**Influence of Diesel Nozzle Geometry on Cavitation Using Eulerian Multi-Fluid Method**

ZHANG Jun (张军)\(^1\), DU Qing (杜青)\(^{1,2}\), YANG Yanxiang (杨延相)\(^1\)

\(^1\) Internal Combustion Engine Research Institute, Tianjin University, Tianjin 300072, China; \(^2\) State Key Laboratory of Engines, Tianjin University, Tianjin 300072, China

© Tianjin University and Springer-Verlag Berlin Heidelberg 2010

**Abstract:** Dependent on automatically generated unstructured grids, a comprehensive computational fluid dynamics (CFD) numerical simulation is performed to analyze the influence of nozzle geometry on the internal flow characteristics of a multi-hole diesel injector with the multi-phase flow model based on Eulerian multi-fluid method. The diesel components in nozzle are considered as two continuous phases, diesel liquid and diesel vapor respectively. Considering that both of them are fully coupled and interpenetrated, separate sets of governing equations are established and solved for each phase. The geometric parameters mainly include the length and exit diameter of nozzle, the rounded radius at inlet of nozzle orifice and the angle between axis of injector and axis of nozzle orifice, and they are individually taken into account to analyze the impact on the cavitating flow in nozzle. The results show that the geometrical characteristics of nozzle have a strong influence on the volume fraction of diesel vapor in nozzle and the outlet flow velocity of injector. So cavitation in nozzle orifice should not be neglected for the in-cylinder fuel atomization process, especially for the primary break-up of liquid jet.

**Keywords:** diesel injection; cavitation; nozzle geometry; multi-phase flow; numerical simulation

Fuel injection plays an important role in the performance and emissions of internal combustion engine, especially for modern diesel engine and gasoline direct injection (GDI) engine. Direct injection engines exhibit high potential for the reduction of fuel consumption, and thus more and more automobile manufacturers start to use high-pressure fuel injectors. The purpose of high inlet pressure is to produce high injection velocity which results in an efficient atomization process with small and dispersed fuel droplets to enable rapid evaporation and traverse rapidly through the combustion chamber\(^{[1]}\). There is experimental evidence to show that cavitation within injector nozzle modifies the flow characteristics of nozzle exit and favors the atomization of fuel\(^{[2-5]}\).

Cavitation in nozzle orifice is desired to a certain degree because the collapse of cavitation bubbles influences the turbulence intensity towards the outlet of injector. Consequently, this enhances the atomization process of fuel in the combustion chamber. At the same time, the accurate prediction of cavitation zones in nozzle is quite necessary in order to avoid efficiently the erosion of the inner surface in nozzle orifice due to collapse of bubbles close to nozzle walls. However, it is usually very difficult to observe the flow state in nozzle for a real injector under operating conditions, therefore, the multidimensional numerical calculation, computational fluid dynamics (CFD) simulation, is an appropriate tool to provide a better understanding of 3D flow features inside and at the exit of injector nozzle.

The Eulerian multi-fluid method introduced in this paper can be used to simulate \(n\)-phase flows and different types of injector, such as Sac-type nozzle and VCO-type nozzle. In the present work, the 3D numerical simulation of a VCO-type diesel nozzle is performed to predict the cavitating flow in nozzle orifice and diesel is viewed as diesel liquid and diesel vapor, two continuous phases. In order to reveal the influence of nozzle geometry characteristics parameters on cavitation, several nozzles of different geometrical shapes are studied by applying unstructured grids, generated through a fully automated process. The calculated flow data at nozzle exit can be transferred into in-cylinder spray simulation as boundary conditions, especially for the primary break-up of liquid jet, and then it is possible to simulate the subsequent
processes of mixture formation, combustion and pollutant formation.

1 Eulerian multi-fluid model

The diesel liquid and diesel vapor are considered as continuous and interpenetrating between phases, in a statistical sense, coexisting with other fluid phases in time and space. This method is assumed that the transport equations, derived from the conservation laws of mass, momentum and energy, are valid for each phase. And also the coupling between phases is transferred through internal interfacial exchanges. The interfacial interaction models are obtained through the ensemble averaging operation of the governing equations.

1.1 Conservation equations

The model is based on the multi-fluid formulation for multi-phase flows and obtained through the ensemble averaging operation of governing equations\(^6\). It treats liquid, vapor, air, droplets, bubbles as continuous phases. Kunz\(^7\) and Carrica\(^8\) studied the numerical approach to the simulation of multi-phase flow. The implemented model can be applied to \(n\)-phase flows, and here it is reduced to a two-phase case because only diesel liquid and diesel vapor are considered. A separate set of conservation equations is solved for each phase and described as follows.

Mass conservation equation:
\[
\frac{\partial \rho_i \alpha_i}{\partial t} + \nabla \cdot \rho_i \alpha_i \mathbf{v}_i = \sum_{l=1, l \neq k} \Gamma_{kl} \tag{1}
\]
where \(\alpha_i\) and \(\mathbf{v}_i\) are the volume fraction and velocity of phase \(k\), respectively. \(\Gamma_{kl}\), which will be introduced in the following cavitation model in detail, represents the interfacial mass exchange between phases \(k\) and \(l\). The compatibility condition must be fulfilled.
\[
\sum_{i=1}^3 \alpha_i = 1 \tag{2}
\]

Momentum conservation equation:
\[
\frac{\partial \rho_i \alpha_i \mathbf{v}_i}{\partial t} + \nabla \cdot \rho_i \alpha_i \mathbf{v}_i \mathbf{v}_i = -\alpha_i \nabla p + \nabla \cdot \alpha_i \mathbf{v}_i (\mathbf{v}_i + \mathbf{T}_i) + \alpha_i \rho_i \mathbf{g} + \sum_{l=1, l \neq k} M_{kl} + \sum_{l=1, l \neq k} \mathbf{v}_l \Gamma_{kl} \tag{3}
\]
where \(\mathbf{T}_i\) is viscous shear stress; \(\mathbf{T}_i\) is Reynolds stress. \(M_{kl}\), which will also be explained in the following cavitation model, represents the interfacial momentum exchange between phases \(k\) and \(l\). Pressure is assumed identical for all phases,
\[
p_k = p \quad k = 1, 2 \tag{4}
\]

1.2 Turbulence model

Turbulence is always an important issue in generating cavitation, particularly low \(Re\) turbulence often encounters near the narrow wall of injector nozzle. For the momentum exchange, the implosion of spherical bubbles is considered as the source of the production of turbulence. The \(k - \varepsilon\) turbulence model is applied to the Eulerian multi-fluid method. Both turbulence kinetic energy and turbulence dissipation transport equations are solved for each phase.

Turbulence kinetic energy equation:
\[
\frac{\partial \rho_i \alpha_i k_i}{\partial t} + \nabla \cdot \rho_i \alpha_i \mathbf{v}_i k_i = \frac{1}{\alpha_i} \frac{\partial}{\partial x_j} \left( \alpha_i \rho_i C_{1k} k_i^2 \frac{\partial k_i}{\partial x_j} \right) - \alpha_i \rho_i \varepsilon_i + \sum_{j=1, j \neq k} \Gamma_{ji} \tag{5}
\]
where \(\Gamma_{ji}\) is the production term, 
\[
P_i = T_i^+ : \nabla \mathbf{v}_i \tag{6}
\]

Turbulence dissipation equation:
\[
\frac{\partial \rho_i \alpha_i \varepsilon_i}{\partial t} + \nabla \cdot \rho_i \alpha_i \mathbf{v}_i \varepsilon_i = -\frac{\partial}{\partial x_j} \left( \rho_i \alpha_i C_{1 \varepsilon} \frac{\varepsilon_i}{k_i} \frac{\partial k_i}{\partial x_j} \right) - \rho_i \alpha_i C_{1 \varepsilon} \varepsilon_i^2 + \sum_{j=1, j \neq k} \Gamma_{ji} \tag{7}
\]
In Eq.\((5)\) and Eq.\((7)\), \(C_{1k}\), \(C_{1 \varepsilon}\), \(C_{\mu}\) and \(C_{\varepsilon}\) are the closure coefficients.

1.3 Cavitation model

For a fluid, cavitation formed by large quantity of small spherical bubbles generated when the static pressure falls below vapor pressure due to the acceleration of fluid. Due to the sharp change in cross-section and flow direction, cavitation appears at the inlet of nozzle orifice, where boundary layer tends to separate from the inner wall of injector hole and the ‘vena contracta’ is established. Cavitation is modeled with a two-fluid method just like the above mentioned. The interfacial mass and momentum exchange between liquid phase and vapor phase are usually considered, while the interfacial heat exchange is not considered, which means that the temperature is a constant. The bubble dynamics is used to calculate the mass exchange rate from one phase to another. The mass exchange describes vapor generation, evaporation or condensation, and it is modeled by the following relation:
\[
\Gamma_i = \rho_i N^\sigma 4\pi R^3 \dot{R} = -\Gamma_i \tag{8}
\]
where \(N^\sigma\) is bubble number density, and \(R\) is bubble.