DES Simulation of Asymmetrical Flow in a High Pressure Diesel Injector

Russell Prater\textsuperscript{1} \quad Yongsheng Lian\textsuperscript{2}

Mechanical Engineering Department
University of Louisville, Louisville KY 40292

Abstract

Delayed detached eddy simulations were performed to study the complex transient turbulent flow features in a high pressure diesel injector with a dual gain orifice at hover. The flow field was described by solving the compressible Navier-Stokes equations. The mass transfer between the liquid and vapor phases of the fuel was modeled using the Zwart-Gerber-Belamri equations. Our study showed that the detached eddy simulation captured high frequency fluctuations in the flow and predicted high lateral force amplitudes than RANS simulation. The DES simulation revealed vortical flow structures in the sac which are responsible for the lateral force on the needle and the hole-to-hole flow variation. The transient motion of the vortical structure also affected vapor formation variations in spray holes. Further analysis showed that the rotational speed of the vortical flow structure is proportional to the lateral force magnitude on the lower needle surfaces.

Key Words: DES, vortical flow, injection system, asymmetrical flow

Nomenclature

\begin{itemize}
  \item Ca \quad Cavitation number, \quad \frac{P_{inlet} - P_v}{P_{inlet} - P_{outlet}}
  \item D \quad Diameter
  \item F_e, F_c \quad Evaporation, condensation coefficient
  \item K \quad K factor
  \item L \quad Length
  \item P \quad Pressure
\end{itemize}

\textsuperscript{1} Ph.D candidate
\textsuperscript{2} Associate professor, yongsheng.lian@louisville.edu
**Introduction**

Rising concerns about energy efficiency and environmental sustainability require engineers to look for new strategies that can maintain extremely low emissions levels and high indicated efficiencies [1]. One such strategy is the development of modern fuel injection system which nowadays can operate over periods as short as a millisecond while delivering precisely metered fuel to the combustion chamber [2, 3]. The injection system plays a critical role in reducing emissions during operation but the underlying mechanism is complex. For diesel engines the emission formation rates are largely determined by the local temperature during combustion [4], which in turn depends on the spray pattern in the combustor. Early studies have shown that characteristics of the internal nozzle flow have a strong impact on the spray pattern and its characteristics [5-7] especially when cavitation occurs in the spray nozzles of high pressure diesel engines.

It is well known that cavitation is beneficial to spray dispersion because cavitation can increase spray atomization [8] and hence promote better mixing of fuel and oxygen in the combustion chamber [9-11]. Low speed experimental studies by Chaves [8] and Hiroyasu [12]
based on real-sized injectors showed cavitation inception point and the size of the cavitation was influenced by injector geometry. But experimental study of real-sized injectors remains challenging due to the extremely small geometry of the spray holes [13]. The use of scale-up nozzle is problematic because the cavitation structures in the scale-up nozzles different from real-sized ones [14] Numerical studies offer an alternative approach. The effect of injector geometry on cavitation was confirmed by numerical studies [15-17]. Other numerical studies [18, 19] showed that the region of significant cavitation shifts from the top of the orifice to the bottom as the needle position changes from fully open to nearly closed. The presence of cavitation can lead to a Kelvin-Helmholtz instability throughout the entire spray hole length [20] and contribute to the fluctuating flow characteristics.

Recently, high speed x-ray experiment showed that the motion of injector needle can significantly alter the flow pattern into the sac and affect the evolution of cavitation zones within a diesel injector [21]. Kilic et al. [22] showed that when the needle was off-center there were large variations in the flow velocity through different holes which led to asymmetrical spray penetration, i.e., hole-to-hole variation. Hole-to-hole variation can lead to degraded combustion performance and increased emissions. In severe situations it can result in spray hole damage and engine failure.

While the lateral needle motion has been observed and its damage is well documented, the underlying causes are not clear. It is therefore important to understand the problem so effective measures can be taken to prevent or remedy the needle lateral motion. In this paper we numerically study a generic high pressure diesel injector aiming to identify the cause of the needle lateral motion. Simulations were performed using the detached eddy simulation (DES). For comparison purpose, RANS based simulations were also performed. In the following we first present the numerical method which includes the detached eddy simulation and cavitation model. Then we provide the computational setup and sensitivity analysis, which is followed by numerical results and discussions.

**Numerical Method**

For the numerical simulations the flow field is described by the following continuity equation and momentum equations
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \]  
\[ \frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla P + \nabla \cdot \left[ \mu \left( \nabla \vec{u} + \nabla \vec{u}^T \right) \right] \]  

where \( \rho \) is the density, \( \vec{u} \) is the velocity vector, \( P \) is the pressure, \( \mu \) is the viscosity. Simulations were based on ANSYS Fluent Version 14.5. The SIMPLEC algorithm \([23]\) was used to couple the pressure the velocity fields. The second-order upwind scheme was used for the convection terms and the second-order central differencing scheme was used for the diffusion terms. A second-order implicit temporal discretization scheme was used for the time derivatives.

**The k-\( \varepsilon \) Turbulence Model**

We used the realizable k-\( \varepsilon \) formulation of Shih et al. \([24]\) as the turbulence closure. This model differs from the standard k-\( \varepsilon \) formulation \([25, 26]\) in that it derives the dissipation rate equation from the mean-square vorticity fluctuation. Constraints for the strain rates were derived from the application of rapid distortion theory. Turbulent realizability principles were applied to impose a requirement of non-negativity on the turbulent normal stresses and produce constraints for the coefficients within the Reynolds stress equations that limit the situations under which the model will produce non-physical results. The transport equations for \( k \) and \( \varepsilon \) are:

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon - Y_M + S_k \]  
\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon \mu}} + S_\varepsilon \]  

here \( Y_M \) is the contribution of the fluctuating dilation in compressible turbulent flow to the overall dissipation rate, \( G_k \) governs the generation of turbulence kinetic energy due to mean velocity gradients, \( S \) represents source terms within the flow field and \( \mu_t \) is the eddy viscosity. The realizable model provides superior performance compared with the standard k-\( \varepsilon \) model for flows involving separation and recirculation, both are encountered in our simulation\([24]\).
Delayed Detached Eddy Simulation

Because RANS based simulations often under-predict the maximum amplitudes in force fluctuations and miss high frequency fluctuations in the forces [27] and transient fluctuations of the vapor formation [28], a high fidelity model is required to accurately capture force magnitudes and transient flow features. For that purpose we use the delayed detached eddy simulation (DDES) [29][30].

DDES uses the RANS model presented above to resolve the flow near the solid boundaries along with a large eddy simulation (LES) [31] to resolve the flow in the far field. The LES equations are produced by filtering the general flow equations, and the filtering is implicitly provided by the volume discretization of the computational area to the scale of a singular computational cell. The filtered continuity and momentum equations are:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = \frac{\partial (\sigma_{ij})}{\partial x_j} - \frac{\partial \bar{P}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j}
\]

where the overbar denotes a filtered variable of the general form

\[
\bar{\phi}(x) = \frac{1}{V} \int \phi(x') dx', \quad x' \in v
\]

here \( V \) is the cell volume and \( x' \) represents special filter cutoff. The filter function, \( G(x, x') \), is defined as follows:

\[
G(x, x') = \begin{cases} 
1/V, & x' \in v \\
0, & x' \in \text{otherwise} 
\end{cases}
\]

and \( \sigma_{ij} \) denotes a stress tensor arising from molecular viscosity described by

\[
\sigma_{ij} = \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial \bar{u}_i}{\partial x_i} \delta_{ij} \right]
\]

and \( \tau_{ij} \) is the subgrid scale stress defined by

\[
\tau_{ij} = \bar{\rho}u_i \bar{u}_j - \bar{\rho} \bar{u}_i \bar{u}_j
\]

where the general form of the density weighted filter is:
\[ \tilde{\phi} = \frac{\rho \phi}{\rho} \]  

(10)

The subgrid scale stress is split into the isotropic and deviatoric parts

\[ \tau_{ij} = \tau_{ij}^d - \frac{1}{3} \tau_{kk} \delta_{ij}^d + \frac{1}{3} \tau_{kk} \delta_{ij}^i \]  

(11)

where the isotropic portion is small enough to be neglected and the deviatoric portion is modeled as

\[ \tau_{ij}^d - \frac{1}{3} \tau_{kk} \delta_{ij}^d = -2 \mu_t \left( S_{ij} - \frac{1}{3} S_{kk} \delta_{ij} \right) \]  

(12)

Here \( S_{ij} \) is the strain rate tensor. The eddy viscosity, \( \mu_t \), is modeled as

\[ \mu_t = \rho L_s^2 \| \vec{S} \| \]  

(13)

where \( L_s \) is the mixing length for the sub-grid scales and \( \| \vec{S} \| \) is computed using:

\[ \| \vec{S} \| = \sqrt{2 S_{ij} S_{ij}} \]  

(14)

**Cavitation model**

To model the cavitation process, the continuity equation is reformulated using the assumption that there is a local liquid-vapor equilibrium exists over short spatial scales. Under this assumption the pressure and velocity fields can be calculated by solving the Navier-Stokes and continuity equations based on volume averaged density, velocity and viscosity. Therefore, there is no need to track the interface between the liquid and vapor phases. The continuity equations for the mixture and vapor phases are expressed as follows:

\[ \frac{\partial}{\partial t} (\rho_m \bar{u}_m) + \nabla \cdot (\rho_m \bar{u}_m) = 0 \]  

(15a)

\[ \frac{\partial}{\partial t} (\rho_v \bar{u}_v) + \nabla \cdot (\rho_v \bar{u}_v) = R \]  

(15b)

Here \( R \) is the mass transfer rate between the liquid and vapor phase and the mixture density \( \rho_m \) is calculated by:
\[ \rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \]  

(16)

where the \( n \) is the number of phases, \( \alpha_k \) is the volume fraction of phase \( k \). The volume fractions are constrained by the following equation,

\[ \sum_{k=r,l} \alpha_k = 1 \]  

(17)

Additionally the viscosity in equation 2 is treated as the local volume averaged viscosity of the phases and is calculated using

\[ \mu_m = \sum_{k=r,l} \alpha_k \mu_k \]  

(18)

The growth and collapse of the cavitation bubble are modelled using the Zwart-Gerber-Belamri [32] model which models the mass transfer between the phases as

\[
\text{If } P \leq P_v: \quad R = F_{evap} \frac{3\alpha_{nuc}(1-\alpha_v)\rho_v}{R_B} \sqrt{\frac{2}{3} \left( \frac{P_v - P}{\rho_l} \right)} \\
\text{If } P > P_v: \quad R = -F_{cond} \frac{3\alpha_v\rho_v}{R_B} \sqrt{\frac{2}{3} \left( \frac{P_v - P}{\rho_l} \right)}
\]

(19)

where \( P_v \) is the vaporization pressure and \( \alpha_v \) is the vapor volume fraction. \( F_{cond} \) and \( F_{vap} \) are the evaporation and condensation coefficients, \( \alpha_{nuc} \) is the nucleation site volume fraction and \( R_B \) is the bubble radius. All of these parameters are constants and are derived from empirical correlations.

**Computational Setup**

The injector geometry used in this study is shown in Figure 1. The inlets to the domain are the gain orifices which have a 60° angle and are offset which imparts a rotational motion to the flow throughout the domain. The outlets are the ends of the spray holes. The model has 8 spray holes that are equally spaced and have a 19° exit angle \( (\theta_{sprayhole}) \) and a \( K \) factor of zero. The \( K \) factor describes the conicity and is defined by:

\[ K = \frac{D_{\text{inlet}} - D_{\text{outlet}}}{L_{\text{sprayhole}}} \]  

(21)
where $D_{inlet}$ and $D_{outlet}$ are the diameters at the inlet and outlet of the spray holes and $L_{sprayhole}$ is the spray hole length. Figure 1 shows the entire fluid domain and a close up of the sac with the inlets and outlets marked as well as other relevant geometrical features.

**Figure 1.** The injector with duel gain orifices and the sac. Left: the injector. The needle is in red. Right: a close up of the sac with pressure outlets. The lift is defined as the vertical distance of the needle tip between its highest and lowest position during one injection cycle.

In our study we focus on the needle at hover because this position accounts for the majority of the injection period. At the inlet we specified a total pressure of 2400 bar and the $k$ and $\varepsilon$ are determined by the following equations [33]:

$$k = \frac{3}{2} (0.16 U_\infty)^2 (Re)^{-1/4}$$  \hspace{1cm} (22)

$$\varepsilon = C_{\mu}^{3/4} \frac{k^{3/2}}{0.09 D}$$  \hspace{1cm} (24)

here $I$ is the turbulent intensity, $Re$ is the Reynolds number based on the inlet orifice diameter and the averaged fuel injection velocity, $C_{\mu}$ is an empirical constant and $l$ is the turbulent length scale based on the inlet diameter $D$. Macian et al.[17] showed that for injector simulations variations between 1 and 20% for the inlet turbulence intensity has no impact on the results. In our study we choose 5% for the turbulence intensity. A pressure outlet boundary condition is used at the spray holes and the pressure is set to the atmospheric pressure.
The fluid properties are based on the calibrated diesel fuel Viscor 1487 at a constant temperature of 150°C. We further assume that the density and viscosity are dependent only on the local pressure. The vapor phase is treated as an ideal gas undergoing isothermal expansion and compression. Both the DDES and RANS simulations used the realizable $k$-$\varepsilon$ turbulence model with the enhanced wall treatment option for the near wall treatment.

**Grid sensitivity analysis**

Grid sensitivity analysis was conducted for both the RANS and DDES models. Here we only report the DDES results. The baseline mesh contained approximately 8.5 million cells and had a maximum $y^+$ of 3.1. The coarse mesh contained approximately 6 million cells a maximum $y^+$ of 6.5. The refined mesh had about 10 million cells and a maximum $y^+$ of 2.2. To balance between accuracy and computational cost, we used the enhanced wall treatment option which generally requires $y^+<5$ to resolve the viscous sublayer\[30,33\]. These simulations were carried out with a time step of 0.05 microseconds which was chosen from the temporal sensitivity analysis.

The lateral forces on the 3 surfaces near the tip of the needle shown in Figure 2 were analyzed. Table 1 shows the maximum amplitude of the lateral force in the X-direction. There is a large discrepancy in the force magnitude between the coarse and medium meshes. The difference is reduced to less than 6% on all surfaces between the medium and fine meshes. To balance the computational cost and accuracy, the rest of this study was performed using the medium mesh.

![Figure 2](image.png)

**Figure 2.** Locations for the bottom four surfaces of the injector needle.
Table 1. Force Amplitude for Grid sensitivity Analysis

<table>
<thead>
<tr>
<th></th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface 1 Amplitude (N)</td>
<td>0.32</td>
<td>0.38</td>
<td>0.38</td>
</tr>
<tr>
<td>Surface 2 Amplitude (N)</td>
<td>2.25</td>
<td>2.88</td>
<td>2.75</td>
</tr>
<tr>
<td>Surface 3 Amplitude (N)</td>
<td>1.10</td>
<td>0.98</td>
<td>0.93</td>
</tr>
</tbody>
</table>

Temporal sensitivity

A temporal sensitivity analysis was performed to choose the time step required to resolve the relevant flow features in the DDES simulation. Three time steps were chosen: 0.1, 0.05 and 0.025 microseconds. Figure 3 shows the lateral force magnitude on surface 3 which was chosen because it experiences relatively large fluctuations in the force magnitude over a short time period. The temporal analysis was performed over a long time period but only 5 microseconds of force history is shown here.

![Lateral Force Magnitude: Surface 3](image)

**Figure 3.** Lateral force magnitudes on surface 3 from the temporal sensitivity analysis.

From the analysis the largest time step was found to be inadequate to capture the frequency and amplitudes of the fluctuations in the lateral forces. The two smaller time steps were better to capture the fluctuations of the forces. We chose the time step of 0.05 microseconds in our simulations because the discrepancies between \( dt=0.05 \mu s \) and \( 0.025 \mu s \) were small, indicating a converged solution is reached at \( 0.05 \mu s \).
Results and Discussions

Lateral Force Histories

We hypothesized that the lateral needle motion was caused by the lateral forces on the needle. Figure 4 shows the lateral force history on the needle tip (surface 1 in Figure 2) and the total lateral force on the needle. The results from the DDES and RANS simulations are compared. Two conclusions can be drawn here. First, the DDES simulation was able to capture the high frequency fluctuations that RANS simulation missed. Second, the DDES simulation predicted high force magnitudes than the RANS simulation. Our observations are also in agreement with those of Constantinescu et al.[27] who showed that DDES simulations can capture high frequency fluctuations not seen in the RANS simulations and predict higher peak fluctuations.

![Figure 4. Lateral force histories on diesel injector needle. Left: force on surface 1; Right: force on the whole needle.](image)

The lateral force histories on Surfaces 2 and 3 are shown in Figure 5. DDES results reveal higher amplitude fluctuations than RANS. Compared to surface 1 the forces on surfaces 2 and 3 also exhibit higher fluctuation amplitudes. To better compare the DDES and RANS results, a moving average window of 2.5 μs is applied to the DDES force history (Figure 6), which has the effect of averaging out the high frequency fluctuations. The averaged force history of DDES has similar peak magnitudes as RANS but DDES has a slightly lower dominant frequency than RANS.
Figure 5. Later force histories on surfaces 2 and 3.

Figure 6. Lateral force histories of surfaces 2 and 3 with a moving average window of 2.5 $\mu$s applied to the DDES simulation.

Pressure Distribution

We further analyze the pressure distribution on the needle. Figure 7 shows time history of the lateral force on the needle tip as well as the deviation of the location of the minimum pressure from the center of the tip from both DDES and RANS simulations. In both cases the deviation strongly correlates with the magnitude of the lateral force. However, the DDES simulation shows a much larger deviation than the RANS simulation.
Figure 7. Time history of the lateral force and derivation of the location of the minimum pressure from the needle tip. Left: DDES; Right: RANS

Flow Rollup in the Sac

The previous section showed that the lateral force on the tip is driven by the location of the minimum pressure on the needle surface. This low pressure is created by a vortical flow structure swirling around the needle within the injector sac (Figure 9). Simulation reveals that the location of the minimum pressure is the point where this vortical structure intersects with the needle surface. The rollup is visually represented using the Q-criterion [34]. The Q-criterion defines the flow rollup region as areas where the second invariant of $\nabla \vec{u}$ is positive (i.e., where the local vorticity magnitude is greater than the strain rate).
Figure 8. The iso-surface of the Q criteria (>0.01) from DDES simulation at time instances of a) high lateral force, b) middle lateral force and c) minimum lateral force with the orientations marked for spray hole 1 and 5. The color is based on the vorticity magnitude.

Our simulations also show that the lateral force is correlated to the speed that the vortical flow structure rotates. In this paper we define the volume averaged lateral speed of the flow inside the sac as follows:

$$\text{Swirl Speed} = \frac{\sum_{\text{sac}} V \cdot \sqrt{u_x^2 + u_z^2}}{\sum_{\text{sac}} V}$$

Figure 9 compares the swirl speed and the lateral forces on Surfaces 2 and 3. It is clear there is a positive correlation between the two variables: a high swirl speed correlates to large lateral force magnitude and a low swirl speed correlates to low lateral force. This suggests that the vortical structure is creating a flow imbalance by covering the entrances to the spray holes. The magnitude of the pressure imbalance on the surface is correlated to the rotational speed of the structure.
**Figure 9.** Correlation between the lateral forces on surfaces 2 and 3 and the swirling speed inside the sac. Left: surface 2; Right: surface 3.

**Hole-to-hole variation**

The swirling flow in the sac causes flow variations in the spray nozzles. In this section we first explore the variation in volume of vapor within the spray holes. Since there are only slight differences between RANS and DDES (Figure 10) the remainder of the section only shows the results from DDES.

**Figure 10.** Comparison between the vapor volume in a single spray hole between the DDES and RANS simulations.

Figure 11 shows the vapor variation in spray Holes 1 and 5. These two holes are on the opposite sides of the sac. The peak in Hole 1 occurs approximately at the time that Hole 5 has the lowest vapor volume and the peak in Hole 5 occurs around the moment that Hole 1 has the lowest vapor volume.
Figure 11. Vapor volume within spray holes on opposite sides of the sac over a singular peak.

The variation in vapor volume is caused by the flow rollup in the sac. As shown in Figure 12 the rollup alternatively covers the entrances to spray holes 1 and 5, leading to the variation in vapor volume. Figure 12 shows the velocity contours and vapor fraction at the points indicated in Figure 11.

When the entrance to spray hole 1 is covered by the vortical structure at time A, fluid enters the spray hole 1 with a high vertical angle with a high velocity. This creates a large flow separation near the entrance to spray hole 1 and also results in a large volume where vapor is generated. At time D the vortical structure covers the entrance to spray hole 5. At that time fluid enters spray hole 5 at a steeper angle, which causes sudden decrease in the pressure and a high rate of vapor formation. Meanwhile, spray hole 1 has more uniform velocity profile at the spray hole entrance, indicating small flow separation and low vapor generation.
At time A the vortical structure has passed in front of the entrance to spray hole 1 and blocks flow at the bottom of the spray hole entrance. This increases both the downward angle and the velocity of flow at the top of the spray hole and leads to increased separation and vapor formation at the entrance to the spray hole. The distance between the vortical structure and spray hole 5 at this point in time is large and there minimal influences between the structure and the flow entering the spray hole. At time instance B the vortical structure has moved beyond the entrance to spray hole 1 decreasing the downward angle and velocity of the flow entering at the top of the spray hole. This leads to a decrease in the separation at the top of the spray hole and a decrease in the amount of vapor generated. The earlier reattachment of the flow leads to a portion of the vapor
to detach and flow out the exit of the spray hole. The vortical structure is approaching spray hole 5 on the opposite side causing slight increases in the velocity at the top of the entrance to spray hole 5 and a slight increase in the vapor generation. At time instance C the vortical structure is close enough to start significantly impacting the downward angle and velocity of the flow entering spray hole 5 which leads to the increases in the generation of vapor. On the opposite side the distance between the vortical structure and the entrance to spray hole 1 is great enough that there is no significant increase to the downward velocity and angle of flow entering spray hole 1. Time instance D creates a flow pattern that is similar to time instance A but for the opposite spray holes. At this time instance the vortical structure is blocking the bottom of the entrance to spray hole 5 which leads to an increased downward angle and flow velocity at the top of the spray hole. This leads to a caviated area that covers more of the cross sectional area of the spray hole and an increase in the vapor volume. Time instances beyond this would be show similar trends. As the vortical structure passes over the spray hole the vapor increases. When the structure moves beyond the entrance to the spray hole excess vapor detaches and exits the spray hole and the vapor volume returns to the lower value.

The mass flow rate is proportional to the vapor volume within the individual spray hole. Figure 13 shows the comparison between the vapor volume and the mass flow rate for spray hole 1 in the DDES simulation. It is clear that not only do the peaks for the vapor volume match up with the local minimums for the mass flow rates but the magnitude of the vapor volume peaks match up with the magnitudes for the mass flow rate minimums.

![Figure 13. Vapor volume and mass flow rate for spray hole 1 in the DDES simulation.](image)
CONCLUSIONS

The internal flow of a dual gain orifice injector has been numerically simulated. The simulations were performed under conditions similar to real world operation with the lift equivalent to being at hover. The major conclusions from this study are:

- The dual gain orifices of the injector creates a rotating vortical structure within the sac of the injector which influences the needle forces and the hole-to-hole flow variation.
- The rotational speed of the vortical structure is proportional to the average lateral force magnitude on the lower needle surfaces.
- The magnitude of the predicted lateral force is 50% higher in the DDES simulation than in the RANS simulation.
- The lateral force magnitude was shown to be a function of the eccentricity of the inception point of the vortical structure. The DDES simulation predicted higher peak eccentricity than the RANS simulation.
- The differences in the force amplitudes and frequencies show that higher fidelity models are needed in order to accurately model the motion of the injector due to the flow forces.
- Proximity of the vortical structure to the entrance to a spray hole created peaks in the vapor formation and minimums in the mass flow rate. The transient motion of the vortical structure created time dependent hole-to-hole mass flow rate and vapor formation variations.

References


