Numerical Analysis of Diesel Sprays Impinging on Combustion Chamber Walls by Means of a Discrete Droplet / Liquid-Film Model

Tomoyuki Wakisaka*, Shin-ich Takeuchi**, Fumihiko Imamura*
Kunikazu Ibaraki*** and Yoshihiro Isshiki****

*Department of Mechanical Engineering, Graduate School of Engineering, Kyoto University, Yoshida, Sakyo-ku, Kyoto 606-8501
**MMC Computer Research Ltd.
***Mitsubishi Motors Corporation
****Department of Mechanical Engineering, Setsunan University Japan

ABSTRACT

The purpose of this study is to numerically analyze the behavior of fuel sprays impinging on the combustion chamber walls of D.I. diesel engines and to investigate the effects of injection conditions and gas flow on the mixture formation. The behavior of droplets impinging on a wall and that of liquid film formed on the wall have been treated by means of a discrete droplet/liquid-film model proposed by the authors. The authors' improved droplet breakup model has been incorporated into the authors' GTT code along with the evaporation model of droplets and that of liquid film by Kadota and Hiroyasu. Using these spray submodels, fuel sprays impinging on a combustion chamber wall have been numerically analyzed, and the validity of the models has been confirmed by comparing the calculated results with the experimental data. As a result, it has been shown that the behavior of sprays impinging on a combustion chamber wall can be predicted reasonably well. Furthermore, the mixture formation process with a hollow-cone spray injected into a D.I. diesel engine cylinder at early time in compression stroke has been numerically simulated. It has been found that tumble flow or inclined swirl is very effective for making the spatial distribution of fuel vapor concentration uniform.

INTRODUCTION

For numerically analyzing the behavior of fuel sprays impinging on the combustion chamber walls in fuel injection engines, the authors have developed spray submodels to describe the droplet impingement on walls and the liquid film formation on the basis of the discrete droplet/liquid-film concept. Namely, the liquid film formed by sprays impinging on a wall is treated as a group of "liquid film elements". When droplets in a droplet parcel impinge on a wall and do not rebound, the droplets are converted individually into liquid film elements in a liquid film parcel. The group of such liquid film parcels is regarded as the liquid film. As the droplet breakup model, the authors have proposed a modified wave breakup model. These submodels are incorporated into the authors' GTT (Generalized Tank and Tube) code. As the submodel of droplet coalescence, the model in the KIVA code is used. Concerning the evaporation model of each droplet, the quasi-steady evaporation model of a single droplet by Kadota and Hiroyasu is incorporated into the code. As the evaporation model of each "liquid film element", a model based on the evaporation model of a single droplet on a hot wall by Hiroyasu et al. is employed. Fundamental examination of the spray submodels incorporated into the GTT code has been carried out for the free sprays injected in high-pressure, high-temperature quiescent gas and for the sprays impinging on a hot wall in high-pressure, high-temperature quiescent gas.

In this study, using the above-mentioned spray submodels, diesel sprays (single and split injection conditions) impinging on the wall of a modeled combustion chamber are numerically analyzed and the calculated results are compared with the experimental data for confirming the validity of the authors' spray submodels. Then, aiming at the generation of homogeneous mixture for premixed lean diesel combustion, the mixture formation process with a hollow-cone spray which is injected into the combustion chamber of a D.I. diesel engine at early time in compression stroke is numerically simulated.

METHOD OF NUMERICAL ANALYSIS

The GTT code calculates the gas flows in engines by means of fully-implicit algorithm based on a finite volume method with generalized curvilinear coordinates, using the \( k - \varepsilon \) two-equation model as a turbulence model. As the differencing scheme of convection terms, the TVD scheme is used for the Navier-Stokes equations, and the Hybrid scheme is used for the conservation equations of enthalpy, turbulence energy \( k \) and its dissipation rate \( \varepsilon \). Pressure-velocity coupling is accomplished by means of the SIMPLEC algorithm. In this study, chemical reaction is not considered. Concerning the interaction between the gas and the fuel droplets, the vaporized mass, momentum and enthalpy lost by the droplets are given to the source terms of their respective conservation equations in the gas phase. The time increment \( \Delta t \) for calculating the velocity field is set at 0.05 - 0.15 ms, and the spray behavior is calculated explicitly in subcycles with a smaller time increment \( \Delta t_s = (1/20 - 1/50) \Delta t \). The concentration field of fuel vapor is calculated explicitly in subcycles using the CIP method, since its false diffusion is very small. All the calculations with respect to spray phenomena in this study are carried out using personal computers.

SIMULATION OF SPRAYS IN A MODELED COMBUSTION CHAMBER

The fuel sprays injected into a combustion chamber
which models the combustion chamber of a D.I. diesel engine are numerically simulated and the calculated results are compared with the experimental results by Hiroyasu et al. In their experiment, a modeled combustion chamber of two-dimensional shape shown in Fig. 1(a) is fixed in a vessel in which high-temperature (833 K), high-pressure (4 MPa) nitrogen gas is filled, and the light diesel fuel oil is injected from a hole nozzle (hole diameter = 0.22 mm, injection pressure = 90 MPa, injected fuel mass = 10.9 mg). The following three injection patterns are employed: single injection (injection period: \( t = 0 - 1.35 \) ms); split injection with the injection mass fractions of 25\% (injection period: \( t = 0 - 0.6 \) ms) and 75\% (injection period: \( t = 1.6 - 2.5 \) ms); and split injection with the injection mass fractions of 75\% (injection period: \( t = 0 - 1.0 \) ms) and 25\% (injection period: \( t = 2.0 - 2.5 \) ms). Under the same injection and ambient gas conditions as those in the experiment, spray behavior is numerically simulated using the computational grid shown in Fig. 1(b) (grid division number = 33x22x24). The number of injected droplet parcels is 8000.

The calculated results of the droplet parcel distribution (projection of all the parcels) and the contour lines of fuel vapor mass fraction \( C \) in the vertical section including the spray axis are shown in Fig. 2. The schlieren photographs taken in the experiment\(^6\) are shown in Figs. 3-5. In these figures, the calculated iso-value surfaces of fuel vapor mass fraction \( C = 0.01 \) and turbulence energy \( k = 7 \mathrm{m}^2/\mathrm{s}^2 \) are also shown. For split injection cases, three-dimensional representations of these iso-value surfaces are shown in Fig. 6. By comparing those calculated results shown in Figs. 2-6 with the schlieren photographs, it is found that the calculated results for the three cases reproduce reasonably well the spray behavior in the experiment, such as the impingement of liquid and vapor phases on the chamber side wall and the development of vapor region along the side and bottom walls. It is also found that strong turbulence is generated by the spray impingement on the chamber wall and the turbulent region enlarges along the wall. This resembles the enlargement of fuel vapor region along the wall. This means that the mixture formation in the spray impinging on the wall is strongly affected by the turbulence.

**NUMERICAL ANALYSIS OF HOLLOW-CONE SPRAYS IN A D.I. DIESEL ENGINE COMBUSTION CHAMBER**

The model engine used in this study is a D.I. diesel engine (bore = 130 mm, stroke = 130 mm, compression ratio = 16.6, connecting rod length = 240 mm, TDC clearance height = 1.12 mm) having a shallow-dish type cavity concentrically on the piston head as shown in Fig. 7(a). The opening diameter of the cavity is 100 mm and its depth is 23 mm. This type of cavity is employed here because it helps the development of tumble flow during intake stroke\(^10\). The computational grid for the model engine is shown in Fig. 7(b) (grid division number = 40x40x20).

A fuel injector which can generate a hollow-cone spray is employed here. The injector is located at the central position of the cylinder head perpendicularly or at a peripheral position (48 mm away from the cylinder head center) obliquely (inclination angle = 30 deg.), as shown in Fig. 8. The fuel used is tridecane and is injected into the cylinder in the early period of compression stroke at a constant injection rate. Namely, injection starts at 60 deg ABDC and ends at 70 deg ABDC (the number of injected droplet parcels = 8000, initial droplet diameter = 0.05 mm, injection velocity = 100 m/s). The total mass of injected fuel is 48.3 mg/stroke, which corresponds to overall air-excess ratio of 3 (fuel vapor mass fraction \( C = 0.0218 \)). Assuming that the injector is a swirl type, a swirl component is added to the injection velocity (swirl angle\(^11\) is set at 45 deg.). As for a method of injecting droplet parcels as a hollow-cone spray, the method described in Ref.\(^11\) is employed. According to experimental results\(^12\), the cone angle of a hollow-cone spray becomes narrower with an increase in ambient gas pressure. Therefore, the outer and inner cone angles are changed according to the cylinder pressure so that at atmospheric pressure they may become 60 deg. and 40 deg., respectively.

As an initial condition of gas flow in the cylinder, swirl or tumble flow (or both) is given at BDC of compression stroke for some test cases (cf. Table 1). The gas flow, spray behavior and mixture formation process in the model engine cylinder are numerically analyzed during compression stroke (from BDC to TDC) at an engine speed of 1000 rpm for eight cases shown in Table 1. The initial air pressure and temperature are set at 0.11 MPa and 350 K, respectively, and the wall temperature is set at 550 K throughout the compression stroke.

<table>
<thead>
<tr>
<th>Test case</th>
<th>Injector position</th>
<th>Swirl ratio</th>
<th>Tumble ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>Central</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Case 2</td>
<td>Central</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>Case 3</td>
<td>Central</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>Case 4</td>
<td>Central</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Case 5</td>
<td>Peripheral</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Case 6</td>
<td>Peripheral</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>Case 7</td>
<td>Peripheral</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>Case 8</td>
<td>Peripheral</td>
<td>2</td>
<td>2</td>
</tr>
</tbody>
</table>

As for a time increment in the calculation, the crank angle interval in the main cycle is set at 2.5 deg. crank angle before the start of fuel injection, while the interval in the main cycle is set at 0.5 deg. crank angle (the interval in the subcycle is set at 0.025 deg. crank angle) after the start of fuel injection. It took about 20 hours on a personal computer (CPU: Pentium II/266 MHz, OS: Windows NT) to calculate the gas flow, spray and fuel vapor behavior during compression stroke for one case.

Figure 9 shows the gas velocity vectors in a vertical section at 65 deg. ABDC for four cases with or without tumble flow. It is found that the spray induces a pair of vortices (turbulent vortex), which are affected by tumble flow (clockwise rotation). From the calculated results of the spatial distribution of fuel vapor mass fraction \( C \) in the cylinder, the spatial mean value and standard deviation of \( C \) at each crank angle are calculated for the all cases. The mean values, \( C_{\text{mean}} \), are shown in Figs. 10(a) and (b), and the standard deviations, \( C_{\text{std}} \), are shown in Figs. 10(c) and (d). The distributions of droplet parcels (projection of all the parcels) and fuel vapor mass fraction at 70 deg. (the end of fuel injection), 90 deg. and 120 deg. ATDC for Cases 1-4, Case 5 and Case 8 are shown in Figs. 11 and 12.
Fig. 1 Modeled combustion chamber

Fig. 2 Droplet parcels and fuel vapor mass fraction (calculated)

Fig. 3 Spray behavior for single injection case

Fig. 4 Spray behavior for split injection (25-75) case

Fig. 5 Spray behavior for split injection (75-25) case

Fig. 6 Calculated iso-value surfaces
Fig. 7 D.I. diesel engine model

(a) Model shape
(b) Computational grid

Fig. 8 Examples of spray shapes (65 deg.ABDC)

(a) Case 1 (Cases 1—4: central injection)
(b) Case 3
(c) Case 5 (Cases 5—8: peripheral injection)
(d) Case 7

Fig. 9 Gas velocity vectors (65 deg.ABDC)

(a) Mean value, Cmean (Cases 1—4) (central injection)
(b) Mean value, Cmean (Cases 5—8) (peripheral injection)
(c) Standard deviation, Cdev (Cases 1—4) (central injection)
(d) Standard deviation, Cdev (Cases 5—8) (peripheral injection)

Fig. 10 Mean value and standard deviation of fuel vapor mass fraction
Fig. 11 Distributions of droplet parcels and fuel vapor mass fraction (central injection)
From the temporal variations of $C_{\text{mean}}$ (Figs.10(a) and (b)), it is found that the droplet evaporation in the central injection cases is faster than that in the peripheral injection cases. Especially in Cases 3 and 4, the droplet evaporation is the fastest, which is obvious in Figs.11(c) and (d). From the temporal variations of $C_{\text{ave}}$, it is found that in the central injection cases (Fig.10(c)) the spatial inhomogeneity of fuel vapor mass fraction is largely reduced by the tumble flow (Case 3; cf. Fig.11(c), or by the combined flow of swirl and tumble (Case 4; cf. Fig.11(d)); the standard deviation $C_{\text{ave}}$ at TDC in Case 4 is the smallest in the eight cases tested. In the peripheral injection cases (Fig.10(d)), the spatial inhomogeneity of fuel vapor mass fraction is considerably reduced by the combined flow of swirl and tumble (Case 8; cf. Fig.12(b)), though the standard deviation $C_{\text{ave}}$ in the case without swirl nor tumble flow (Case 5; cf. Fig.12(a)) is the largest among the eight cases.

From Figs.11 and 12, it is found that the dispersion and evaporation of droplets and the spatial distribution of fuel vapor concentration are greatly affected by gas flow, and the tumble flow or the combined flow of swirl and tumble (i.e. inclined swirl) can make the spatial inhomogeneity of fuel vapor concentration considerably small.

CONCLUSIONS

From the calculated results in this study, it has been shown that the behavior of fuel sprays impinging on the combustion chamber wall can be predicted reasonably well by means of the authors' spray submodels. Furthermore, the hollow-cone spray which is injected into the cylinder of a D.I. diesel engine with a shallow-dish type cavity at early time in compression stroke has been numerically analyzed, and the effects of the injection position and gas flow on the spray behavior and mixture formation process have been made clear. It has been found that tumble flow or inclined swirl is very effective for making the spatial distribution of fuel vapor concentration uniform.

REFERENCES