Application of CFD Analysis to the Development of a Direct-Injection System

*by Hideaki Katashiba and Kazuhiko Kawajiri**

Direct-injection gasoline engines have been widely introduced in global markets in recent years as environmentally friendly engines. To improve their fuel economy and low-emission performance further, it is necessary to have a better understanding of the fuel spray pattern, mixture formation and combustion state in the cylinder so that fuel supply control and component characteristics can be optimized. This article describes a computational fluid dynamics (CFD) analysis method that has been applied to ascertaining these in-cylinder conditions and presents the results of basic experiments conducted during the development process.

Fuel Spray Model

The behavior of fuel injected directly into the cylinder is a major factor governing mixture formation. A fuel spray model capable of calculating fuel spray behavior accurately must be developed in order to calculate the mixture formation process. In the CFD analysis, in-cylinder gas flow was found by applying the finite element method to solve equations for the mass and momentum of a compressible fluid and the equation for conservation of energy. The standard k-ε turbulence model was used. Fuel spray behavior was calculated based on a discrete droplet model, taking into consideration droplet break-up, coalescence and evaporation. The initial spray condition was given as a function of the plunger operation, including the initial injection velocity, injection angle and droplet size that were determined from the injector specifications. These values were then corrected based on a comparison with the results of experimental measurements of fuel spray properties. The gasoline fuel was assumed to be C_8H_{18} and its physical properties were used in the analysis. The distribution function proposed by Nukiyama and Tanasawa was assumed for the initial droplet size distribution.

The simulation results obtained with the fuel spray model developed in this study are compared in Fig. 1 with the experimental results for fuel spray growth when fuel was injected under a backpressure condition of atmospheric pressure. The experimental results are shown as images captured with a high-speed camera fitted with a photomultiplier. These images are cross-sections of the fuel spray that were visualized by means of an Ar laser light sheet. The simulation results show overall images of the fuel spray. It is clear from this figure that the newly developed fuel spray model accurately reproduced fuel spray growth behavior. Fig. 2 compares the experimental and simulation results for spray penetration and the spray cone angle.

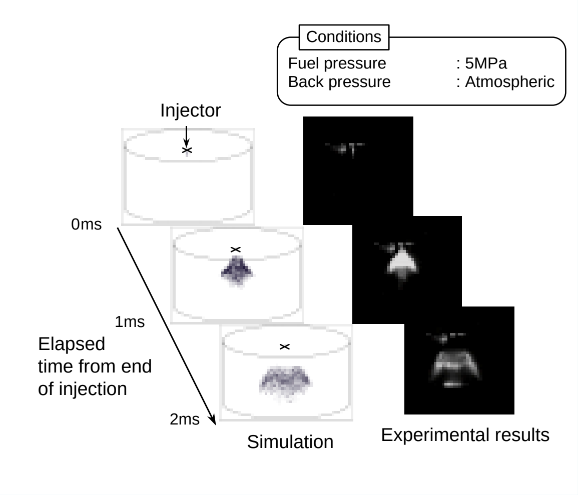

Fig. 1 Simulation results for fuel spray growth.

Good agreement is seen between the two sets of results for both spray penetration and the spray cone angle. It was also confirmed that the experimental and simulation results showed good agreement with respect to the Sauter Mean Diameter of the spray droplets following injection.

In-Cylinder Mixture Formation

Visualization of in-cylinder mixture formation by simulation allows quantitative evaluation of the phenomena involved, which is a critical factor in designing the optimum fuel injection system. In stratified-charge combustion, it is necessary to make effective use of the cavity provided at the top of the piston to transport a compact combustible mixture cloud to the vicinity of the spark plug for ignition and combustion. In this study, an analysis was made of the

**Hideaki Katashiba is with Automotive Electronics Development Center and Kazuhiko Kawajiri is with Advanced Technology R&D Center.*

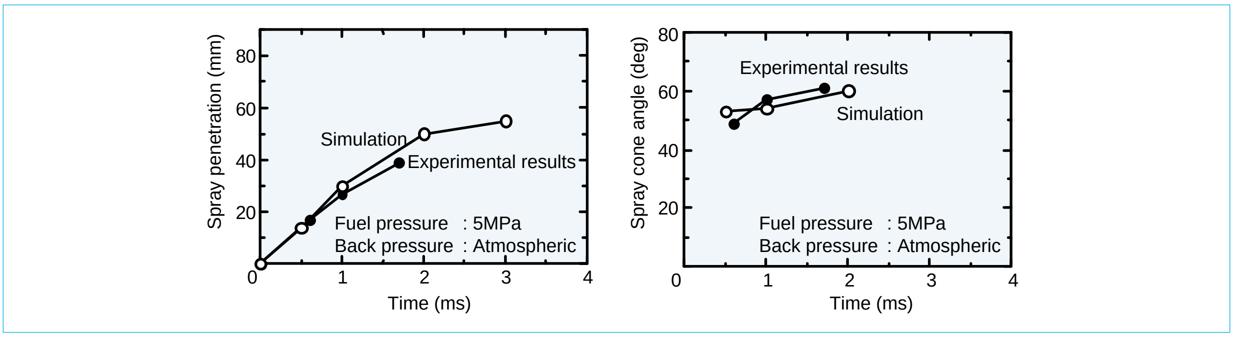

Fig. 2 Spray penetration and spray angle obtained in fuel injection simulation.

influence of the fuel spray angle produced by the injector, which represents a key engine component, on mixture formation in the cylinder.

The configuration of the combustion chamber of the direct-injection gasoline engine that was analyzed is shown schematically in Fig. 3. The injector was positioned between the two intake ports, and fuel was injected at an oblique angle toward the eccentric piston cavity. The spark plug was centrally located at the top of the cylinder. Calculations were performed for three types of initial spray cone angles of 50°, 60° and 70° and the spray behavior was compared.

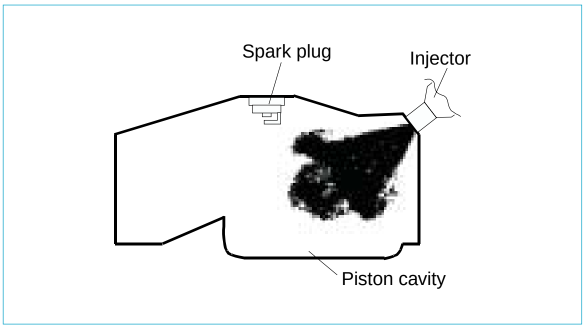

Fig. 3 Combustion chamber of direct injection gasoline engine.

Fig. 4 compares the fuel spray behavior in the combustion chamber. With a spray cone angle of 70°, the broad spray pattern causes the fuel to overflow the cavity, with the result that a large quantity of fuel sticks to the walls. It is estimated that a larger quantity of unburned fuel would be evacuated from the cylinder under this condition. With a small spray cone angle of 50°, many spray droplets overflow the cavity following collision with the piston top surface, resulting in a short interval for obtaining the optimum A/F ratio. As described here, simulations allow the spray behavior and mixture distribution to be estimated, making it possible to design the optimum fuel spray properties and other parameters.

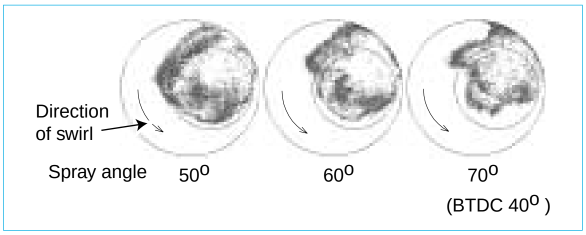

Fig. 4 Behavior of fuel spray in combustion chamber.

Combustion Simulation

This section describes the combustion simulation program that was developed for making detailed visualizations and evaluations of complex combustion phenomena in the cylinder and presents an example of the simulation results. This combustion simulation technology is based on the combustion analysis expertise that we have accumulated over many years in connection with fan heaters, furnaces and the like. Combustion reactions were assumed to be single-stage irreversible reactions. A flame area progression model was used as the combustion model, which assumed that combustion proceeded by means of an advancing of a thin-flame front in the combustible mixture. The mean propagation velocity of the flame front ω is given by the following equation, assuming that it is proportional to the spatial gradient of the turbulent combustion velocity S_t and the unburned mass fraction m_{fu}.

$$\omega = C_r \rho_u S_t |\nabla m_{fu}| \text{ .... Eq. (1)}$$

$$S_t / S_u = 1 + C_s u' / S_u \text{ .... Eq. (2)}$$

where ρ_u is the density of the unburned gas, S_u is the laminar burning velocity, u' is the turbulence intensity and C_r and C_s are constants.

The simulation results for flame propagation behavior in the cylinder during homogeneous-charge combustion are compared in Fig. 5 with the experimental data. A single-cylinder optical engine having the piston head and part of the cylinder liner made of quartz glass was used in the experiments to visualize in-cylinder combustion phenomena by means of a high-speed camera fitted with a photomultiplier. The simulation results are shown only for the visualized region relative to the piston diameter. As shown in the figure, the simulation reproduced flame propagation behavior with excellent accuracy as a result of suitably adjusting the model constants used in the combustion model.

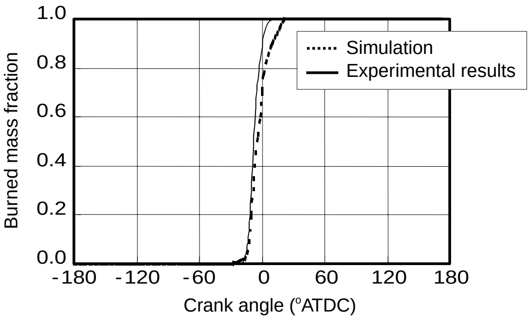

Fig.7 Simulation results of burned mass fraction.

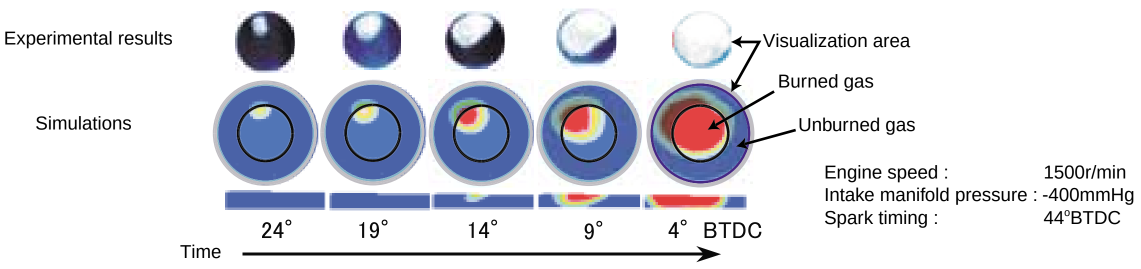

Fig.5 Behavior of flame propagation.

The newly developed combustion model was then applied to an analysis of stratified-charge combustion. The simulation results for cylinder pressure and burned mass fraction are compared with the experimental data in Figs. 6 and 7, respectively. Although the peak combustion pressure calculated in the simulation was several percentage points higher than the experimental value, the simulation results for cylinder pressure and burned mass fraction show good overall agreement with the experimental data. As indicated here, the newly developed combustion model can be applied to combustion calculations for a direct-injection gasoline engine, making it possible to ascertain combustion phenomena and to utilize the resulting data in engine design studies.

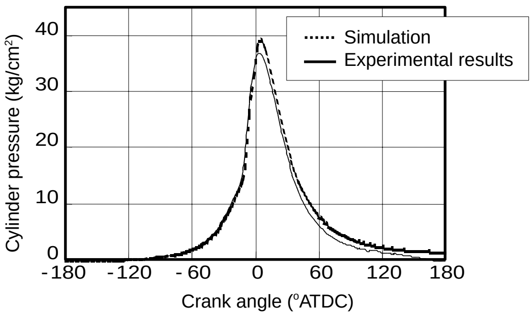

Fig.6 Simulation results for cylinder pressure.

In future work, this simulation technology will be applied to the development of next-generation direct-injection engines and their components with the aim of contributing to the creation of products that achieve even higher fuel economy and lower exhaust emissions.❑