is the formation of good quality fuel-air mixture in such a way that a better combustion is attained, whilst discharge of soots and pollutant are diminished. The mode- ling technology of computational fluid dynamics (CFD) for diesel spray in DI engine is an impressive approach to study and conclude spray attributes as well as extreme reduction of experimental work during engine development process. For this objective, a precise numerical simulation of the fuel spray evaluation process is essential**.** At high injection pressure the effects of cavitation and spray turbulence shorten the liquid core length, therefore in this research the primary atomization is neglected. Considering the secondary breakup the high injection pressure systems lead to high droplet velocities, Due to high droplet velocity the catastrophic or stripping breakup regime may occur. The parent droplets disintegrate into child droplets that further breaks up to the smallest possible droplet size. At higher weber number the atomization is simulated by using the Kelvin –Helmholtz model with the Rayleigh-Taylor accelerative instabilities. The effects of turbulence are handled by $k-\varepsilon$ Realizable model. The VOF (volume of fluid) multi-phase model is used to simulate the interaction between fuel and air. The discrete phase model is applied with coupling of VOF. Finally the spray characteristics such as spray tip penetration, spread angle, droplet average mean diameter and velocity at different injection pressures are computed. The results are validated from published experimental work and found a good agreement with respect to spray characteristics at all injection conditions. Thus an obvious improvement for fuel spray modeling is observed for a heavy duty diesel engine.

## Introduction

The heavy duty DI diesel engines have many applications as automotive, marine industry, tool driving, or electrical power generation. SUV's manufacturers of the world are made to follow strict emission control policies but industries where speed has less importance and requirement of power is more, emission control policies are less likely to be followed. However this has no excuse and must come under the umbrella of world policies. The engine in question is such like and detail of that engine will be numerated in subsequent sections. The diesel engine technology used in heavy duty engines must be improved to meet the strict emissions and performance regulations. There are different ways to create spray according to the requirement and applications [1, 2]. The engine performance strongly depends upon fuel injection and specially sprays formation that leads to complete combustion inside the chamber. To control the combustion process and emission exhaust gasses a well formed mixture is needed. To meet all requirements some typical numerical methods have been developed to study the spray. These numerical methods decrease the dependence on experimental setups. When the fuel is injected inside the chamber it transforms into fine droplets that further breaks into child droplets and evaporate. The early the mechanism of breakup starts, better mixture formulation will occur and ultimately better would be the combustion and engine efficiency thereof. In conventional CFD modelling RANS equations are very common to use for numerical approximation of fluid flow. The Basic principal of RANS is to average the flow properties such as pressure, temperature, velocity and density. More convenient choice for the spray calculations is LES (Large Eddy Simulation) modelling [2]. In which large eddies were resolved and small eddies were filtered. Eulerian LES model is an effective choice for modelling the spray in primary breakup region. There is difference in both models on the basis of computational cost. LES required a very high computational power while RANS doesn't. The RANS is computationally less expensive scheme to deal with turbulent flow but with compromised accuracy.

At high pressure conditions cavitation [3] inside the nozzle causes air bubbles. Entrapping with the liquid fuel, when these bubbles come out from the nozzle, causes the liquid jet to break into ligaments, and shorten the liquid length. Thus in this case the primary breakup is negligible and spray is atomized in form of droplets. These droplets further break into fine droplets that is called the secondary breakup. To investigate the secondary breakup of diesel fuel spray in DI diesel engine cylinder different models have been used in literature. These models includes the TAB, ETAB, KHRT, and TLKH-RT [4, 5 and 6]. The TAB and ETAB model have been used for low Weber number, whereas KH-RT is normally used for high Weber number. In this research the injection velocity is high and Weber number is greater than 150, at this Weber number the sheet and striping breakup occur. The KR-RT model is being used for this breakup phenomenon for the heavy duty diesel engine.

The objective of this work is to investigate the effect of injection pressure and ambient air density at spray characteristics (penetration length, spray velocity, spray SMD and droplet diameter) for our case i.e. High Torque Engine. The obtained results are compared with available experimental and theoretical data and good agreement has been found.

## Governing Equations

### Continuous Phase

$$\frac{\partial \bar{\rho}}{\partial t}+\frac{\partial}{\partial x}(\overline{\rho u_j})=0 \tag{1}$$

The Reynolds average Navier's-Stokes equations in index notation for 3D flow in Cartesian coordinate system is given as

$$\frac{\partial}{\partial t}(\bar{\rho}\bar{u}_i)+\frac{\partial}{\partial x_j}\left(\rho\bar{u}_i\bar{u}_j\right)=-\frac{\partial \bar{p}}{\partial x_i}+\bar{\mu}\left(\frac{\partial^2 \bar{u}_i}{\partial x_j^2}\right)-\frac{\partial}{\partial x_j}\left[\rho\overline{u'_j u'_i}\right] \tag{2}$$

Where the transient, convective, pressure and diffusive term in mean form and the last term in right hand side represents the Reynolds stresses which shows turbulence in momentum equation.

## Turbulence Model

The Reynolds stresses in momentum equation show the fluctuating component of fluid properties. The Reynolds stresses assumed to be proportional to the rate of deformation and these stresses are solved by turbulent eddy viscosity model.

$$\tau_{ij} = -\rho\overline{u'_j u'_i} = \mu_t\left[\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right] - \frac{2}{3}\delta_{ij}k \qquad (3)$$

The turbulent eddy viscosity calculated by using this relation

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} = C\rho\upsilon\ell \qquad (4)$$

Most of computational works on diesel engine spray employ two-equation turbulence models, primarily the Realizable $k-\varepsilon$ model for modelling turbulence phenomena of continuous phase. The Realizable model is same as standard $k-\varepsilon$ but the difference is as $C_\mu$ considered as variable parameter and also improved $\varepsilon$ equation is used.

The kinetic energy equation

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho k u_i) = \qquad (5)$$

$$\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + 2\mu_t S_{ij}S_{ij} - \rho\varepsilon + G_k + G_b - Y_M + S_k$$

The improved epsilon equation is

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\varepsilon u_i) = \qquad (6)$$

$$\frac{\partial}{\partial x_j}\left(\left(\varepsilon + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + \rho C_1 S_\varepsilon - \rho C_2\frac{\varepsilon^2}{k+\sqrt{\upsilon\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b$$

The constant values in above equations

$C_{1\varepsilon}$ =1.44, $C_2$ =1.9, $\sigma_k$ =1.0, $\sigma_\varepsilon$ =0.85

## Discrete Phase

The diesel fuel is injected as discrete droplets form with different diameter by using Rosin-Rammler distribution function. The maximum diameter of droplet is equal to the nozzle size. The diesel fuel is injected as discrete phase (Droplets) in engine cylinder and every droplet is treated as single constant mass that moves in continuous gas phase [7]. The governing equation used for discrete partical phase

$$\frac{du_p}{dt} = \frac{3\rho_a}{4\rho_l}C_d\frac{|u_{rel}|}{d_p}u_{rel} + g_i + F_x \qquad (7)$$

where $u_{rel} = u - u_p$ is relative velocity between continuous (air) and discrete phase (diesel). The $u, u_p$ is air and droplet velocity respectivley. The terms on the right-hand side of above equation show the drag force, gravitation and additional forces which contribtes in air motion. The diesel drops deform during motion due to aerodynamic drag, in the drag coffiecient deformation of droplet effect is counted [8].

The droplet breakup is captured using KHRT model. The figure 1 reprsent the primary and secondary breakup and break-up models which use to capture these phenamena's. The Kelvin-Helmholtz & Rayleigh-Taylor (KHRT) model have two modes of atomization [9]. The KH breakup capture the unstable waves developing on the liquid jet due to the relative velocity of jet and gas. The RT atomization consider the waves developing on spray drops due to acceleration normal to the droplet-gas interaface.

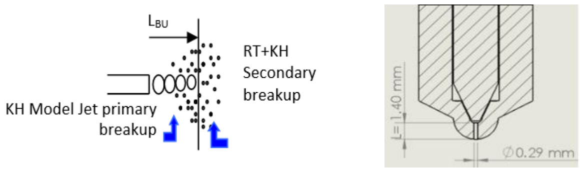


Figure 1, 2: Primary, Secondary breakup and nozzle geometry

## Numerical Methadology

As menstioned before, for simplicity assumed that diesel is atomized at nozzle exit, so eulerian-Lagrangian approach [3,10] used in this reseach paper. The discritization of spatial term in Navier's Stokes equation is done by finite volume method, using second order-upwind algorithm for convective terms in turbulacne and momentum equations, PRESTO method is used for pressure term. The discritization of transient term is done by fully implicit second-order scheme. The pressure velocity coupling in countinues phase is deal with SIMPLEC algorithm. The droplets are considered as single mass praticle in Langraigian approach and its veolicty decrease as droplets move in contineues phase due to the aerodynamic drag force. As the drop break into child droplet its mass decrease. The compressed air and descrete phase continuously exchange the energy, velocity, mass and momentum.

## Inlet Conditions

In langragian spray simulation the solid cone spray injection model is used. For applying Rosin-Rammler distribution fuction user provids the probability density function of spray drops diameters. Its required droplet maximum, minimum and average diameter as well as spray cone angle and spread factor. Here taken different fuel injection pressures. That injections pressure at nozzle inlet (pump pressure) but at exit of nozzle pressure vary. So in this paper the theoratical inlet

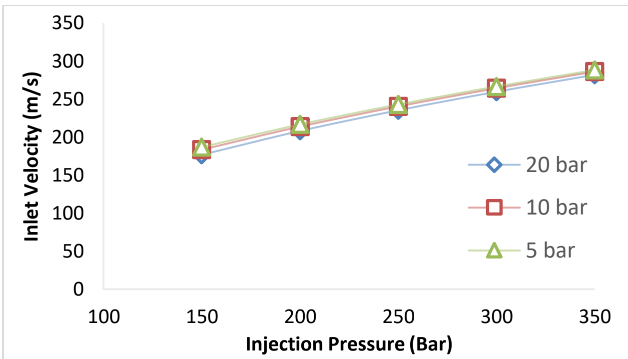


Figure 3: The injection pressure and inlet velocity relation

velocity is calculated base on injection pressure and cylinder pressure by using this formula, and given the velocity inlet of langragian phase with calculated velocites. The figure 3 shown the inlet velocities with different injection pressure. Its calculted by using bernollli equation 8.

$$U_{theor} = C_v\sqrt{2\left(\frac{P_1}{\rho_l} - \frac{P_2}{\rho_a}\right)} = C_v\sqrt{2\frac{\Delta P}{\rho_l}} \qquad (8)$$

### Nozzle & Injection Parameters

The multi-hole diesel injector is complex in nature due to the phenomena of jet-to-jet interaction and inside cavitation of each nozzle-hole [14]. To handle these phenomena's for multi-hole nozzle required high computational power and take much time so in this research single-hole nozzle is considered with diameter of 290 micron. The nozzle is perpendicular to the top surface of cylinder and the nozzle length to diameter ratio is 4.85. The nozzle geometry in shown in figure 2 with dimensions. The diesel was injected at 5 different injection pressures (150, 200,250,300,350 bar) into compressed quiescent air at 600 K and density of 2.906, 5.8 and 11.6 kg/m$^3$ corresponding pressure 5, 10 and 20 bar. The number of parcel injected in cylinder same as number of time step and each parcel have constant mass.

### Validation

The Sauter Mean Diameter (SMD), droplet velocity and spray penetration results are validated with the published research work of Daliang Jing et al. [12] and Abbas Ghasemi et al. [7] and found the good agreement between the results. The spray tip penetration results are compared with experimental result of Jing et al. and numerical results show similar profile. The results at 200 bar injection pressure and 5.8 kg/m$^3$ in cylinder air density for spray tip penetration is validated from Jing et.al research data as shown in figure 7. The Phase Doppler Particle Analyzer (PDPA) test results of Jing et al. research presented the droplet velocity the firstly decease gradually in spray head and suddenly decline in tail at end of injection. The similar result found in this numerical simulation which shown in figure 4 at time 2 msec. The Droplet diameter normalized by nozzle diameter is plotted and compared with Abbas Ghasemi et al. at maximum 57.5 mm tip penetration, the obtained results show a good agreement with the published work trend. All the resutls have same trend as published in above menstioned litrature.

### Results & Discussion

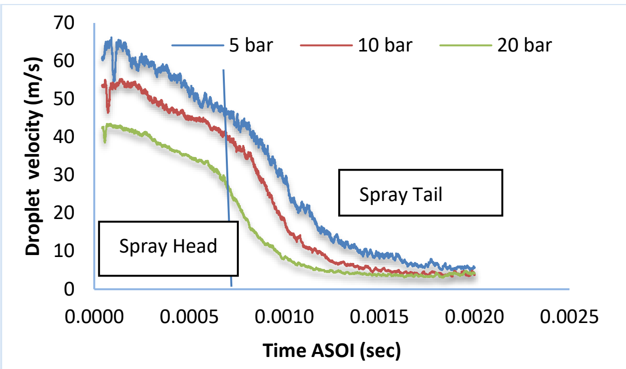


Figure 4: Droplet velocity

Sprays can be divided into two distinct parts based on the velocity profiles of drops as presented in figure 4. These are called spray heads and tails. [13]. After the injection delay, first velocity sudden decreases and then increases upto a stable time period which forms the spray head. The droplets present in the spray head encounter the drag force and due to this fact their velocity decreases but the droplets present behind the spray head relatively experience a small drag force and velocity of this part is high. After reaching the measuring volume a catastrophic change in velocity occurs, this is called the spray tail. As shown in figure 4 at the spray tail the droplet velocty decrease slowly and reach minimum droplet velocity. The veolcity not further decrease after 2 milisecond.

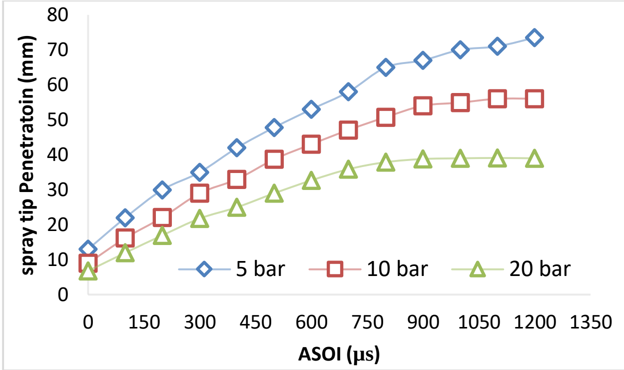


Figure 5: The spray tip penetration at different back pressures at 350 bar injection pressure

The variation in ambient air density (back pressure) considerably effect the macroscopic properties of spray. In this work the cylinder air pressure varied from 5 to 20 bar and diameter of spray nozzle kept constant 0.290 mm. The effects of density variation are shown in the in figure 5 with constant injection pressure of 350 bar. The results show that as the air density increase spray penetration reduce because the air drag resist the spray penetration, also change the spray shape as shown in figure 6. In this case, the tip penetration and droplet velocity is maximum at 5 bar back pressure and 350 bar injection pressure. As the ambient density was increased the droplet velocity and spray penetration was reduced, and SMD was increased results shown in figure 4,5 and 8 respectively.

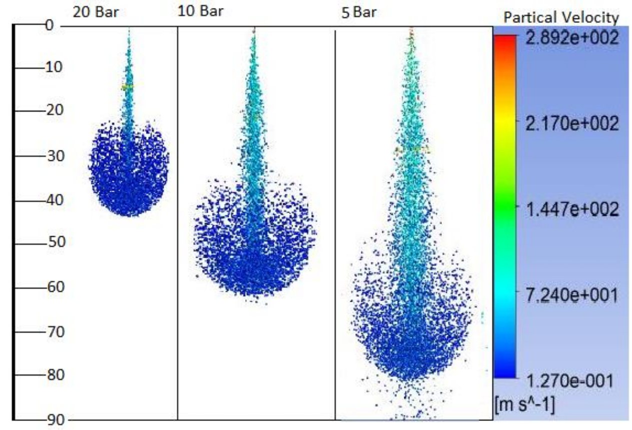


Figure 6: The Spray contours at 350 bar injection pressure

The spray contours in figure 6 are represent the spray shape at different in cylinder air density with 350 bar injection pressure. As the in cylinder air pressure decrease the spray penetrate more and the variation of spray color present the variation of spray velocity. The spray velocity is maximum at inlet and decrease as it is penetrate in compressed air. At near the nozzle-hole drops are close to each other, in larger size and face low drag effect. As the lager drops break into smaller and spread, these particles face more drag, due to that velocity of particles decrease.

Injection pressure has immense effect on the spray characteristics. High velocity and fine spread of droplets occur due to higher inertia. When injection pressure is increased, same amount of fuel injected is achieved in small interval of time. It is observed that the maximum spray tip penetration gradually increase with raising of injection pressure from 150 bar to 350 bar. The spray outlet velocity at nozzle exit is function of injection, back pressure and fuel density as shown in equation 8. So as the injection pressure raise in results the spray velocity also increase. The spray tip penetration is plotted at different injection pressures shown in figure 7 and the ambient density kept constant at 5.8 kg/m3.

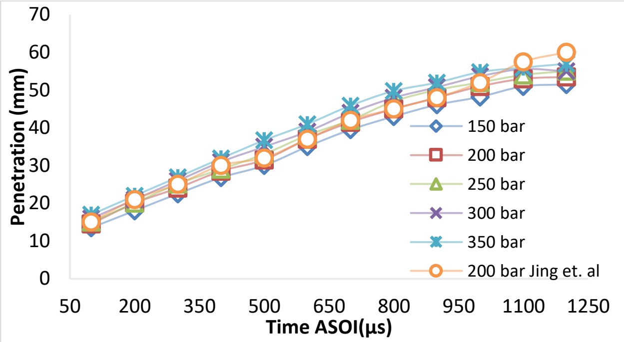


Figure 7: The spray tip penetration at different injection pressure and 10 bar back pressure

The penetration is not much effected by raising in injection pressure as it is effected by back pressure.

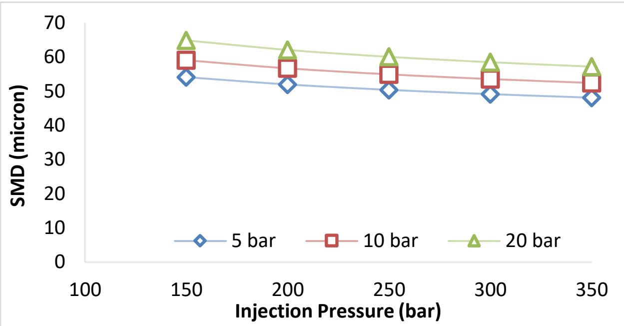


Figure 8. SMD of spray at different back pressure

Here Souter mean diameter (SMD) is average droplet size of spray, it is ratio between the total volumes of drops to the surface area of drops. The figure 8 represent the relation between SMD with injection pressure at different back pressure. The variation in injection velocity effect the SMD. At higher injection pressure the SMD is decreased. The graph 8, show inverse relation between injection pressure and SMD, whenever the back pressure have direct relation with SMD. These results generated at time 1.2 ms.

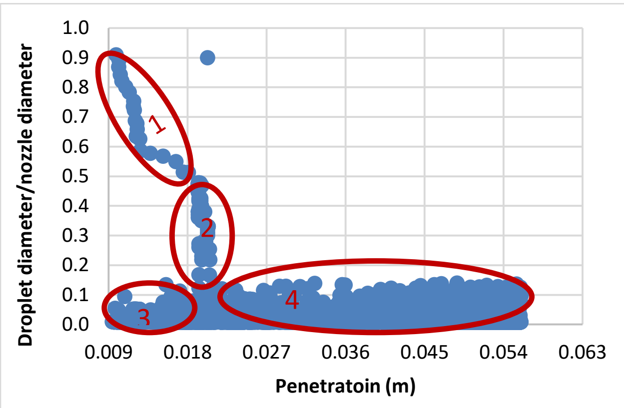


Figure 9: Droplet diameter normalized by nozzle diameter

Jet breakup can be analysed by the SMD but the distribution of droplet in time and space is more helpful in analysis. Droplet diameters are plotted at 350 bar injection pressure and 5.8 kg/m3 ambient air density. In figure 9, it can be seen that the size of droplet is near to the size of nozzle-hole in region 1. In the region 3 there is more aerodynamic effect on the jet due to faster atomization of droplets in this region. Region 2 is at a distance of 19-22 mm from the nozzle-exit and droplets have travelled a pretty distance and large number of droplets breakup occur. Thus in region 2 big droplets have become finer ones. Region 4 is also away the from the nozzle exit and because of high injection pressure droplets are in a fine smaller size.

## Conclusion

The Fluent 14.5 software is used for simulation of diesel spray in this research and Eulerain-Langragian model used. The fuel is injected in the form of droplets (Lagrangian phase) in continuous (Eulerian) air medium, which is considered quiescent before injection and calculated the Eulerian phase properties by RANS $k-\varepsilon$ Realizable model. The number of simulations have done on different conditions, by changing the injection pressure as well as the back pressure. The numerical results are validated by Abbas Ghasemi [7] and Daliang Jing [12]. The spray tip penetration is increase by injection pressure but increment of ambient density decreases the penetration.

The Souter mean diameter variation with injection pressure and ambient air density is presented. As the injection pressure increases the SMD decreases because at higher inlet velocity the catastrophic breakup occur and fuel evaporate slightly faster in higher injection pressure and slow breakup at higher ambient density. The droplet size distribution is done to understand the drop size variation in space and injection time. The different atomization regions, location and time of breakup were illustrated. The detail analysis have done on constant volume cylinder, it would be helpful to increase the engine performance.

## References


[1] Zhang, J. & Fang, T., Spray Combustion of Biodiesel and Diesel in a Constant Volume Combustion Chamber, *SAE Technical Paper* 2011-01-1380.

[2] Irannejad A, Jaberi F., Large Eddy Simulation of Turbulent Spray Breakup and Evaporation, *International Journal of Multiphase Flow 2014*; 61:108–128.

[3] Som, S., Aggarwal, S. K., El-Hannouny, E. M., Investigation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector, *Journal of Engineering for Gas Turbines and Power*, Vol. 132 / 042802-1, 2010.

[4] Reitz, R.D., Modeling Atomization Processes in High-Pressure Vaporizing Sprays, *Atomization and Spray Technology*, 1987, pp. 309-337.

[5] Park, S. W., Kim, H. J., & Lee, C. S., Investigation of Atomization Characteristics and Prediction Accuracy of Hybrid Models for High-speed Diesel Fuel Sprays, *SAE Paper* 2003-01-1045.

[6] Rotondi, R., Bella, G., & Grimaldi, C., Atomization of High-Pressure Diesel Spray: Experimental Validation of a New Breakup Model, *SAE paper* 2001-01-1070.

[7] Abbas Ghasemi, Kohei Fukuda, Numerical Investigation of Spray Characteristics of Diesel Alternative Fuels, *SAE paper* 2012-01-1265.

[8] Baumgarten C., Mixture Formation in Internal Combustion Engines, *Springer-Verlag: Berlin, Heidelberg*, 2006.

[9] Beale, J.C. and Reitz, R.D., Modeling Spray Atomization with the Kelvin-Helmholtz/Rayleigh-Taylor Hybrid Model, *Atom and Sprays*, Vol.9, pp.623-650.

[10] Abbas Ghasemi, Aaron Pereira, Evolution of Liquid and Gas Phases in Multi-plume Spray Injection, *Int.J. Energy Res. 2016*, DOI: 10.1002/er.3562.

[11] Reitz, R.D., A photographic study of flash-boiling atomization, *Aerosol Science and Technology*, 1990. 12(3): p.561-569.

[12] Daliang Jing, Zhi Wang, A Turbulence and Cavitation Induced Breakup Model for Fuel Spray Modeling, *SAE paper* 2014-01-2737.

[13] Ziman Wang, X. Dai et al., Nozzle Internal Flow and Spray Primary Breakup with the Application of Closely Coupled Split Injection Strategy, *fuels*. DOI: 10.101