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

Computational Study Of Induced In Nozzle Cavitation And Its Effects On Nozzle Flow For A Heavy Duty Diesel Engine

Ali Javaid[1], Rizwan Latif[2], Imran Shafi[3], Muteeb ul Haq[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

For the combustion process to be controlled inside a heavy duty diesel engine, fuel atomization is thought to be the best tool. Engine efficiency and emissions reduction can only be controlled by controlling combustion. Atomization of fuel or breaking down the fuel into tiny droplets to increase the surface area which will result in for complete combustion. This occurs due the cavitation effect occurring inside the injector nozzle. Cavitation can be said to be one of the most important factor in the combustion process due to the fact that it has a strong effect on the nature of the spray injecting into the chamber. The phenomenon will lead to complete combustion process where as if the fuel will is not burnt completely, the unburnt hydrocarbon will pollute the atmosphere moreover the efficiency of the engine will be compromised which is indeed not desired. In diesel injectors the main role of atomization can be observed due to cavitation and the flow behaviour is significantly affected by the cavitation. Cavitation affect will determine the flow entering into the combustion chamber for combustion. The problem to limit the study of fuel injectors is that the fuel injectors are very small and they are operated at very high pressure. This study carried out will help us in better understanding of cavitation process occurring and the effects of cavitation on the flow. Different types of nozzles were studied. These parametric study will help us to develop better injectors in future and can improve engine efficiency. The process of cavitation has strong effects on different parameters of the flow which include mass flux, momentum flux, and the velocity of the flow entering into the chamber. Different geometries were modelled on Solid works and were imported in Ansys for simulating the effects of cavitation. Results obtained by a wide range of simulations can be analysed for better engine performance and low emission.

Introduction

Heavy duty diesel engines are mostly used in non-commercial applications. These engines have a role to play in many areas such as electrical power generators, heavy duty vehicles etc. The literature found on these engines is very less so aim of the research is mainly on heavy duty diesel engines. We have seen that in heavy duty diesel engines the emissions are very high. Black suit is clearly observed from a very large distance. No control over the emissions is leading an unfriendly environment for a healthy living. To reduce emissions and making the environment clean and safe we need to focus on optimized fuel injection into the combustion change for complete combustion. To achieve efficient combustion we have to achieve good spray characteristics. Inside the injector nozzle the cavitation process takes place and fuel is break down into tiny droplets which will help in more efficient combustion as the surface area is increased.

Many researches have accomplished to anticipate the cavitation effect [4, 7, 8, 10], for planar nozzles. To aid the features of the problem, analysis on cylindrical nozzle are side by side [1, 3]. With all this knowledge this work targets the cavitation in a 3D injector nozzle with different pressure differences for diesel fuel. The cavitation is required to a certain limit beyond that limit cavitation causes choking to the fuel which will again lead to the incomplete combustion. The combustible is at very high pressure which is to be injected into the ignition chamber due to which the pressure gradients are very high and the nozzle hole is very small. In commercial engines the diameter is about 10μm whereas length is 1mm. In case of heavy duty diesel engine nozzle diameter is 300μm and length is 1.4mm. Figure 1 shows the 3D view of the injector. Cavitation phenomenon will determine the nature of the spray. It has strong effect on the spray entering into the combustion chamber.

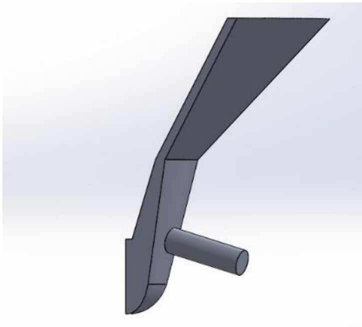

Figure 1. 3D view of the 1/6 of injector nozzle modelled in Solid works

The study was carried out on the 6-hole injector and all the simulations were performed for one hole as the geometry is axisymmetric along y-axis. This simplification saved a lot of computational power as the geometry was small so the meshing was less leading to less number of nodes on which the calculations were performed. Total time consumed for simulation was reduced.

Theoretical Background

The change of phase from liquid to vapours occurs. The reason behind is the pressure falls down to the saturated vapour pressure, where the liquid phase changes to vapour phase and this is called cavitation. In injector nozzles of diesel engines cavitation appears inside the injectors. The pressure gradient, change in flow direction and cross sectional area are the three factors which separate the fuel from the wall establishing ‘vena contracta’. Due to the vena contracta appearance of recirculation zone is observed in-between the boundary and vena contracta. This is the region where cavitation will occur, where static pressure has fallen below the vapour pressure P_v [2]. Figure 2 shows the simple sketch of the phenomenon [5]

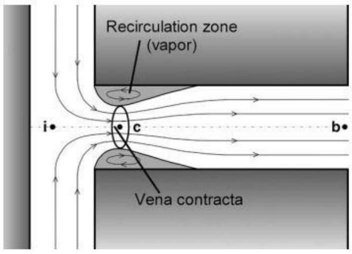

Figure 2. Recirculation zone and vena contracta formation

In the study some of the parameters are fixed and rest are varied to see the variation in outcome. P_i will be the injection pressure and P_b will be back pressure. At two different injection pressures we will vary the back pressures to demonstrate the spray behaviour, mass flow rate and velocity.

From mass flow rate and momentum we can conclude the important results. We can define them as follows

$$\dot{m}_f = \int_{A_o} u\rho dA \tag{1}$$

$$\dot{M}_f = \int_{A_o} u^2\rho dA \tag{2}$$

Due to the cavitating conditions the flow will have a random velocity profile. The complex flow is simplified that the fuel passes through an area which is termed as effective area A_{eff} assuming at a constant velocity U_{eff}, to maintain the momentum flux. We can define them as

$$U_{eff} = \frac{\dot{M}_f}{\dot{m}_f} \tag{3}$$

$$A_{eff} = \frac{\dot{m}_f^2}{\rho_l \dot{M}_f} \tag{4}$$

We can calculate the velocity at exit by using Bernoulli's equation by using pressure difference at the inlet and the outlet, we can assume that inlet velocity is negligible.

$$U_{th} = \sqrt{\frac{2\Delta P}{\rho_l}} \tag{5}$$

After calculating the above we can theoretical mass flow from theoretical velocity by keeping the cross section and density constant. From these we can get the non-dimensional coefficients. The coefficients are coefficient of discharge C_d, coefficient of velocity C_v and momentum coefficient C_M.

$$C_d = \frac{\dot{m}_f}{A_o \rho_l U_{th}} \tag{6}$$

$$C_v = \frac{U_{eff}}{U_{th}} \tag{7}$$

$$C_M = \frac{\dot{M}_f}{A_o \rho_l U_{th}^2} \tag{8}$$

$$K = \frac{P_i - P_v}{P_i - P_b} \tag{9}$$

The relation in equation 9 is used for calculating cavitation number. Under cavitating conditions cavitation number is very useful where as in non-cavitating conditions Reynolds number is handy. Figure 3 [9] represents the cavitation number K vs coefficient of discharge.

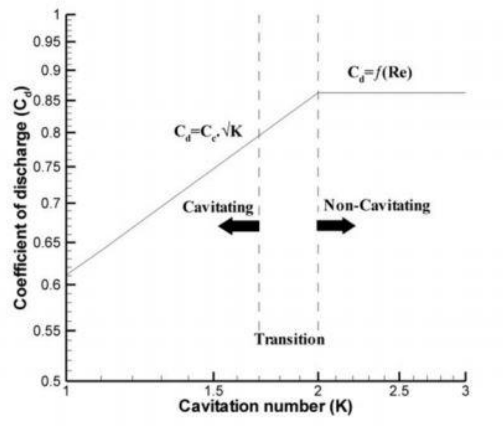

Figure 3. Plot between Coefficient of discharge vs cavitation number

Methodology and CFD Approach

Ansys fluent was used for the simulation which is basically a CFD tool and can solve a wide range problems. Multiphase flow are also dealt in fluent which is the basic phenomenon occurring in the injector. The scheme used in this study was validated first according to the numerical scheme used in the research [9]. The same conditions were taken into account for the validation purpose, after that the scheme was applied on our injector and desired conditions were applied to get the results. Later we will see the trends of the graph for further verification.

After modelling of the injector the next step and most critical step is of meshing, if the meshing is poor the results will vary drastically. Computational cost increases with the increase in mesh size and more time will be consumed to obtain the results but somehow we have to compromise and play with time and accuracy till our desires. Mesh independency was assured before moving onto the next step. Figure 4 shows the meshing of the injector.

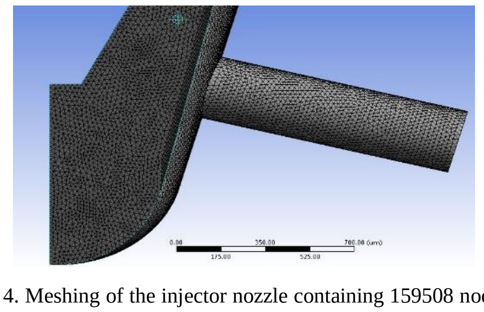

Figure 4. Meshing of the injector nozzle containing 159508 nodes and 665147 elements.

After meshing boundary conditions and fluid properties were added to the setup. Table 1 [6] shows the fluid (diesel) properties passed down in the simulations whereas Table 2 flashes the pressures sets worn for simulations.

Diesel Fuel		
Property	**Liquid Phase**	**Vapour Phase**
Density in kg/m^3	832	13.61E-2
Viscosity in kg/ms	0.0065	5.953E-6
Compressibility in s^2/m^2	5E-7	2.5E-6

Table 1. Data of diesel fuel used in the simulations

Pressure sets (MPa)								
P_i	P_b							
200	5	10	15	20	25	30	35	40
50	1	2.5	5	7.5	10	12.5	15	17.5

Table 2. Different pressure sets used in simulations

Proceeding ahead, Schnerr and Sauer cavitation model was pre owned in addition with the realizable k-ε turbulence model for obtaining the results. Rayleigh-Plesset equations are the main fundamental equation on which the cavitation model stands on, in this model, how the development of a solitary vapour bubble happens, is described.

Results and Discussion

Matlab code was developed for calculating the theoretical values and numerical values were input to the code to visualize the mass flow rate, velocity and non-dimensional coefficients by plotting them which can be of most importance in predicting the behaviour of the spray. Disappearance of cavitation showed a loss in mass flow rate depicted in figure 5. The graphs of non-dimensional coefficients with Reynold number and cavitation number also show a change in trend when the cavitation phenomenon vanishes in case of injection pressure 50 MPa. While in second case when injection pressure is 200 MPa the cavitation appears in all the back pressures given in table 2. Resulting in a gradual change in all the graphs.

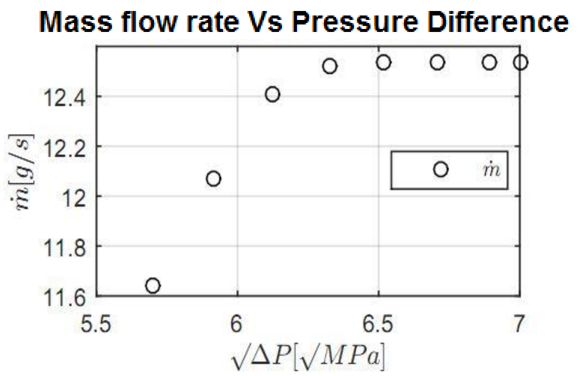

Figure 5. Graph between Mass flow rate and pressure difference under root with P_i = 50MPa

Figure 5 representing the mass flow rate remains constant till the squre root of pressure diffrence of 6.25$\sqrt{MPa}$ after that the cavitation no longer exists and the it decreases as the back pressures is increasing.

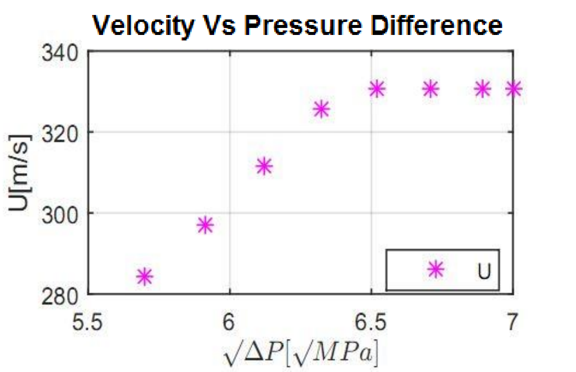

Figure 6. Graph between velocity and pressure difference under root with P_i = 50MPa

In figure 6 velocity is following the same trend as in figure 5, the mass flow rate experiences. Both figures 5 and 6 are effected sharply. Reason behind, elemination of cavitation phenomenon.

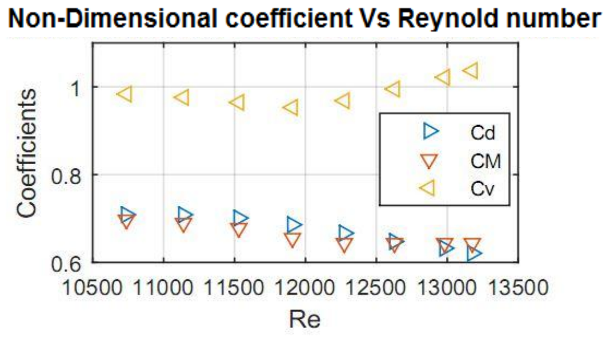

Figure 7. Graph between Non-dimensional coefficients and Reynold number with P_i = 50MPa

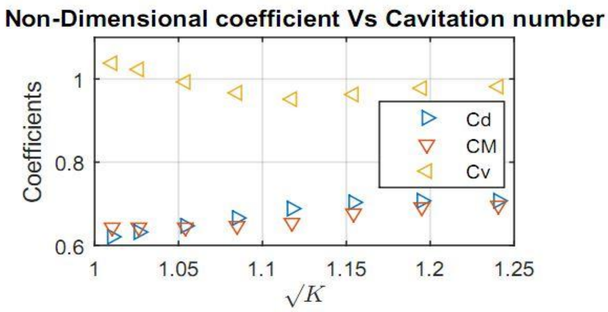

Figure 8. Graph between Non-dimensional coefficients and cavitation number under root with P_i = 50MPa

Figure 7 and figure 8 represent the graphs of non-dimensinaol coefficients, as desribed in equation 6,7 and 8, Reynold number and cavitation number. Both of the graphs practice the same trend change behaviour when the cavitation phenomenon ends.

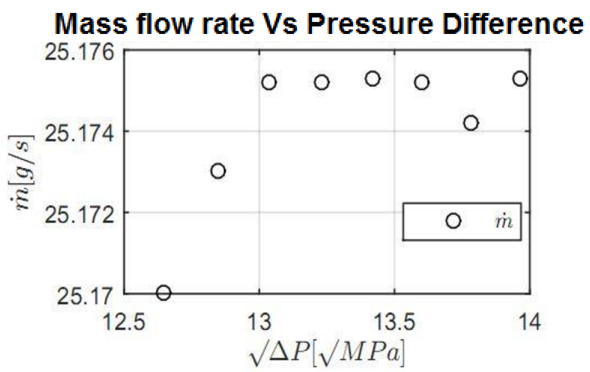

Figure 9. Graph between Mass flow rate and pressure difference under root with P_i = 200MPa

Mass flow rate almost remanis constant as shown in figure 9 as cavitation effect does not elimnates, the figure 5 has a reduction as the cavitation effect is no longer present in it. Increase in back pressure results in slight reduction in mass flow rate.

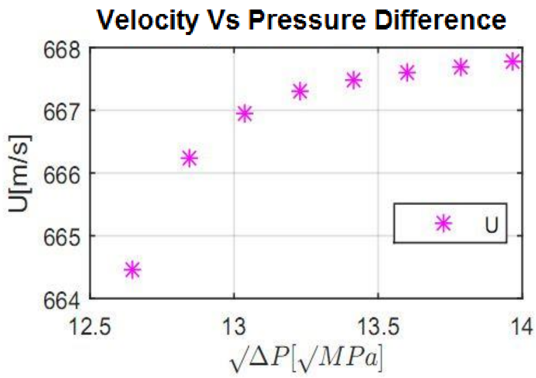

Figure 10. Graph between velocity and root of pressure difference under root with P_i = 200MPa

As described earlier the velocity follows the same trend as mass flow rate in figure 10. Mass flow rate in figure 9 was almost constant, so does the velocity. Slight but more reduction as compared to mass flow rate was shown in figure 9 by the velcoity as the outlet pressures is increased.

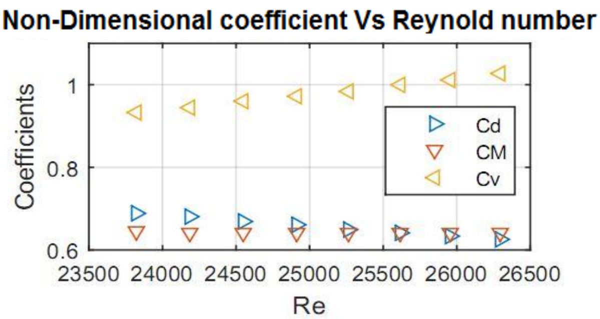

Figure 11. Graph between Non-dimensional coefficients vs Reynold number with P_i = 200MPa

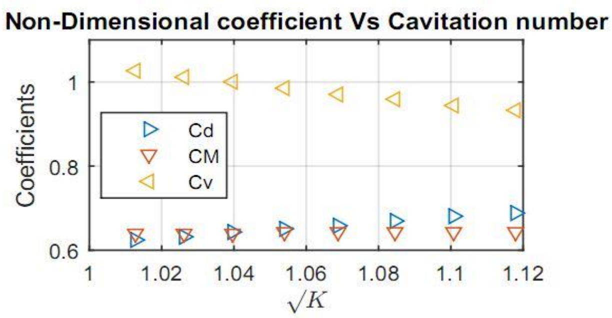

Figure 12. Graph between Non-dimensional coefficients and cavitation number under root with P_i = 200MPa

In figure 11 and figure 12 the trend for non-dimensional coefficients remain the same as the cavitation effect is still dominant there. Same behaviour is expect at high inlet pressure if the cavitation number goes into the transition or non-cavitation range as shown in figure 3.

Mass flow rate is effected by the increase in back pressure and it remains constant till the cavitation is experienced. Same trend is followed by the velocity. We need the velocity to be increased and from the graphs it is clear that the velocity does increases when the cavitation is there. Coefficient of discharge decreases as the increase in K number. If computational power is available then mesh can be refined more or different mesh structures study can be carried out. Non-conventional geometries can be entertained for predicting the internal flow and cavitation effect. Other geometry parameters can be varied i.e. length, nozzle angle, diameter etc. and keeping the pressure set fixed.

Conclusion

The conclusion for the study can be summarized as follows:

- Mass flow rate is constant as long as there is cavitation. In figure 7 velocity coefficient trend changes as the cavitation vanishes.
- Velocity decreases gradually while increasing the back pressure but when cavitation disappears then a sharp reduction is observed in the velocity.
- Mass flow rate and exit velocity experience an increase when inlet pressure is set to 200MPa.

References

[1] Andriotis, A., & Gavaises, M. (2009). , Influence of vortex flow and cavitation on near nozzle diesel spray dispersion angle. Atomization and Sprays, 19(3), 247-261.

[2] Bergwerk, W. (1959). Flow Pattern in Diesel Nozzle Spray Holes. Proceedings of the Institution of Mechanical Engineers, 173(1), 655-660.

[3] Ganippa L. C., Bark G., Andersson S., Chomiak J., Comparison of cavitation phenomena in transparent scale-up single-hole diesel nozzles, in: Proc. CAV2001, 2001, p.A9.005.

[4] Mishra, C., & Peles, Y. (2005). Cavitation in flow through a micro-orifice inside a silicon microchannel. Physics of Fluids, 17(1), 013601.

[5] Payri, R., Garcia, J., Salvador, F., & Gimeno, J. (2005). Using spray momentum flux measurements to understand the influence of diesel nozzle geometry on spray characteristics. Fuel, 84(5), 551-561.

[6] Peng F., Weller Henry, Nordin Nikals, Modelling injector flow including cavitation effects for diesel applications, FEDSM 2007.

[7] Sou, A., Hosokawa, S., & Tomiyama, A. (2007). Effects of cavitation in a nozzle on liquid jet atomization. International Journal of Heat and Mass Transfer, 50(17-18), 3575-3582.

[8] Sou, A., Tomiyama, A., Hosokawa, S., Nigorikawa, S., & Maeda, T. (2006). Cavitation in a Two-Dimensional Nozzle and Liquid Jet Atomization (LDV Measurement of Liquid Velocity in a Nozzle). JSME International Journal Series B, 49(4), 1253-1259.

[9] Sukanta Rakshit, High speed flow simulation in fuel injector nozzles.

[10] Winklhofer E., Kull E., Kelz E., Morozov A., Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions, ILASS-Europe 2001, Zurich, 2–6 September, 2001.