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 \text{s}^{-1} and 600 m \text{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.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot \left( \rho \vec{v} \right) = S_m
\]  

(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:

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

(2)

Where $p$ is the static pressure, $\tau$ 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 $\tau$ is given by equation 3:

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

(3)

Where $\mu$ is the molecular viscosity, $I$ is the unit tensor.
TURBULENCE MODELING APPROACH

Several turbulence models 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

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \\
+ G_k + G_b - \rho \epsilon Y_M + S_k
\]

And,

\[
\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_j} (\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] \\
+ C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{2\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{K} + S_\epsilon
\]

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. \( C_{1\epsilon} \) and \( C_{2\epsilon} \) are constant. \( S_K \) and \( S_\epsilon \) are user-defined source terms if needed.

The turbulent (or eddy) viscosity, \( \mu_t \), has computed by combining \( k \) and \( \epsilon \) as follows;

\[
\mu_t = \rho C_\mu \frac{k^2}{\epsilon}
\]

Where, \( C_\mu \) is a constant.

The model constants have the following values:

\( C_{1\epsilon} = 1.44, C_{2\epsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\epsilon = 1.3 \)

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 equation 4 and equation 5.

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

\[
d \left( \frac{\rho^2 k}{\sqrt{\epsilon \mu}} \right) = 1.72 \frac{\nu^*}{1 + C_\nu} \frac{d}{d \nu^*}
\]

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

\[
\mu_t = \rho C_\mu \frac{k^2}{\epsilon}
\]

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:

\[
R_\epsilon = \frac{C_\mu \rho \eta^3 (1 - \eta / \eta_0)}{1 + \beta \eta^3} \frac{\epsilon^2}{k}
\]

Where,

\( \eta = S_k / \epsilon, \quad \eta_0 = 4.38, \quad \beta = 0.012 \)

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 \( \epsilon \) 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 is computed from the following equation:

\[ C_μ = \frac{1}{A_0 + A_S} \left( \frac{k}{\varepsilon} \right) \]

(10)

Where,

\[ U^* = \sqrt{S_{ij} S_{ij} + \Omega_{ij} \Omega_{ij}} \]

(11)

And,

\[ \Omega_{ij} = \Omega_{ij} - 2 \varepsilon_{ijk} \omega_k, \quad \Omega_{ij} = \Omega_{ij} - \varepsilon_{ijk} \omega_k \]

Where

\[ \Omega_{ij} \]

is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity $\omega_k$. The model constants $A_0$ and $A_S$ are given by

\[ A_0 = 4.04, \quad A_S = 6 \cos \phi \]

Where,

\[ \phi = \frac{1}{3} \cos^{-1} \left( \sqrt{\frac{\partial W}{\partial S}} \right), \quad W = \frac{S_{,i} S_{,j} S_{,k}}{S_{,i} S_{,j} S_{,k}}, \quad S = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]

(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 $\varepsilon$). $C_μ$ 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 $\alpha$ is the vapor volume fraction, $\nu$ 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.

\[ \frac{\partial}{\partial t} \left( \alpha \rho \nu \right) + \nabla \left( \alpha \rho \nu \mathbf{V} \right) = R_e - R_c \]

(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, $\rho_l$ is the liquid density, $P_B$ is the bubble surface pressure, $P$ is the local far-field pressure, $\nu_1$ is the liquid kinematic viscosity:

\[ R_B \frac{D^2 R_B}{Dt^2} + \frac{3}{2} \left( \frac{D R_B}{Dt} \right)^2 = \frac{4 \nu_1}{R_B} \frac{P_B - P}{\rho_l} - \frac{2S}{\rho_l} \]

(14)

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

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

(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 “$\alpha_0$”, which represents the ratio between the mass of vapor phase to

\[ \frac{\partial}{\partial t} \left( \alpha \rho \nu \right) + \nabla \left( \alpha \rho \nu \mathbf{V} \right) = R_e - R_c \]
system total mass, as can be seen in equation (16). However, the quality is defined as a quantitative description of the usefulness of a vapor formed during the cavitation process. The second quantity is known as the voidage “*α*” which is a measure of the void “empty” spaces in a cavitating material, and is a fraction of the volume of voids over the total volume. The relationship between both the voidage and quality is deduced in equation (16) and (17) as the following:

\[
x = \frac{1}{1 + \left(1 - \alpha \right) \left( \frac{V_f}{V_g} \right)} \quad (16)
\]

\[
\alpha = \frac{1}{1 + \left(1 - x \right) \left( \frac{V_f}{V_g} \right)} \quad (17)
\]

Traditionally, the most significant describe dimensionless numbers which to correlate cavitating flow in diesel nozzles are the cavitation number, CN, and the Reynolds number, Re.

\[
CN = \frac{P_{\text{inj}} - P_{\text{out}}}{P_{\text{out}} - P_v}
\]

\[
Re = \frac{\rho_l UD}{\mu_l}
\]

Where, \(P_{\text{inj}}\) is the injection pressure, \(P_{\text{out}}\) is the downstream pressure, \(P_v\) is the fuel vapor pressure, \(U\) is the average flow velocity, and \(D\) is the nozzle hydraulic diameter. However, recent experimental studies of cavitating flows have shown that although the both of Reynolds and cavitation numbers are necessary, they are not necessarily sufficient to characterize the cavitating flow inside a diesel injector nozzle [38, 39]. It has been demonstrated that knowledge of both the cavitation and Reynolds numbers is not sufficient to provide a detailed description of the internal flow pattern, as it is observed to also depend on the flow boundary conditions.

The calculations performed by both of the above-mentioned cavitation models were compared, with both utilizing algorithms that were implemented properly at the identical diesel engine conditions. These conditions were 0.1 MPa back Pressure, 120 MPa injection pressure, and a system temperature of 400 K. Figure 1 illustrated the sac chamber of the actual nozzle configuration, and also illustrates the nozzle basic geometry at maximum needle lift. The injector nozzle boundaries for which boundary conditions must be specified are: two inlets the flow, one flow exit, two inner walls of nozzle, one outer wall of the nozzle. The boundary conditions for the two inlets are taken to be a specified volume flow rate normal to the boundary, with all wall boundaries taken as no-slip. The injection rate is assumed to be a steady stream of diesel liquid with constant flow rate. In the cavitation models applied, there is no heat transfer to/from the ambient medium.

### Table 1. Comparison between applied cavitation models.

<table>
<thead>
<tr>
<th>Criterion</th>
<th>The Singhal et al. model</th>
<th>Zwart-Gerber-Belamri model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Phase change rate</td>
<td>[ R = \frac{3\alpha \rho_f \rho_l}{\mathcal{R}_b} \sqrt{\frac{2}{3} \left( \frac{P_v - P}{\rho_l} \right)} ]</td>
<td>[ R = \frac{3\alpha \rho_f \rho_l}{\mathcal{R}_b} \sqrt{\frac{2}{3} \left( \frac{P_v - P}{\rho_l} \right)} ]</td>
</tr>
</tbody>
</table>

\[ fP > P_v \]

\[ R = \frac{F_{\text{con}}}{\sigma} \left( \alpha \right) \frac{2}{3} \frac{P_v - P}{\rho_l} \]

\[ \sigma \]

\[ fP > P_v \]

\[ R = \frac{F_{\text{con}}}{\sigma} \left( \alpha \right) \frac{2}{3} \frac{P_v - P}{\rho_l} \]

\[ \sigma \]
and the energy dissipation is negligible. Figure 1 also illustrates the three different positions along the nozzle length that were examined by our new proposed criterion. However, the achieved simulation results have been obtained by implementing the new criterion equations at three axial distances of 0.5mm, 1mm, and 1.4mm from the nozzle entrance, respectively. The magnified region of the nozzle is the most important part of the whole geometry, as it has the most influence on the cavitation formation and its devolvement history. Additional details about the generation of the computational grid are available in reference [30].

Figure 2 and Figure 3 illustrate the vapor volume fractions along the three different nozzle positions that are computed using both the Zwart-Gerber-Belamri model and Singhal et al, model. It is evident that both of the numerical cavitation models predict a large void zone inside the selected three positions along the injector nozzle. Nevertheless, at the third nozzle position near the nozzle exit, the Singhal, et al. model predicted a larger cavitation region and a higher concentration of vapor volume fraction. Regarding the Zwart-Gerber-Belamri model, the appearance of the vapor field computed is completely stable with negligible variations of the vapor phase behavior from location to location along the nozzle length. Nevertheless, the behavior of the flow predicted by the Singhal, et al. model appears to be more accurate when the experimental results are compared for the same conditions as presented in reference [40].

Figure 1. Schematic illustrating the three examined positions within a diesel nozzle.

Figure 2. Vapor volume fraction along three different nozzle positions when applying Zwart-Gerber-Belamri model.
Results of Comparison between Three Different k-ε Turbulence Models

The predicted values of Turbulence Kinetic Energy (TKE) along the three nozzle positions with the three different k-ε models are illustrated in Fig 4. The three models investigated give significantly different results for the TKE prediction along these selected positions. The TKE distribution in the nozzle is variable from one position to another, but each model shows a different distribution trend. In the case of applying RNG with the realized k-ε model, the TKE variation along the nozzle shows the expected distribution, similar to that encountered in a two-dimensional cavitating micro channel at high pressure. The computed TKE increases rapidly near the nozzle walls where the fluid shear stress is high and the pressure is lowest, and it keeps decreasing computed toward the center of the nozzle. It is evident that in this flow field, both the RNG and the realized k-ε turbulence models predict the TKE distribution almost identically in trends, but different in values. The RNG k-ε turbulence model clearly shows a strong distribution of the TKE along the nozzle radius, with a higher peak value that matches the pressure distribution presented in the literature. While the realized k-ε model adequately captures the extent and intensity of the TKE distribution, it does show some minor oscillatory behavior in pressure distribution along the nozzle radius. In spite of these differences, the general shape of the RNG and the realized k-ε turbulence models distributions resembles each other, with different corresponding peaks and troughs in the TKE plots. However, the standard k-ε model fails to predict the regions of high and low fluid shear along the diesel injector nozzle, and it produces an almost constant distribution of TKE. The standard k-ε model does predict some changes in TKE distribution, but the given distribution trend does not extend upstream or downstream as in the case of RNG, and realized k-ε model. It is evident that the standard k-ε turbulent model shows less satisfactory performance at the high velocity flows that are associated with large pressure differences.

The volume of diesel vapor produced in the cavitating nozzle was also computed using the three turbulence models. Fig 5 shows the computed volume of vapor produced at each nozzle position with the three different k-ε turbulent models. At the first nozzle position shown in Fig 5, the volume of fuel vapor produced by each of the three turbulence models is seen to be very similar to the experimental measurements that are available in the literature [24,25,26,27]. For the second nozzle position shown in Fig 5, the standard k-ε turbulent model predictions agree well with the other two models and with the experimental referenced data. At nozzle position 3, a zero volume of fuel vapor was predicted when applying the standard k-ε turbulent model. However, both the RNG k-ε, and realized k-ε turbulence models agree reasonably well in

Figure 3. Vapor volume fraction along three different nozzle positions when applying Singhal et al. model.
Figure 4. Computed Change in TKE along the nozzle at the three different positions for three $k$-$\varepsilon$ models.

Figure 5. Computed Change in Vapor Volume Percentage along the nozzle at the three different positions for three $k$-$\varepsilon$ models.
predicting the vapor phase present at the nozzle third position. The results of computations for the fuel vapor produced inside the nozzle show that the cavitation is concentrated near the nozzle walls and a double vortex structure starts forming inside the nozzle. This vapor propagates downstream towards the nozzle exit and the vapor presence more pronounced when fluid moves towards the nozzle exit. Therefore, the vapor concentration decreases when the flow leaves the wall, but the vapor that is formed recondenses when the flow moves away from the vena contracta region.

**Results of Effect of Injection Pressure on Cavitation and Nozzle Flow Parametrs**

During an injection event, the injection pressure generally ramps up reaching a peak value. In typical diesel engines, the injection pressure can vary from few hundred bars to peak values of 2500 bar or more and, therefore, it is important to examine the internal nozzle flow characteristics over this wide pressure range. Simulations were performed at three values of the injection pressure, which are 30, 60, and 120 MPa at constant chamber pressure of 2 MPa. Analysis of the effect of injection pressure on cavitation and nozzle flow performed by using a newly-introduced technique, this technique based mainly on the dimensionless numbers calculations. However, the whole system cavitation and Reynolds number does not provide a more insight of the cavitation and nozzle flow features in the nozzle. So that, the integral dimensionless numbers calculations concept have been performed in the present study to go through the nozzle flow and nozzle cavitation in details. Briefly, the diesel injector nozzle computational domain has been divided into some infinitesimal domains at different three locations along the nozzle geometry. At each infinitesimal domain the flow velocity, pressure, density, and turbulence important parameters have been calculated. Each of the infinitesimal domain considered as an independent area can be used in the calculation of the dimensionless parameters.

Fig.6 illustrates the infinitesimal cavitation number at three different positions along the nozzle with three different values of injection pressures; Fig.7 illustrates the infinitesimal Reynolds number at three different positions along the nozzle with three different values of injection pressures. Regarding Fig. 6, and 7 the basic features of the cavitating nozzle flow are shown clearly as the infinitesimal cavitation number decreases while the infinitesimal Reynolds number increases for the same nozzle position, and the vapor presence increases as well. From Fig 6 it is distinct that for a continuous spray condition (steady state), when the injection pressure is 30 MPa, there were some emerged vapor distributions at the vena contracta region, which is denoted by high infinitesimal cavitation number at nozzle first position. The vapor concentration decreased when the flow leaves the wall, but the vapor formed is recondensed as the flow moving down away from the vena contracta region and finally reached the fully developed diesel flow at the nozzle exit at position 3, where the infinitesimal cavitation number calculated is almost higher than its initial values. This effect was studied and traced through the results obtained when the injection pressure increased to reach 60 MPa except that the formed vapor phase continued to exist and increases its

**Figure 6. Description of the Infinitesimal cavitation number along the nozzle at the three different positions for three values on injection pressure.**
more than 30 MPa, exactly at 60 MPa, the continuous liquid phase decayed its volume fraction near the wall inside the nozzle due to cavitation and the formed vapor presents with a higher percentage until the nozzle exit region. Comparing the results for the infinitesimal cavitation number for the three positions, we can observe that the vapor presence is almost similar in 30 MPa and 60 MPa pressure. Moreover, the vapor phase existence at nozzle exit is more pronounced at higher injection 120 MPa. It is clearly shown that cavitation occurs at the entrance region of the injection hole and a larger cavitation region is established on the right-hand side where the pressure drop from flow deflection and the acceleration are most pronounced. Vapor content is reduced again towards the orifice exit section due to the pressure recovery which may causes condensation.

Also, when the injection pressure was 120 MPa and the vapor flow is still creeping to reach the nozzle exit, the vapor presence does not change. The infinitesimal cavitation number at injection pressure 120 MPa at the third nozzle position differs slightly from other infinitesimal calculation domain. In spite of, the vapor presence does not follow a certain constant distribution manner along a nozzle rather than it follows an oscillating manner. Obviously, we guess that a shock wave happened in the injection nozzle when the pressure increased to reach 120 MPa, however it is not evident still to now upon.

Linking Fig 6, and Fig 7, we can deduce that the increased fuel injection pressure not only affecting the pressure gradient near the nozzle inner walls, but also increase the velocity of the fuel flow when get out from the nozzle. However, it is not a rule that by increasing the injection pressure the vapor presence will be more pronounced.

**SUMMARY/CONCLUSIONS**

In the present study, the inception of fuel cavitation inside the diesel injector nozzle was evaluated for a steady-state continuous spray condition. A numerical simulation of the cavitation inside diesel injector nozzle was conducted using two different mathematical cavitation models, the Singhal, et al. model, and the Zwart-Gerber-Belamri model. It was verified that the results obtained using the Singhal, et al. model are more reliable for cavitation prediction than those obtained using the Zwart-Gerber-Belamri model. It was found that the Zwart-Gerber-Belamri model fails to capture the transition from incipient to fully developed cavitation, and that the value of vapor present in the nozzle is almost the same all along the nozzle length.

Three different turbulence models of different complexity were utilized to determine the accuracy of results. Based on the computational results, the RNG $k$-$\varepsilon$, and realized $k$-$\varepsilon$ models clearly performed better than the standard $k$-$\varepsilon$ Model when an almost constant TKE gradient was present. The
standard $k$-$\varepsilon$ delivered very poor results for the predictions of both the vapor present inside the diesel injector nozzle in cavitating conditions, as well as turbulent kinetic energy. The RNG $k$-$\varepsilon$, and Realized $k$-$\varepsilon$ models show an acceptable agreement with the referenced experimental data of turbulent kinetic energy profiles. In addition, the mentioned two models predict the location of separation and vapor production regions in nozzle in a good agreement with the experimental referenced data.

Simulation of 2D cavitating flow considering the nozzle flow as a homogenous mixture of liquid and vapor is used for modeling cavitation phenomena and an extended validation has been performed in the diesel injector nozzle. CFD results show a maximum deviation around 7% from the referenced corresponding experimental data. The present work provides improved criteria describing the cavitation and flow application inside the diesel injector nozzle. The improved criteria provide reliable quantitative description for the amount of vapor present inside the nozzle. The criteria embrace voidage, quality, infinitesimal cavitation number, and infinitesimal Reynolds number. However, when cavitation appears in the nozzle, infinitesimal Reynolds number and infinitesimal cavitation number are meaningless, and the phenomenon is better characterized by either the quality or voidage.

REFERENCES

6. Song, D, Tompkins, BT, Bittle, JA, Jacobs, TJ, Comparisons of NO emissions and soot concentrations from biodiesel-fuelled diesel engine, Fuel, 2012.01:004
30. El-DinMohmad Bastawissi, Hagar Alm (2009), Numerical Analysis of Flow and Cavitation in Real Size Diesel Engine Injector Nozzle, Master dissertation, Huazhong University of Science and Technology, China.
ACKNOWLEDGMENTS

The authors would like to introduce special thanks to Prof. Zhang Yusheng, School of Energy and Power Engineering-Huazhong University of Science & Technology, for his encouragement and support that he provided truly help and experienced suggestions for the progression and smoothness of the internship program. Prof. Zhang Yusheng cooperation is much indeed appreciated. Also, the authors acknowledge the financial supports of National Natural Science Foundation of China (Project 51176057) and the Specialized Research Fund for the Doctoral Program of Higher Education of China (20100142110081).

DEFINITIONS/ABBREVIATIONS

CFD - Computational Fluid Dynamics.
L/D - Nozzle length to diameter ratio.
TKE - Turbulence Kinetic Energy
RNG - Renormalization group theory of turbulence.
P_v - Liquid vapor pressure.
P_B - The bubble surface pressure.
t - Time.
T - Temperature.
TKE - Turbulence kinetic Energy
CN - Cavitation Number
Re - Reynolds Number
= - The stress tensor.
ρ - Density.
μ - Molecular viscosity
I - The unit tensor
U - The average flow velocity
D - The nozzle hydraulic diameter
ρg - The gravitational body force
F - The external body