IJSTE - International Journal of Science Technology & Engineering | Volume 3 | Issue 06 | December 2016
ISSN (online): 2349-784X

A Review on Cavitation in Fuel Injector Nozzle

Ronak Gohel
PG Student
Department of Mechanical Engineering
LDRP-ITR, Ganghinagar

Prof. Neel Joshi
Assistant Professor
Department of Mechanical Engineering
LDRP-ITR, Ganghinagar

Prof. Hemal Patel
Assistant Professor
Department of Mechanical Engineering
LDRP-ITR, Ganghinagar

Abstract

Over the last several decades, there has been a tremendous increase in the demand of automobile vehicles for the purpose of public transport. The reason behind such increasing demand is attributed to the increase in world population. However, the fuel economy and exhaust emission depends to a great extent on how well the combustion process takes place, which depends on the fuel spray behavior. Cavitation directly influence on fuel spray behavior. The characteristics of a cavitation depend on the fuel properties, geometry of the injector and the flow conditions upstream (inside) and downstream of the injection nozzle. It is required to understand behavior of cavitation inside fuel nozzle. Effort has been made to understand this phenomena and effect on injector nozzle. Detailed review has been carried out to highlight work has been done by various researcher.

Keywords: Cavitation, Fuel Injector, Nozzle, Choking

I. INTRODUCTION

In a flowing liquid whenever the local pressure in certain regions drop below the vapour pressure of the fuel concerned then vapour cavities or bubbles appear and the phenomenon is known as "Cavitation". The flow inside the fuel injector thus becomes complicated to study as in most cases it is a high speed two-phase flow in a small passage. This paper reviews the wide range of studies conducted based on cavitation in fuel injector nozzle. Complete review is carried out based on Experimental & CFD studies carried out by various researchers.

II. EXPERIMENTAL INVESTIGATION

Depending on nature of combustion process, the injection pressure reaches 1000 to 2000 bar in Diesel engines and is around 100 bar in new techniques of direct injection in Otto engines [11]. Due to extreme parametric condition it is very difficult to capture cavitation inside diesel engine fuel injector. Only few research is available with real size nozzle geometry & high injection pressure.

Winklhofer et al (2001) initiate experiment study with realistic condition of fuel injector and developed optical method to capture cavitation. Study reported extensive analysis of the structure of cavitation flow in Diesel injector-like geometries (transparent nozzle of rectangular cross-section). The flow geometry of interest was eroded into 0.3 mm thick steel sheets, which were sandwiched between a pair of sapphire windows. Visualization of the vapour field distribution, the pressure in the liquid, measurements of velocity, as well as, mass flow were reported for standardized throttle sets in stationary flow conditions at inlet pressure levels of 100 bar. The back-pressure was adjusted to provide the desired pressure drop. These experiments were performed for injection into liquid, so no hydraulic flip could occur. The results showed that the discharge coefficient was significantly affected by the cavitation level at super-cavitation flow regime, when the flow was choked and the mass flow rate became independent on the downstream pressure. According to the observations, the conicity of the orifice had no apparent effect on the variation of the discharge coefficient with the cavitation number when the flow was choked. Though, the nozzle shape was found to determine the inception and critical flow conditions. Recently in 2015 Winklhofer et.al describe effect of injection pressure on diesel engine combustion. Tests have been performed of fuel pressure's influence on injector fuel flow, on spray and mixture formation and on combustion. Optical diagnostics were applied to measure fuel flow in 2D model spray hole geometries, to study spray stability and analyse flow in the bowl of combustion chambers.

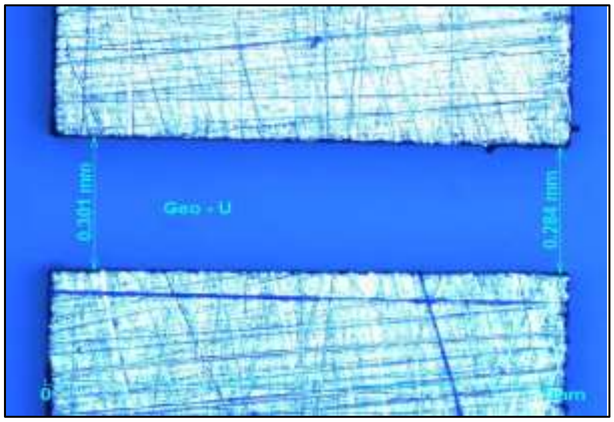

Fig. 1: Throttle geometry "U" in 300 µm thick sheet steel

Akira Sou et.al (2016) capture transient cavitation motion and turbulent velocity in a rectangular nozzle by using a high speed camera and Laser Doppler Velocimetry. (LDV). Akira et.al validate his mathematical model with experimental results with transient cavitation motion. In this study we propose a new combination of Large Eddy Simulation (LES), Eulerian-Lagrangian Bubble Tracking Method (BTM), and the Rayleigh–Plesset (RP) equation to simulate an incipient cavitation, in which only cavitation bubble clouds appear. As a result, a recirculation flow and a cavitation cloud shedding are accurately predicted by LES using a fine grid, and the RP equation for all nuclei tracked in a Lagrangian manner.

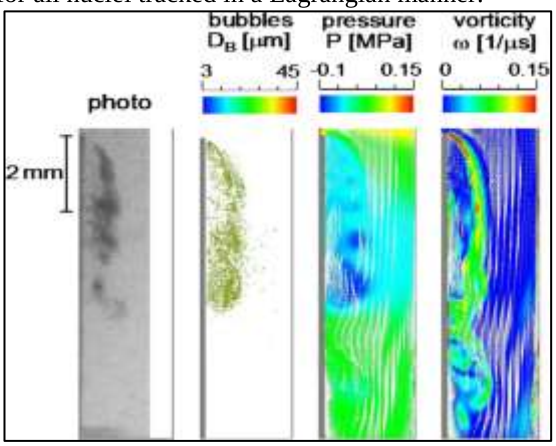

Fig. 2: Observed and calculated cavitation flows (Vn = 12.8 m/s)

Hiroyasu et al. conducted one of the useful experiment using large-scale transparent nozzle to predict the presence of cavitation. They took pictures in the low speed of nozzle flow to observe the correlation between the nozzle cavitation and spray atomization. They found out that spray atomization is improved by the extension of cavitation to the exit of injector nozzle, and first break-up length is decreased due to the present of cavitation inside nozzle. These efforts provide an important correlation to better understand the cavitating nozzle flow and its influence on spray break-up. They also showed that the nozzle length does not have any important effect on the discharge coefficient.

A.Morozov et al. (2004) studied surface erosion due to cavitation. He concluded that: (a) Position of erosion start is always connected with microscopic phase boundaries of the flow. (b) Erosion strongly depend on the flow temperature. Higher flow temperature causes earlier and fast erosion. (c) Higher target surface roughness causes earlier and faster surface erosion in case of stationary flow conditions.

III. NUMERICAL INVESTIGATION

Since Experimental method required great expertise to capture Cavitation phenomena, researcher put deep interest in CFD analysis of Cavitation. Although cavitation is two-phase flow problem, which is always difficult and required high end computing facility. Recent development in computational processing capacity helps to simulate complex fuel injection phenomena.

Weixing Yuan et. al (2001) developed a numerical model to investigate effect of viscous and turbulent effects and rounding of inlet tip of fuel injector nozzle on cavitation region. Numerical results indicated that the overall extension of the cavitation region depended on the liquid quality and the nozzle pressure difference, i.e. the lower pressure at the nozzle exit caused larger overall extension of the cavitation. He demonstrated that the cavitation region of the Euler solution is much larger than those of Navier–Stokes solutions. This is because the primary role of viscosity in this flow is to increase dissipation and flow losses downstream of the contraction at the nozzle throat. Due to the viscous dissipation the vapor fraction is transported more slowly, therefore the two-phase regions of the Navier– Stokes simulations are smaller. The comparison with the experimental velocity distribution indicates that the turbulent Navier–Stokes simulation yields the best agreement with the experiment. The numerical study also demonstrated that the rounding of the nozzle inlet tip inhibited the overall extension of the cavitation region.

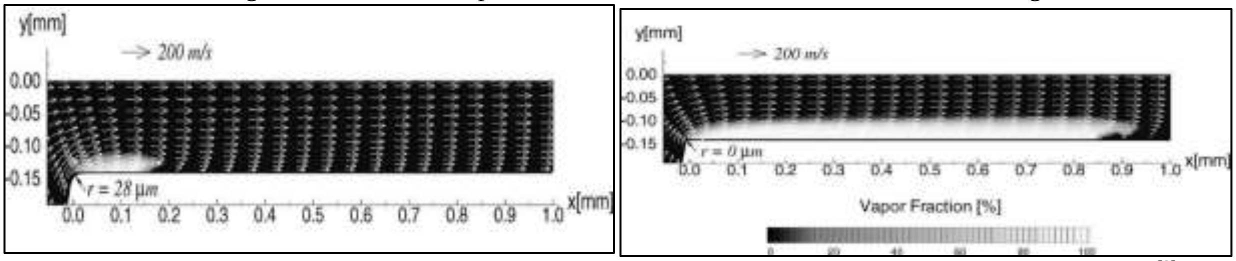

Fig. 3: Computed steady vapour fraction distributions for different tip rounding r. (For P_{in}=80 bar and P_{out}=21 bar)[9]

At the same time, the numerical simulations pointed out that the flow separation in injector nozzles might be caused due to the sharpness of the nozzle inlet tip as well as by the cavitation process. The length of the two-phase flow region in the case of r =0 µm was shorter than that in the case of r =14 µm. according to him this interesting phenomenon was most probably a result of the effect of the separation of the primary fluid on the cavitation. The separation formed by the primary single flow influences the extension of the two-phase region.

S.Som et.al (2010) Investigate nozzle flow and cavitation characteristics in diesel injector. A mixture based model in FLUENT V6.2 software is employed for simulations. In addition, a new criterion for cavitation inception based on the total stress is implemented, and its effectiveness in predicting cavitation is evaluated. Injection pressure (800-1600 bar), needle lift position, and fuel type has been studied for cavitation. Four different fuel has been0 tested i.e. n-Dodecane, European diesel, Chevron diesel & Viscor/cerium blend. Results indicate that the cavitation characteristics of n-dodecane are significantly different from those of the other three fuels investigated. The effect of needle movement on cavitation is investigated by performing simulations at different needle lift positions. Cavitation patterns are seen to shift dramatically as the needle lift position is changed during an injection event.

Saha Kaushik et al. (2013) has developed a cavitation model for the internal two-phase flow of diesel and biodiesel fuels in fuel injectors under high injection pressure conditions. The model was based on the single-fluid mixture approach with newly derived expressions for the phase change rate and local mean effective pressure the two key components of the model. The effects of the turbulence, compressibility, and wall roughness were accounted in the model and model validation was carried out by comparing the model predictions of probable cavitation regions, velocity distribution, and fuel mass flow rate with the experimental measurement available in literature. It was found that cavitation inception for biodiesel occurs at a higher injection pressure, compared to diesel, due to its higher viscosity. However, super-cavitation occurred for both diesel and biodiesel at high injection pressures. The effect of liquid phase compressibility became considerable for high injection pressures. Wall roughness was not an important factor for cavitation in fuel injectors. In 2016 Saha Kaushik et. al assess available cavitation model based on single and two fluid approach. Investigation has been carried out with Saha-Abu-Ramandan-Li (SAL), Schnerr-Sauer (SS), and Zwart-Gerber-Belamri (ZGB) models. Numerical predictions are compared with experimental results available in literature, qualitatively with experimental images of two-phase flow in an optically accessible nozzle, and quantitatively with measured mass flow rate and velocity profile.

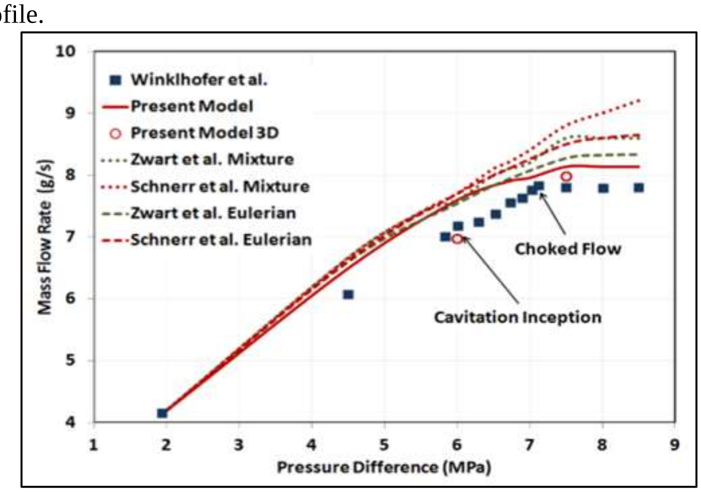

Fig. 4: Comparison of results of various cavitation models [6]

Balaji Mohan et al. (2014) performed a numerical study to investigate the effect of different fuels, injection pressure and different geometries on the cavitation phenomenon inside the injector nozzle hole. The following observations were made based on the results from the numerical simulations performed. The cavitation phenomenon was found more pronounced with diesel fuel than with SME bio-diesel because of its fluid properties. Cavitation in diesel occurs at higher Reynolds number compared to bio-diesel because of higher dynamic pressure due to lower viscosity. The formation of cavitation of both fuels with time were compared with the formation of vapour phase which also showed that bio-diesel inhibited the cavitation phenomenon because of higher viscosity. The mass flow rate increased with increasing conicity (K-factor) for larger inlet and exit diameter. But no significant effect was observed with smaller diameter nozzles. Significant decrease in mass flow rate was observed for smaller diameter with same K-factor. The exit velocity for higher K-factor was higher, which might help in finer atomization of fuel.

Eric Pomraning et.al (2014) investigate cavitation phenomena with three-dimensional nozzle geometry. Simulation of a five-hole injector, including moving needle effects, are compared to experimental measurements. Both the transient needle moving case and steady no-needle case are simulated. The influence of the minimum needle lift is studied in the transient case. The study shows that the minimum needle lift only affects the initial needle opening and closing stage and will not have a significant influence on the steady mass flow rate at a fully nozzle-opened status. Both the simulated transient and steady mass rate approach the experimental measured values within a reasonable error range. The error may be attributed to the difference of the Diesel fuel properties and nozzle geometry between the simulations and experiments.

Zandi A. et al. in his work, simulated the cavitation phenomenon in an injector nozzle using the Eulerian-Eulerian two-fluid method. The effects of dynamic and geometrical parameters on the cavitational flow of the nozzle were investigated. The following observations were made based on the results obtained. As the injection pressure was increased and the cavitation developed, the amount of the vapour phase at the outlet of the nozzle increased which resulted in the decrease in the discharge coefficient. Moreover, the flow regime inside the nozzle changed by increasing the injection pressure. The flow regime at lower pressures was non-cavitational, and then, it was cavitational. The effect of the back pressure on the nozzle flow depended on the difference between the inlet and the outlet pressures. At lower levels of pressure difference where the flow was in non-cavitational or cavitation inception condition, lowering the back pressure increases the mass flow rate. In this situation, since the cavitation does not develop to the outlet of the nozzle, the discharge coefficient remains constant. At high enough levels of pressure difference, because of the occurrence of the choked flow phenomenon, the back pressure does not affect the characteristics of the internal nozzle flow like mass flow rate, and cavitation quality. In this situation, since the cavitation developed to the outlet of the nozzle, the discharge coefficient decreased as the back pressure was lowered. As the inlet curvature radius of the orifice increased, resistance to flow decreased and less pressure drop occurred at this point. As the pressure drop decreased, the cavitation intensity decreased, and as a result, the effective area increased which led to increasing the discharge coefficient. Conicity of the orifice had a similar effect as the inlet curvature radius of the orifice. Increasing the conicity of the orifice increased the uniformity of the flow. Increasing the uniformity of the flow decreased the disturbances of the flow; it, also, decreased the cavitation intensity, and consequently, increased the discharge coefficient. The effect of the orifice length on the internal nozzle flow depended on the flow regime. When the orifice was in a non-cavitational condition, friction losses increased as the length of the orifice increased, and the discharge coefficient decreased. Nevertheless, when the nozzle was in super-cavitational condition, because of the flow detachment, there was no contact between the liquid fuel and the solid surface. So, increasing the length of the orifice did not have any considerable effects on either the friction losses or the discharge coefficient.

IV. Conclusion

Literature review shows that CFD is strongly capable tool to analyzed Cavitation within fuel injector nozzle in diesel engine. Moreover study helps to rectify various parameter which influence on fuel spray. Some of the literature provide helps to model cavitation problem with real values of parameter i.e. nozzle geometry & dimension, injection pressure & back pressure, vapour pressure of different fuel with other properties. At the end of literature survey, it is observed that some of the parameter still required further refinement. While effect of certain condition is required to explore, i.e. effect of working environment on Cavitation phenomena with temperature effect required to exploration.

References

[1] A. Morozov A., T. Neger, E.Winklhofer “study of surface erosion by high pressure cavitating diesel flow” (2004)
[2] Akira Sou Akira, Baris Bicer, Akio Tomiyama “Numerical simulation of incipient cavitation flow in a nozzle of fuel injector” Journal of Computers & Fluids, (2014)
[3] Balaji Mohan, Wenming Yang, Siawkiang Chou, “Cavitation in injector nozzle holes-A parametric study”, National University of Singapore, Singapore, 2014
[4] David Grief, Alexander Morozov, E Winklhofer, Reinhard Tatschl “Experimental and numerical investigation of erosive effects due to cavitation within injection equipment”, 4th International Conference on Computational Heat and Mass Transfer-France (2005)
[5] Eric Pomraning, Senecal Peter, Battistoni Mitchele, “Validation of a Three-Dimensional Internal Nozzle Flow Model Including Automatic Mesh Generation and Cavitation Effects”, article in journal of engineering for gas turbines and power, September 2014
[6] H. Hiroyasu, “Break-up length of a liquid jet and internal flow in a nozzle”, Proceedings of 5th International Conference of ICLASS, 1991
[7] Mehul Bambhania, Nikul Patel, “Numerical simulation to predict cavitation inside diesel engine fuel injector nozzle”, International conference on Multidisciplinary Research Approach for the accomplishment of academic excellence in Higher technical education through industrial practices, June 2016, Pattaya-Bangkok
[8] Saha Kaushik, Ehab Abu-Ramadan, Xianguo Li1, “Modified Single-Fluid Cavitation Model for Pure Diesel and Biodiesel Fuels in Direct Injection Fuel Injectors”, University of Waterloo, Waterloo, Canada, June 2013
[9] Saha Kaushik, “Modelling of cavitation in nozzles for diesel injection applications”, Waterloo, Ontario, Canada, (2014)

[10] Som S., Aggarwal S.K., El-Hannouny E. M., Longman D.E., “Investigation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector”, article in journal of engineering for gas turbines and power, April 2010
[11] Weixing Yuan, Jürgen Sauer, Günter H. Schnerr, “Modelling and computation of unsteady cavitation flows in injection nozzles”, University of Karlsruhe, Germany (2001)
[12] Winklhofer E, Kull E, Kelz E, Morozov A., “Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions”, ILASS-EUROPE, Zurich, Switzerland (2001)
[13] Winklhofer E, H. Dembinski, H.-E. Angstrom “Injection pressure as a means to guide air utilization in Diesel engine combustion”, Fuel System Conference (2016)
[14] Weixing Yuan, Gunter H. Schnerr, “Numerical simulation of two phase flow in injection nozzle: Interaction of cavitation and External jet formation” Journal of Fluids Engineering 2003 by ASME NOVEMBER 2003

