Effect of Needle Height on Discharge Coefficient and Cavitation Number

Azadeh Yazdi, Mohammadreza Nezamirad, Sepideh Amirahmadian, Nasim Sabetpour, Amirmasoud Hamedi

***Abstract*—**Cavitation inside diesel injector nozzle is investigated using Reynolds-Stress-Navier stokes equations. Schnerr-Sauer cavitation model is used for modeling cavitation inside diesel injector nozzle. The carrying fluid utilized in the current study is diesel fuel. The flow is verified at the beginning by comparing with the previous experimental data and it was found that K-Epsilon turbulent model could lead to a better accuracy comparing to K-Omega turbulent model. Moreover, mass flow rate obtained numerically is compared with the experimental value and discrepancy was found to be less than 5% - which shows the accuracy of the current results. Finally, a real-size four-hole nozzle is investigated and the flow inside it is visualized based on velocity profile, discharge coefficient and cavitation number. It was found that the mesh density could be reduced significantly by utilizing periodic boundary condition. Velocity contour at the mid nozzle showed that maximum value of velocity occurs at the end of the needle before entering the orifice area. Last but not least, at the same boundary conditions, when different needle heights were utilized, it was found that as needle height increases with an increase in cavitation number, discharge coefficient increases, while the mentioned increases is more tangible at smaller values of needle heights.

***Keywords*—**Cavitation, diesel fuel, CFD, real size nozzle, mass flow rate.

I. Introduction

CAVITATION is related to bubble cluster formation in a flow that leads to two-phase mixture of vapor and gas [1] as the local pressure decreases to a very minimal value called vapor pressure or more accurately critical pressure. Generally, liquid to vapor transition occurs in two away in which the first one is by heating the carrying fluid at a constant pressure that is called boiling and the second one is diminishing the pressure at a constant temperature which is called cavitation [2]-[4]. Isothermal process is assumed for the phase transition as vapor density is smaller than liquid by about two orders [5]. Excessive stress that can cause liquid rupture of liquid continuum is another scientific definition for cavitation [6], [7]. Cavitation is mainly seen in pumps, valves, nozzles, etc., in which mostly is not desirable as it can decrease system efficiency and can also cause malfunction. Among injectors, cavitation can positively improve spray process by causing primary jet break up and enhancing liquid atomization. It is believed that primary breakup happens at the very neighborhood of nozzle tip due to turbulence, inherent instability, aerodynamics caused by formation of cavitation in orifice of the nozzles [8], [9]. Moreover, cavitation enhances velocity of the liquid by diminishing exit area of the liquid at the nozzle exit [10]. Cavitation occupies in the orifice of the nozzle from the starting point of orifice toward the orifice outlet in which emerging spray formation is affected [11]. A better combustion quality, diminishing particulate emission and exhaust gas, improving efficiency and lowering fuel consumption are among the results of improving spray development. On the other side, decreasing flow efficiency is one of the side effects of cavitation as it affects jet exiting the nozzle [12], [13]. Decreasing the life and performance of the orifice is a result of material erosion caused from imploding cavitation bubbles [14]. Therefore, in order to make nozzle design more efficient, understanding the amount and source of cavitation beside optimum amount of cavitation is of interest. Both dynamic factors and geometrical factors can cause cavitation inception and extension of the mentioned phenomena depends on the pressure difference [15], [16]. In the current study we have two major objectives. First of all we will simulate three dimensional rectangular shape nozzle using Ansys Fluent software, then the obtained results will be verified with Winklhofer et al. [17] experimental data.

II. Mathematical Model

In this study two phase flow is simulated based on single fluid approach for homogenous framework. The continuous and momentum equations for homogenous flow can be written as following:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla.(\rho_m \vec{v}_m) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho_m) + \nabla.(\rho_m \vec{v}_m.\vec{v}_m) = -\nabla p + \nabla.[\mu_m(\nabla\vec{v}_m + \nabla\vec{v}_m^T)] + \rho_m\vec{g} \tag{2}$$

In which, ρ_m is mixture density, $\vec{v}_m$ is the mixture velocity vector and μ_m is the mixture viscosity. Where mixture density can be written as $\rho_m=\alpha_l\rho_l+\alpha_v\rho_v$ and mixture viscosity can be written as $\mu_m=\alpha_l\mu_l+\alpha_v\rho_v$.

Vapor volume fraction is a criterion in which it determines the estimate of vapor presence versus liquid and can be shown as following:

Azadeh Yazdi is with the Department of Civil and Environmental Engineering, George Mason University, (e-mail: agharibr@gmu.edu).
Mohammadreza Nezamirad is with the Department of Chemical, Biological and Materials Engineering, University of South Florida, Tampa, FL, (Corresponding author, e-mail: mnezamirad@usf.edu).
Sepideh Amirahmadian is with the WSP, Ontario, Canada.
Nasim Sabetpour is with the Department of Computer Science, Iowa State University, Ames, Iowa, (e-mail: nasim@iastate.edu).
Amirmasoud Hamedi is with the Department of Civil and Environmental Engineering, Florida International University, Miami, FL, USA.

$$\alpha_v = \frac{n_b \frac{4}{3}\pi R_B^3}{1 + n_b \frac{4}{3}\pi R_B^3} \tag{3}$$

where, α_v is the vapor volume fraction, n_b is number of bubbles per unit volume and R_B is the bubble radius.

In order to obtain mass transfer, Schnerr-Sauer model is utilized [18]:

$$\frac{\partial}{\partial t}(\alpha_v . \rho_v) + \nabla.(\alpha_v . \rho_v . V_v) = \frac{\rho_v . \rho_l}{\rho} . \frac{d\alpha_v}{dt} \tag{4}$$

In which the source term is define as $R = \frac{\rho_v \rho_l}{\rho} \frac{d\alpha_v}{dt}$. Bubble radius is also defined as $R_B = (\frac{\alpha_v}{1-\alpha_v} \frac{3}{4\pi} \frac{1}{n_b})^{\frac{1}{3}}$.

Discharge coefficient and cavitation number are the two important parameters that are respectively defined as following:

$$Cd = \frac{\dot{m}}{A\sqrt{2\rho_l(P_{in} - P_{back})}} \tag{5}$$

$$K = \frac{P_{in} - P_v}{P_{in} - P_{back}} \tag{6}$$

where $\dot{m}$ is mass flow rate, P_v is vaporization pressure, P_{in} is inlet pressure and P_{back} is the outlet pressure.

III. Verification

The result is first verified with Winklhofer et al. [17] experimental data for rectangular shape nozzle based on mass flow rate obtained in the simulation versus pressure difference and then compared to the experimental results. Also, Fig. 1, compares the results obtained from $K-\varepsilon$ and $K-\omega$ and it was observed that K-Epsilon turbulent model could predict choke cavitation and choke region better; therefore, for the rest of the current study, we will use $K-\varepsilon$ model.

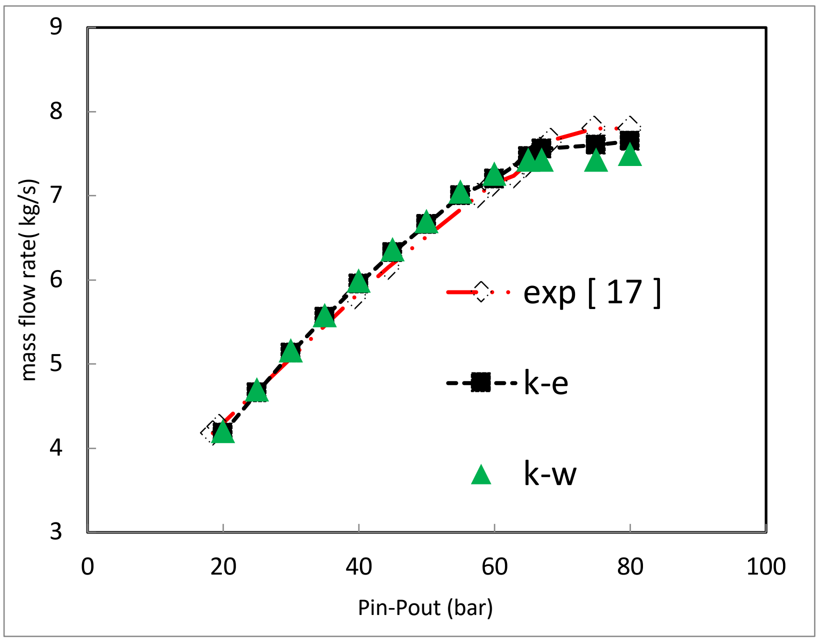

Fig. 1 Mass flow rate versus pressure difference

IV. Geometry of the Four-Hole Nozzle

Fig. 2 shows the geometry and applied boundary condition to only one-sixth of the nozzle. In order to save time and resources, only one-fourth of the nozzle is simulated and instead periodic boundary condition is utilized to investigated flow inside a real size nozzle. As can be seen in Fig. 2, inlet radius is $0.066\ mm$, orifice diameter is $0.32\ mm$ and orifice length is $1.3\ mm$. Also, both inlet and outlet are specified via pressure boundary condition.

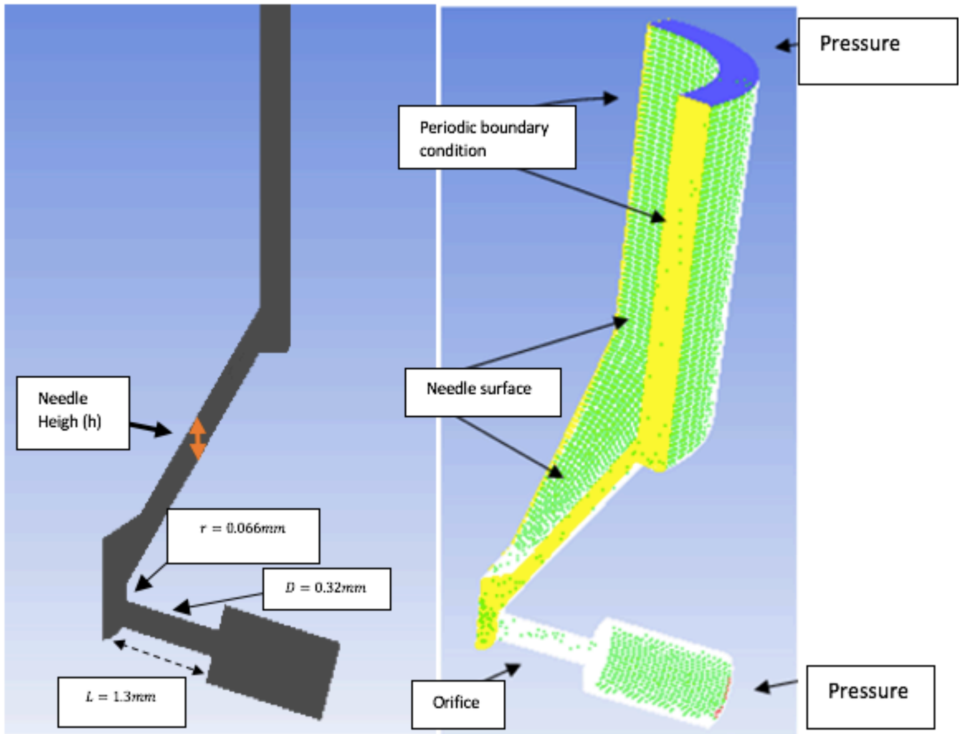

Fig. 2 Geometry and boundary condition of real size six-hole nozzle

V. Result and Discussion

Fig. 3 shows velocity distribution inside mid-nozzle plane when $K^{1/2} = 1.094$ and $h = 0.1\ mm$, where K is cavitation number and h is needle height. As can be seen from the mentioned figure, the highest value of pressure can be visualized at the end of needle and also at the entrance of orifice which can be both useful and harmful. Higher values of velocity is very interesting at the nozzle outlet since it can improve atomization and efficiency of combustion process, while occurrence of high-speed flow before orifice can be harmful to the needle surface and can reduce efficiency of combustion process in a long-term process.

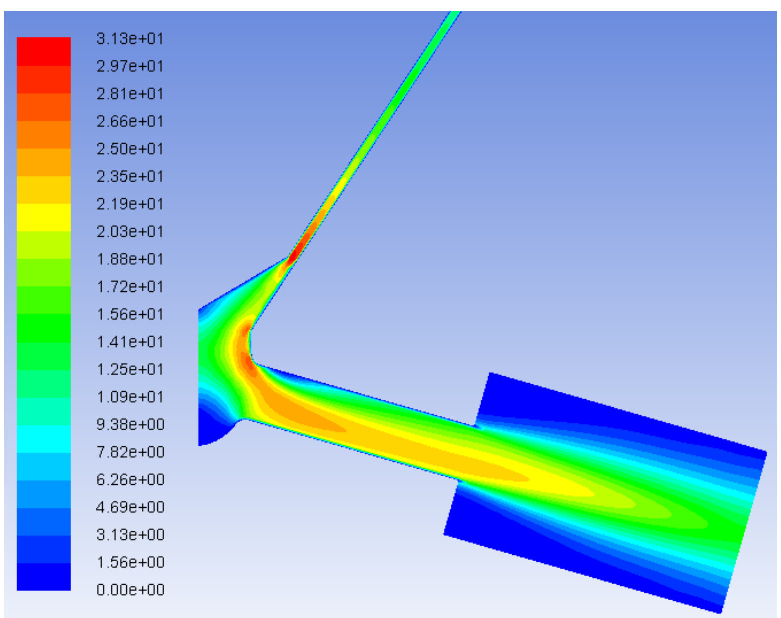

Fig. 3 Velocity distribution inside mid-nozzle plane when $K^{1/2} = 1.094$ and $h = 0.1\ mm$

Fig. 4 shows variation of discharge coefficient versus cavitation number for different needle heights. It can be seen that for higher and lower values of cavitation number, discharge coefficient is independent of cavitation number. For higher cavitation numbers, there is no cavitational hole and discharge coefficient is only function of Reynolds number. Also, it can be visualized that increasing needle height after a certain threshold would not increase discharge coefficient as cavitation number increases. In all needle heights, any increase to cavitation number, increased discharge coefficient at the beginning but then the plateau region is reached in which any further increase in cavitation number, will not increase discharge coefficient in all needle heights.

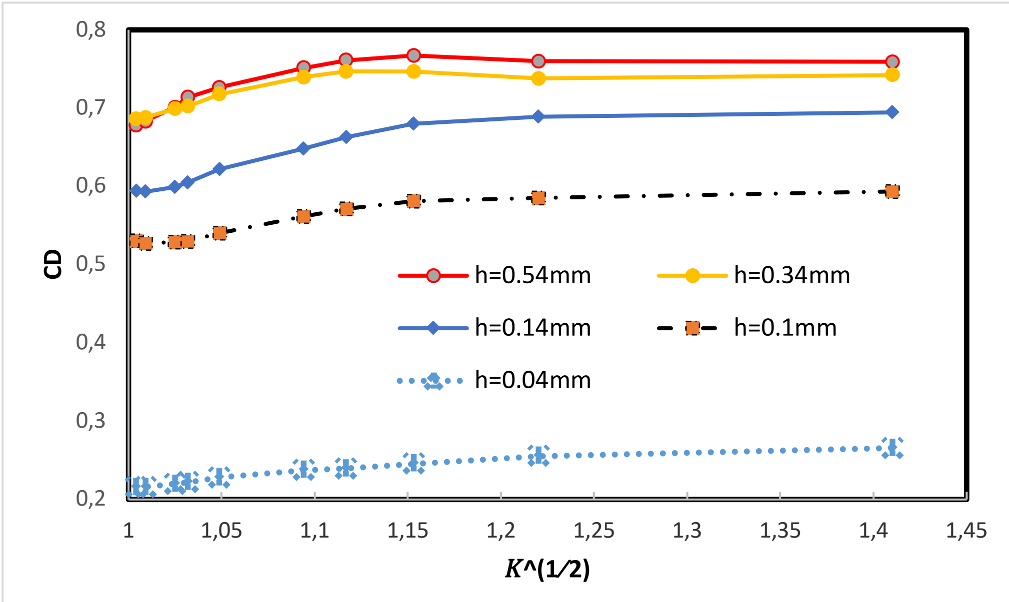

Fig. 4 Discharge coefficient versus cavitation number for different needle height

VI. Conclusion

In the current study, cavitation is investigated using Schnerr-Sauer cavitation model. RANS set of equations is also utilized to mathematically model flow behavior. Pressure and velocity are connected using SIMPLE algorithm. Pressure is solved using second order method, momentum equation is solved using second order upwind method.

The flow is verified using previous experimental study done by Winklfhofer. Then flow behavior inside diesel injector nozzle is obtained. Contour of velocity in the mid-plane of the orifice showed that the highest value of pressure mostly occurs at the end of needle before flow enters the orifice. Moreover, it was found that in all needle heights, by increasing cavitation number, discharge coefficient increases as well.

References

[1] Som, S., et al., *Investigation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector*. Journal of Engineering for Gas Turbines and Power-Transactions of the Asme, 2010. **132**(4).

[2] Shneider, M. N. and M. Pekker, *Classic cavitation*. Liquid Dielectrics in an Inhomogeneous Pulsed Electric Field: Dynamics, Cavitation and Related Phenomena, 2nd Edition, 2020.

[3] Stuppioni, U., et al., *Computational Fluid Dynamics Modeling of Gaseous Cavitation in Lubricating Vane Pumps: An Approach Based on Dimensional Analysis*. Journal of Fluids Engineering-Transactions of the Asme, 2020. **142**(7).

[4] Sun, T. Z., et al., *Numerical Investigation of Unsteady Cavitation Dynamics over a NACA66 Hydrofoil near a Free Surface*. Journal of Marine Science and Engineering, 2020. **8**(5).

[5] Sagar, H. J., et al., *Experimental and numerical investigation of damage on an aluminum surface by single-bubble cavitation*. Materials Performance and Characterization, 2018. **7**(5): p. 985-1003.

[6] Zeidi, S. and M. Mahdi, *Investigation the effects of injection pressure and compressibility and nozzle entry in diesel injector nozzle's flow*. Journal of Applied and Computational Mechanics, 2015. **1**(2): p. 83-94.

[7] Zeidi, S. M. J. and M. Mahdi, *Evaluation of the physical forces exerted on a spherical bubble inside the nozzle in a cavitating flow with an Eulerian/Lagrangian approach*. European Journal of Physics, 2015. **36**(6).

[8] Payri, F., et al., *The influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzles*. Fuel, 2004. **83**(4-5): p. 419-431.

[9] Payri, R., et al., *Critical cavitation number determination in diesel injection nozzles*. Experimental Techniques, 2004. **28**(3): p. 49-52.

[10] Jia, M., et al., *Numerical simulation of cavitation in the conical-spray nozzle for diesel premixed charge compression ignition engines*. Fuel, 2011. **90**(8): p. 2652-2661.

[11] Giannadakis, E., M. Gavaises, and C. Arcoumanis, *Modelling of cavitation in diesel injector nozzles*. Journal of Fluid Mechanics, 2008. **616**: p. 153-193.

[12] Kubo, M., T. Araki, and S. Kimura, *Internal flow analysis of nozzles for DI diesel engines using a cavitation model*. Jsae Review, 2003. **24**(3): p. 255-261.

[13] Reid, B., et al., *An optical comparison of the cavitation characteristics of diesel and bio-diesel blends in a true-scale nozzle geometry*. International Journal of Engine Research, 2013. **14**(6): p. 622-629.

[14] Bastawissi, H.A.E. and M. Elkelawy, *Computational Evaluation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector*. Sae International Journal of Engines, 2012. **5**(4): p. 1605-1616.

[15] Andriotis, A. and M. Gavaises, *Influence of Vortex Flow and Cavitation on near-Nozzle Diesel Spray Dispersion Angle*. Atomization and Sprays, 2009. **19**(3): p. 247-261.

[16] Andriotis, A., M. Gavaises, and C. Arcoumanis, *Vortex flow and cavitation in diesel injector nozzles*. Journal of Fluid Mechanics, 2008. **610**: p. 195-215.

[17] Winklhofer, E., et al. *Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions*. in *Proceedings of the ILASS-Europe conference, Zurich*. 2001.

[18] Schnerr, G. H. and J. Sauer. *Physical and numerical modeling of unsteady cavitation dynamics*. in *Fourth international conference on multiphase flow*. 2001. ICMF New Orleans.