Impact of Cavitation on Velocity Distribution

Erik Arthur
Department of Civil and Environmental Engineering
Morgan State University, Baltimore, Maryland, USA

Abstract:
The flow inside rectangular shape nozzle is simulated in which the rectangular shape orifice was also considered with incline. CFD (Computational Fluid Dynamic) approach is used in this study along with finite volume scheme. The two-phase flow is considered homogenous and k-ε turbulent model is used to consider velocity and pressure fluctuations. Schnerr and Sauer cavitation model is used as the approach for evaluating the cavitation phenomena. Validation was done with comparing the velocity profile obtained thorough the simulation and previous experimental data obtained by Winklhofer. Effect of cavitation on velocity profile has also been stated.

Keywords: Cavitation, Multiphase Flow, Bubble Dynamic, Bubble Motion, Maxey Equation.

I. INTRODUCTION

Compression ignitions are affected enormously by spray process and atomization of the fuel. As technology improves for diesel engines, shape and efficiency of the orifices are changed and improved. Cavitation is the phenomena that occurs before atomization process [1-4]. Dynamic factors are in charge of controlling several parameters such as needle lift and injection pressure. The mentioned factor has been investigated thoroughly in the previous studies. Nozzle orifice and nozzle geometry are among the factors that were investigated at the beginning [5, 6]. The cavitation process was determined at the beginning thorough the codes in Fortran by coupling bubble dynamic equation and bubble motion equation. At the beginning of the mentioned studies for verifying bubble dynamic equation many approaches were used to discretize the mentioned equations in which the most significant part was solving it thorough higher accuracy which is so called higher order of equation. Sonoluminescent in the experimental labs were used to trigger the bubble so that it’s radius and bubble wall velocity could be determined one after another. Then mostly after this step which is used commonly for verification, bubble motion equation is added to the bubble dynamic equation [1, 2, 7-9]. Moreover, there has been several studies regarding to nature of the cavitation both experimentally and numerically. One of the most significant one was done by Winklhofer et al. [10] in which a rectangular shape nozzle was introduced in the experimental lab and then velocity profile and mass profile was obtained in a specific part of the orifice area which gave a very good insight in understanding the cavitation phenomena.

II. MATHEMATICAL EQUATION

In this study, the governing equation is based on single fluid approach of two-phase flow that considers the flow homogenous. Continuity and momentum equation of the mixture approach can be demonstrated as following[5, 6, 11-13]

\[ \frac{\partial}{\partial t} \left( \rho_m \right) + \nabla \cdot (\rho_m \vec{u}_m) = \rho_m \nabla \cdot \vec{v} \]  

\[ \frac{\partial}{\partial t} (\rho_m \vec{u}_m) + \nabla \cdot (\rho_m \vec{u}_m \vec{u}_m) = -\nabla p + \nabla \left[ \mu_m \left( \nabla \vec{u}_m + \nabla \vec{u}_m^T \right) \right] + \rho_m \vec{g} \]  

Then the mixture density and viscosity expressed as following

\[ \rho_m = \rho_r + \alpha_r \rho_v \]  

\[ \rho_v = \rho_0 \alpha_v \]  

Vapor volume fraction obtained based on number of bubbles per unit volume and bubble radius that can be shown as the following ralation

\[ \alpha_v = \frac{n_b}{3 \pi R_b^3} \]  

For obtaining the mass transfer between liquid and vapor a first order different equation is defined that considers density of the vapor and liquid separately with vapor volume fraction which expressed as following

\[ \frac{\partial}{\partial t} (\alpha_v \rho_r \rho_v) + \nabla \cdot (\alpha_v \rho_r \vec{u}_v \rho_v) - \frac{\rho_r \rho_v}{\rho} \frac{\partial \alpha_v}{\partial t} \frac{dR}{dt} \]  

Finally, the source term is defined as following

\[ \rho_v \frac{dR}{dt} \]  

In which equation 6 can be expanded as following

\[ \frac{\rho_r \rho_v}{\rho} \frac{dR}{dt} = \frac{\rho_r \rho_v}{\rho} \frac{dR}{dt} \left[ 1 - \alpha_v \right] \frac{3}{R_b} \sqrt{\frac{2 \left( \rho_v - \rho_l \right)}{\rho_l}} \]  

Bubble radius is also obtained from the following relation

\[ R_b = \left( \frac{\rho_r}{\rho_l} \right) \]  

Discharge coefficient is also defined mass row rate over cross sectional area of the orifice that is multiplied by two times of square root of liquid density and pressure difference

\[ C_d = \frac{\dot{m}}{2 \rho_l (P_l - P_{back})} \]  

By dividing the difference of inlet pressure and vaporization pressure over the pressure difference in inlet and outlet of the orifice, cavitation number is obtained

\[ \kappa = \frac{P_{in} - P_{in}}{P_{in} - P_{back}} \]  

II. GEOMETRY

In the present study, as figure 1 shows, the rectangular shape nozzle is modeled with a rectangular shape orifice. Inlet of the nozzle is 1 mm by 300 µm and outlet of the nozzle is 1 mm by 300 µm. The orifice inlet is 301 µm by 300 µm and the orifice outlet is 300 µm by 284 µm. Length of the orifice is 1 m and inlet radius of the orifice is 20 µm. Figure 2 shows the mesh
topology used in the current study which consist of a structured dominant mesh element. The orifice area is also meshed fully structured to improve the accuracy of the results in the orifice area in which flow changes drastically as pressure drop is noticeable and that’s why the mesh quality in the orifice region is very important.

III. VALIDATION

Table 1 shows physical properties of the carrying fluid used in this study which is diesel fuel and both liquid and vapor phase are mentioned. Table 2 shows the boundary condition used in this study in which for the inlet pressure inlet is 10 MPa, for outlet the pressure outlet boundary condition varies between 2 to 5 MPa. Table 3 is also showing the cavitation model used in the present study in which the model utilized is Schnerr-Sauer [14]. Number of bubble density is not fixed to a certain number in this study and a correlation is defined for that. Critical pressure which is not always equal to vaporization is are defined based on turbulent parameters and vaporization pressure.

Table.1. Physical properties of the carrying fluid

<table>
<thead>
<tr>
<th></th>
<th>Diesel (l)</th>
<th>Diesel (v)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>840</td>
<td>0.029</td>
</tr>
<tr>
<td>Viscosity (kg/ms)</td>
<td>0.0025</td>
<td>3.1×10⁻⁶</td>
</tr>
<tr>
<td>Surface tension (N/m)</td>
<td>0.02</td>
<td>-</td>
</tr>
<tr>
<td>Vaporization pressure (Pa)</td>
<td>870</td>
<td>-</td>
</tr>
</tbody>
</table>

Table.2. Boundary condition

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>Pressure inlet=10 MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet boundary condition</td>
<td>Pressure outlet= 2.5MPa</td>
</tr>
<tr>
<td>Intensity of turbulence</td>
<td>Turbulence intensity=0.16×Re⁻¹/₈</td>
</tr>
<tr>
<td>Turbulence length scale</td>
<td>0.7D</td>
</tr>
</tbody>
</table>

IV. RESULT AND DISCUSSION

Figure 3, shows velocity profile at two different pressure differences which are 55 bar and 67 bar. In which when the pressure difference is 55 bar, cavitation starts to appear and then as the pressure difference increases, length of the cavitating area increases until it fully covers the whole orifice area which is so called super cavitation. As can be seen in figure 3, the pressure was predicted with less than 5% average error compared to the experimental data previously obtained by Winklhofer et al. The deviation is more tangible near the wall since we are using wall function as resolving near the wall is very expensive computationally.

Figure 4 shows distribution of the pressure when the pressure difference is fixed to 60 bar. It can be seen that the velocity increases noticeably in the orifice area of the nozzle especially near the inlet radius of the orifice. The close up view also shows the mentioned phenomena in a better detail.
IV. CONCLUSION

The flow inside diesel injector nozzle is simulated using cfd (computational fluid dynamic) scheme. two pahse flow is considered as homogenous mixture. validation is performed using previous data obtained by winklhofer et al. velocity profile did perfectly match with previous velocity profiles obtained from the experimental study, moreover, fully structured mesh is used in the orifice area, finally, velocity contour when the pressure differenc is 60 bar is investigated and high amount of velocity at the orifice area is visualized.

V. REFERENCES


