

Computational Evaluation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector

Hagar Alm El-Din Bastawissi
Huazhong University of Science and Technology

Medhat Elkelawy
Tanta University

ABSTRACT

The capabilities of various numerical models to accurately account for the onset and development of cavitation in diesel injector nozzles is assessed and evaluated. The numerical predictions of the models are computed, and are compared to measured experimental data and observations. The numerical predictions for actual diesel nozzle geometry have been validated with experimental measurements of the total vapor mass flow rate. This vapor flow is found to be developed along the nozzle length due to the nucleation of the cavitation bubbles inside the diesel injector. The cavitation inception criteria that is used for the quantitative cavitation calculations included vapor quality, voidage, cavitation kinetic energy and cavitation energy. The results indicate that the cavitation simulation model predicts a diffused and gradual vapor distribution inside the nozzle in agreement with the experimental data. The cavitation inception criteria used in current numerical modeling has matured to the level where it can usefully identify many of the characteristics of cavitation, and can significantly contribute to the optimization of cavitation production inside diesel injector. The calculations performed indicate that for the same model assumptions, such as the numerical implementation, discretization scheme and turbulence model, the predictions of the different cavitation sub-models are phenomenologically different. The calculations indicate that the Zwart-Gerber-Belamri model predicts a large void zone inside the injector delivery orifice and fail to capture the transition from incipient to fully developed cavitation, while the Singhal et al. model predicts a more diffused and gradual vapor distribution in agreement with the experimental model data. From the raw results, the values of the relevant parameters were computed, and the occurrence of cavitation was clearly identified. The results evidenced interesting differences in the permeability of the characteristics and strength of cavitation in nozzle under real diesel engine conditions.

CITATION: Bastawissi, H. and Elkelawy, M., "Computational Evaluation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector," *SAE Int. J. Engines* 5(4):2012, doi:10.4271/2012-01-1652.

INTRODUCTION

For diesel engines the fuel injection process and the characteristics of an issuing spray are some of the more important parameters in combustion and emission of pollutants [1,2,3,4,5]. One proven method to decrease both the particulate and nitrogen oxide emissions is to improve the performance of the fuel injection system. Modifications to improve the fuel injection system can reduce emissions dramatically [6,7,8]. Furthermore, under the injection conditions in modern diesel engines that have an injection pressure of up to 180 MPa, cavitation phenomena often occur in the fuel injector nozzles. When cavitation occurs vapor bubbles are introduced into the flow, which increases the maximum velocity in the nozzle. The flow velocity is

increased due to two reasons when the fuel is cavitating. First, if there is vapor along the wall, the liquid will have a slip condition boundary, thus allowing the velocity of the liquid to increase [9,10,11]. Second, cavitation inside the diesel injector nozzle leads to an increase of the spray cone angle which can improve the air fuel mixing process [12,13,14]. The fuel flow at the nozzle exit was characterized by measuring both the injection rate and the momentum flux for a wide range of typical rail and cylinder pressures experimentally. CFD calculations of the internal nozzle flow have shown that cavitation strings are formed in the areas where large vertical structures are present [15, 16]. The injection rate is found to be directly affected by the cavitation, and the injection nozzle flow will be choked when

this phenomenon occurs. Thus, the measurement of the momentum flux can be combined with the injection rate to derive the velocity at the nozzle exit [17, 18]. Also, the influence of nozzle geometry on the internal flow characteristics of a diesel injector has been previously conducted using a simulation analysis [19,20,21].

The physics of cavitation at the nozzle exit is strongly linked to the behavior of the liquid flow inside the injector. Cavitation may occur, depending on the injector geometry [22] and the flow, and if it occurs will have a strong influence on the spray velocity, spray angle and atomization [23, 24]. Furthermore, due to the high speed and high density of the liquid jet, traditional optical methods are limited regarding obtaining measurements in the dense liquid spray region. Therefore, the primary break-up region of the spray is difficult to observe experimentally, thus numerical models are not easy to validate. In addition, numerical difficulties arise that are linked to the high liquid volume fraction and the high fluid velocities (between 400 m s^{-1} and 600 m s^{-1}).

The study of the nozzle internal flow and the cavitation in actual injectors is challenging, both theoretically and experimentally. In order to obtain the change in atomization characteristics due to the cavitation in the nozzle flow, the visualization of the cavitation needs to be obtained by a visualization system, and the atomization characteristics such as Sauter mean diameter (SMD) and droplet mean velocity is determined by using a particle analyzer system [25,26]. In this regard, laser light-sheet photographs taken with a CCD camera were compared with photographs taken by the shadowgraph technique under the same injection conditions [27, 28]. As a result, the cavitation regions could be observed as thin objects which do not extend into the internal flow of the injection nozzle. In addition, even under higher injection pressures, no accumulation or foam of bubbles could be detected in the nozzle hole. This leads to the conclusion that there is an intact liquid core leaving the nozzle, even at high injection pressures. The flow in nozzles of different sizes has also been studied by using transparent nozzles of the same size as in diesel injectors, using refractive index matching. In a steady flow test facility, short-exposure video pictures of the flow were made at injection pressure up to 100 MPa [29].

Previous the numerical cavitation model results [30] indicated that cavitation modeling has achieved an advanced degree of sophistication. Cavitation modeling can now identify many of the vapor structures that exist in internal nozzle flows, and can ascertain their dependence on both nozzle design and flow conditions. The present numerical study aims at furthering of the cavitation processes that occurs inside diesel injector, especially under high-pressure injection conditions. Moreover, the previously discussed cavitation models have been utilized to analyze the strength

of cavitation phenomenon inside actual diesel injector nozzles. Comparisons between the obtained numerical results from the two main cavitation models have been performed through a newly introduced criterion issue. Also, a comparison between different turbulence modeling approaches to select which is more suitable for the injector nozzle simulation is introduced also in the present study. Finally, the criteria validation has been described in the last section.

MATHEMATICAL AND NUMERICAL DESCRIPTIONS

The recently developed CFD-tool FLUENT V6.3 software has been employed in the current study. The basic equations and physical models used for a single phase flow, which are the continuity, momentum and turbulence modeling equations. The continuity equation shown in Equation 1 is the general form of the mass conservation equation, and is valid for incompressible as well as compressible flows.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (1)$$

The source term in the right hand side S_m is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

The applied equation for conservation of momentum in an inertial (non-accelerating) reference frame is described by equation 2:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + \vec{F} \quad (2)$$

Where p is the static pressure, $\vec{\tau}$ is the stress tensor, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively.

The stress tensor $\vec{\tau}$ is given by equation 3:

$$\vec{\tau} = \mu \left[\left(\nabla \vec{v} + \nabla \vec{v}^T \right) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (3)$$

Where μ is the molecular viscosity, I is the unit tensor.

TURBULENCE MODELING APPROACH

Several turbulence models those are available to predict the flow and cavitation behavior for our CFD problem in diesel injector nozzle. Most of these models are derived from the standard k - ε model, and vary in complexity and robustness from the two-equation turbulence models to a more validated turbulence model. The standard, RNG, and realizable k - ε models have similar forms, with transport equations for k and ε . The major differences in the models are the method of calculating turbulent viscosity, in the turbulent Prandtl numbers governing the turbulent diffusion of k and ε and in the generation and destruction terms within the k equation. The standard k - ε model in **FLUENT** software assumes that the flow is fully turbulent and that the effects of molecular viscosity are negligible [31].

Transport Equations for the Standard k - ε Model

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_K \quad (4)$$

And,

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{K} + S_\varepsilon \quad (5)$$

In these equations, G_K represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constant. S_K and S_ε are user-defined source terms if needed.

The turbulent (or eddy) viscosity, μ_t , has computed by combining k and ε as follows;

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (6)$$

Where, C_μ is a constant.

The model constants have the following values:

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

Transport Equations for the RNG k - ε Model

RNG k - ε model has been derived using a rigorous statistical technique called renormalization group theory. It is similar in form to the standard k - ε model, but includes refinements for enhancing the accuracy, especially for the complex flows [32, 33]. The RNG k - ε model has a similar form to the standard k - ε model, that pre-described in equations 4 and equation 5.

The scale elimination procedure in RNG theory results in a differential equation for turbulent viscosity:

$$d \left(\frac{\rho^2 k}{\sqrt{\varepsilon} \mu} \right) = 1.72 \frac{\nu^*}{\sqrt{\nu^{*3} - 1 + C_\nu}} d \nu^* \quad (7)$$

In the high-Reynolds-number limit, the turbulent viscosity equation given by:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (8)$$

The RNG model in FLUENT provides an option to account for the effects of swirl or rotation by modifying the turbulent viscosity appropriately.

The main difference between the RNG and standard k - ε models lies in the additional term in the ε equation that is given by:

$$R_\varepsilon = \frac{C_\mu \rho \eta^3 (1 - \eta / \eta_0) \varepsilon^2}{1 + \beta \eta^3} \frac{1}{k} \quad (9)$$

Where,

$$\eta = S_\kappa / \varepsilon, \quad \eta_0 = 4.38, \quad \beta = 0.012$$

Transport Equations for the Realized k - ε Model

The realizable k - ε model [34] is a relatively recent development and differs from the standard k - ε model in two important ways: The realizable k - ε model contains a new formulation for the turbulent viscosity; a new transport equation for the dissipation rate ε derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The difference between the realizable k - ε model and the standard and RNG k - ε models is that C_μ is no longer constant, but it computed from the following equation:

$$C_\mu = \frac{1}{A_0 + A_s \frac{k U^*}{\varepsilon}} \quad (10)$$

Where,

$$U^* \equiv \sqrt{S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}} \quad (11)$$

And,

$$\tilde{\Omega}_{ij} = \Omega_{ij} - 2\varepsilon_{ijk} \omega_k, \quad \Omega_{ij} = \bar{\Omega}_{ij} - \varepsilon_{ijk} \omega_k$$

Where,

$\bar{\Omega}_{ij}$ is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity ω_k . The model constants A_0 and A_s are given by

$$A_0 = 4.04, A_s = \sqrt{6} \cos \phi$$

Where,

$$\phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W), W = \frac{S_{ij} S_{jk} S_{ki}}{\tilde{S}^3}, \tilde{S} = \sqrt{S_{ij} S_{jk}}, S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (12)$$

It is clear that, C_μ is a function of the mean strain and rotation rates, the angular velocity of the system rotation, and the turbulence fields (k and ε). C_μ may be shown to recover the standard value of 0.09 for an inertial sublayer in an equilibrium boundary layer.

Cavitation Model

With the multiphase cavitation modeling approach, the two-fluid, two-phase, cavitation model uses the viscous flow equations that govern the transport of mixture (liquid and vapor) a conventional turbulence model (k - ε model). Cavitation conditions, the liquid-vapor mass transfer evaporation and condensation is governed by the vapor transport equation (13) where α is the vapor volume fraction, v denotes the vapor phase, and R_e - R_c represents the mass transfer source terms connected to the growth and collapse of the vapor bubbles respectively.

$$\frac{\partial}{\partial t} [\alpha \rho_v] + \nabla \cdot [\alpha \rho_v \bar{V}_v] = R_e - R_c \quad (13)$$

In the cavitation model it is assumed that there are sufficient nuclei for the inception of cavitation. Thus, the primary focus is on proper accounting of bubble growth and collapse. The bubble dynamics equation can be derived from the generalized Rayleigh-Plesset equation (14), in which R_B is the bubble radius, and S is liquid surface tension coefficient, ρ_l is the liquid density, P_B is the bubble surface pressure, P is the local far-field pressure, ν_l is the liquid kinematic viscosity:

$$\Re_B \frac{D^2 \Re_B}{Dt^2} + \frac{3}{2} \left(\frac{D \Re_B}{Dt} \right)^2 = \left[\frac{P_B - P}{\rho_l} \right] - \frac{4\nu_l}{\Re_B} \Re_B - \frac{2S}{\rho_l \Re_B} \quad (14)$$

Neglecting the second-order terms and the surface tension force the, equation (14) can be simplified as shown in equation (15):

$$\left(\frac{D \Re_B}{Dt} \right) = \sqrt{\frac{2}{3} \left[\frac{P_B - P}{\rho_l} \right]} \quad (15)$$

The numerical models used for the comparative purposes are the Singhal et al. model [36] and Zwart-Gerber-Belamri model [37]. The full cavitation model developed by Singhal et al. accounts for all first-order effects such as phase change, bubble dynamics, turbulent pressure fluctuations and noncondensable gases. The Singhal model has the capability of accounting for multiphase (N-phase) flows or flows with multiphase species transport. The Zwart-Gerber-Belamri model assumes that all the bubbles in a system have the same size. In this model the total interphase mass transfer rate per unit volume is calculated using the bubble density numbers that are based upon the mass exchange rate of a single bubble. Additional information regarding the numerical equations are presented in Table 1, which lists the model equations. Where, F_{vap} , is the evaporation coefficient, F_{cond} , is the condensation coefficient, and f_v denotes the vapor volume fraction.

RESULTS AND DISCUSSION

Results of Comparison between Two Cavitation Models

In what follows, a comparison between two different cavitation models will be introduced in order to select the best model for simulating the cavitation in diesel injector nozzle. A new criterion has been applied for the quantitative comparison between both of the introduced cavitation models; the comparison criterion consists mainly of two quantities. The first one is known as vapor quality "x", which represents the ratio between the mass of vapor phase to