

Practical Application of Combustion Simulation Using CFD for Two-wheeled Vehicle Small Engine

Yasuo MURAKAMI*

Hitoshi KUROSAKA*

ABSTRACT

The Computational Fluid Dynamics simulation of the flame propagation phenomena in a cylinder is carried out to visualize the combustion condition in a small displacement, high-speed 4-stroke motorcycle engine. For verifying the simulation results, steady-state in-cylinder flow, fuel injection behavior, and flame propagation behavior were compared. Each experimental result verifies the simulation results, validating that the CFD simulation is an effective and useful tool to visualize the inside of the cylinder of the engine. This CFD visualizing technology is applied to study the combustion condition of a commercial motorcycle engine, and clarified that the technology is effective to reduce the fuel consumption and expand the lean combustion limit. This technology was applied to the development of the 4-stroke 50 cc PGM-FI (Programmed Fuel Injection) Engine.

1. Introduction

In recent years, demands for motorcycles with greater fuel economy and reduced exhaust emissions have been steadily increasing. For that purpose, measures to protect the environment, such as the adoption of 4-stroke engines and PGM-FI systems for mass-production motorcycles, are being promoted, and also increased research is being carried out for improving the combustion condition in the engine cylinder. In order to improve the combustion condition, it is important to obtain a detailed grasp of physical phenomena such as the gas mixture formation process inside the cylinder, and the flame propagation. Previously, it was common practice to observe these phenomena using an optically accessible engine⁽¹⁾ installed with quartz windows. However, such an engine did not have adequate durability under high temperature and pressure, particularly when running at high engine speeds, so observation of the inside of the cylinder of a small motorcycle engine, which is a high-speed type, is not necessarily the an optimal way for monitoring conditions inside an engine such as this.

Meanwhile, along with amazing progress of computer performance in recent years, Computation Fluid Dynamics (CFD) technology has come into wide use, particularly, in the development of an in-cylinder direct injection engine. For example, there are many papers^{(2), (3)} where CFD simulation is being used for the visualization of the gas mixture formation in order to study the shape of the combustion chamber. Nowadays, it has become an important method for visualization of the inside of the cylinder. However, in a port injection engine, the overall crankshaft displacement angle

duration necessary for analyzing the gas mixture formation process is two or three times longer than that of an in-cylinder direct injection engine in which fuel injection starts during the compression stroke. In addition, the combustion speed is greatly affected by the gas flow condition inside the cylinder. Consequently, there are many issues concerning the calculation accuracy, and to date there are only a few reported cases^{(4), (5)} of verifications having been performed both experimentally and by CFD analysis including the combustion process as well.

In this research, we targeted port injection type motorcycle engines featuring small displacement and high engine speed, and we verified and clarified the fact that CFD analysis of the gas mixture formation process and the combustion process in these engines can be used as a method of visualization of the inside of the cylinder. These results were used in the development of an actual mass production engine, and the effectiveness of this method for optimizing the specifications of the combustion system was confirmed. In addition, some actual applications of the simulation visualizing method are described.

2. Calculation Method

2.1. Gas Flow

In order to calculate the gas flow condition in the cylinder and also the inlet and exhaust ports, it is necessary to discretize the calculation space and to solve the continuity equation of gas flow at each element, the conservation equations for the momentum, the energy and each reaction

* Asaka R&D Center

component, and the equation of state. When the engine is running, the calculation space change varies constantly due to the reciprocating piston and the moving inlet and exhaust valves. Consequently, it is necessary to use a special sliding grid, called a moving boundary grid, in the calculation space. There are several methods for this moving boundary grid. The method used in this research involved many preset periods during which each grid expands or contracts following the deformation of the calculation space, and switching over to the next grid before the grid quality becomes low due to the grid deformation. Some of the computational grids are shown in Fig. 1. Twenty nine grids were used for one cycle. Each grid used a boundary-fitted grid consisting of hexahedron cells. Two-layers of boundary grids were installed inside the inlet and exhaust ports, and one-layer of the boundary grid was installed inside the cylinder. The $k - \varepsilon$ turbulence model was adopted, which was based on the turbulence kinetic energy k , and the rate of turbulent energy dissipation ε . FIRE (AVL List Co.), which is a commercially available 3D CFD code, was used for these calculations.

2.2. Fuel Spray Behavior

The Discrete Droplet Model (DDM) method⁽⁶⁾ was adopted for the spray particle behavior calculations. The DDM method divides the several million fuel droplets supplied to the inlet port into several thousand droplet groups called parcels that have equivalent characteristics (position, droplet diameter, velocity and temperature), do not have mutual interaction, and are used for behavior calculation by tracing these groups using a Lagrangian method. Calculation of the flow in the injector was not performed and the nozzle hole was set to be a boundary condition. Experimental parameters, such as the initial injection velocity, injection rate, spray cone angle and particle size distribution obtained from an open spray experiment using a constant capacity box, were assigned to this model. For the breakup of the droplets, the Taylor Analogy Breakup (TAB) model⁽⁷⁾ was adopted. Regarding the impingement and coalescence of the droplets in the spray, a collision frequency based on a Poisson distribution was assigned only for the droplets in the same calculation call. In this sub-model, the state of a droplet after impingement (coalescence or momentum exchange) was determined by the criterion value at the moment of collision. In addition, the wall film model was adopted to calculate the behavior of reflex and adhered, resulting from droplets impinging with the wall, and also for the flow and evaporation of the fuel liquid film in the inlet port.

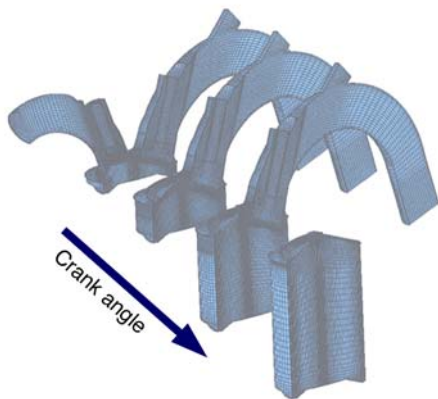


Fig. 1 Computational grids

2.3. Combustion Reaction

Concerning combustion in a spark ignition engine, the combustion reaction starts as a result of the activation of a high-voltage discharge by a spark plug to the mixture gas formed inside the cylinder. When performing a combustion simulation, it is necessary to create some kind of chemical reaction model to calculate the thermodynamic quantity due to the chemical reaction in the energy conservation equation and each generated quantity in the chemical species conservation equation. In the case of a spark ignition engine, the chemical reaction takes place in an extremely short time along the flame surface, so it was considered adequate to take into consideration the chemical equilibrium after combustion, and hence the Eddy Dissipation Model⁽⁸⁾ propounded by Magnussen was used. With this model, the mean reaction rate R_{fu} that determines the quantity of a generated chemical species was calculated by Eq. (1).

$$R_{fu} = A(\rho\varepsilon/k)\min\{y_{fu}, y_{ox}/S, B \cdot y_{pr}/(1+S)\} \quad (1)$$

Here, A and B are constants, y_{fu} , y_{ox} and y_{pr} are the mass fractions of fuel, oxygen and combustion products respectively, S is the theoretical quantity of oxygen, and ρ is the density. In an actual engine, heat generates rapidly after an ignition delay time subsequent to the spark discharge. However, this model cannot take into account the oxidation reaction such as the cool flame that occurs at low temperature during the ignition delay period. Thus in the simulation, it is necessary to estimate the ignition delay by experience and set the ignition timing delayed by this amount.

2.4. Boundary Conditions

At the flange of the inlet port and exhaust port, which are the ends of the calculation area, the time history gas pressure and temperature obtained by experiments were defined as boundary conditions. Also, on the walls of the combustion chamber, piston and inlet and exhaust ports, the temperature obtained by experiments was defined, so that boundary conditions were set more closely approaching the operating conditions in an actual engine. In addition, the residual gas concentration subsequent to the valve overlap was taken into account by starting the calculation from the exhaust stroke.

3. Experimental Investigation of Correlation

3.1. Test Engine

Figure 2 shows the shape of the test engine, and Table 1 shows the main specifications of the engine. A two-valve, side ignition plug type engine was used as a typical small-displacement motorcycle engine, and squish is located only in part of the hemispherical type combustion chamber. The inlet port is U-shaped, and the tumble ratio according to the Ricardo measurement method is slightly less than 1. This engine was used to investigate the correlation between the simulations and experiments about the steady-state flow field of the inlet port, the behavior of the fuel spray and the combustion process.

3.2. Verification in the Inlet Port Steady State Flow Field

The inlet port is an important part that affects the output performance of the engine, and the effective valve opening area $Z^{(9)}$ is used as an index for evaluating it. Generally, the

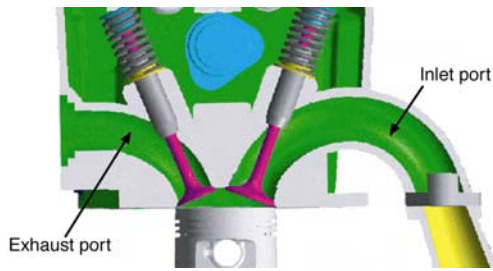


Fig. 2 Sample engine

Table 1 Test engine specifications

Engine type	4-stroke single cylinder, spark ignition
Bore × Stroke (mm)	36 × 49
Displacement (cm ³)	49.8
Valve layout	1 inlet and 1 exhaust valve
Compression ratio	12.0
Combustion chamber	Hemispherical type

inlet port is measured under constant pressure using a steady state flow test rig, but recently there are an increasing number of cases in which this measurement is being replaced by CFD instead. Before an unsteady state flow was calculated, a flow of the inlet port was calculated under steady state conditions, and the correlation between the calculated value and the measured value was verified. The verification items were Z and the velocity distribution at the horizontal cross-section of the cylinder. Laser Doppler Velocimetry (LDV) shown in Fig. 3 was used for measuring the air flow velocity. Multi-point measurement was performed on the same plane, and a comparison with the calculated values was made. Figure 4 shows the results of the comparison of Z , and Fig. 5 shows the results of a comparison of the velocity distribution at the cross-section of the cylinder. It was found that the value of Z obtained by the simulation agreed closely with the measured value, and also that the tendency of Z that reaches its peak off in the high valve lift area could be accurately reproduced. For a velocity distribution at the horizontal cross-section of the cylinder, the changing velocity distribution according to the increase of the valve lift was well estimated. In addition, it was found that the strong downstream flow generated along the cylinder wall, the upstream flow generated directly beneath the inlet valve, the flow pattern divided by the valve stem, and other detailed flow features agreed closely with the experimental results. In this way, adequate correlation concerning the steady state flow calculations was obtained.

3.3. Validity Verification of the Fuel Spray Model

Figure 6 shows a comparison between the results of the open spray test performed in a constant capacity box, and the results of calculation using a spray model that was set based on data, such as the injection velocity, injection rate, spray cone angle and particle size distribution, obtained from this test. The test conditions were a fuel injection pressure of 250 kPa, a fuel injection time of 10 ms, and an atmospheric pressure equal to ambient pressure.

From the results of the calculation, it was confirmed that the spread of the spray, its penetration, the profile of the spray at the end of injection, and so on, obtained by the DDM method was consistent with the experimental results, thus validity of the spray model was confirmed.

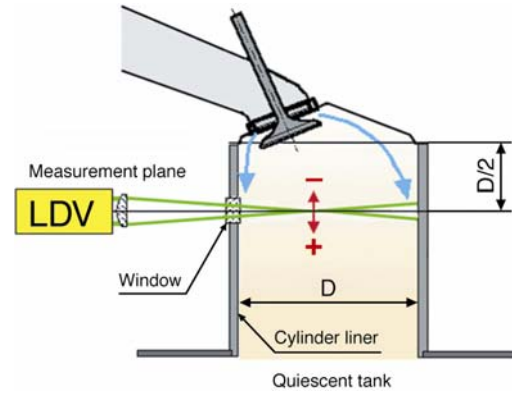


Fig. 3 Schematic diagram of LDV measurement

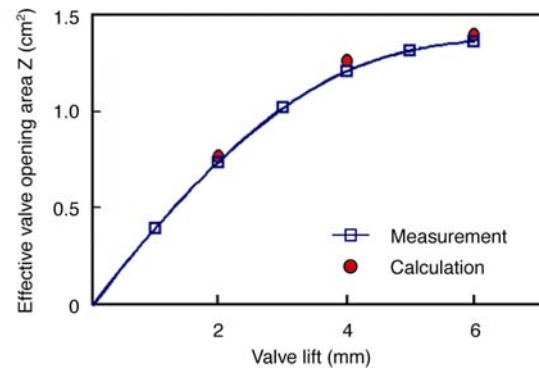


Fig. 4 Comparison of effective valve opening area between measurement and calculation

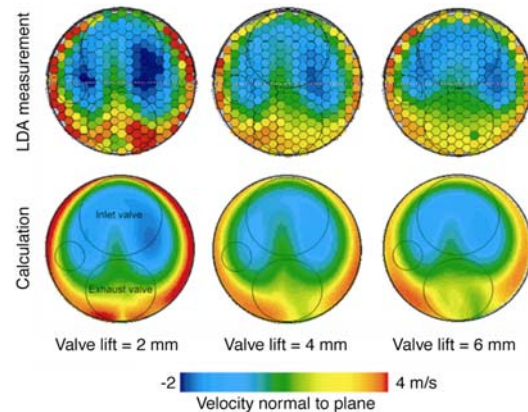


Fig. 5 Comparison of flow field structure on horizontal plane between measurement and calculation

3.4. Verification of Flame Propagation Correlation

3.4.1. Flame behavior observation apparatus

A Tomographic Combustion Analyzer⁽¹⁰⁾ (TCA) produced by AVL List Co. was used to verify the flame behavior. The TCA uses one hundred and several tens of optical fibers arranged in a circle in the cylinder head gasket for measuring the luminescence intensity emitted by the combustion flame. The obtained luminescence intensity signal is used to observe the behavior of the flame. Figures 7 and 8 show the measurement concept and configuration of the apparatus, and Fig. 9 shows the special gasket containing the optical fibers. The apparatus has the following features.

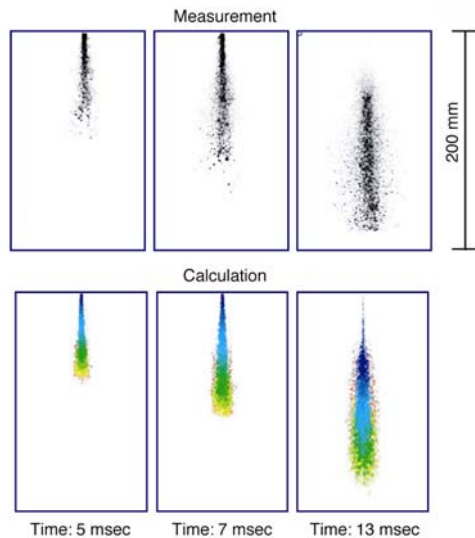


Fig. 6 Comparison of spray form between measurement and calculation (Ambient pressure)

- (1) It enables the actual engine to be used without modification, so there is no change in the combustion condition due to the visualization.
- (2) It has high strength and durability, enabling the flame to be observed over a long period under a wide range of engine operating conditions.
- (3) It enables quantitative flame observation and data processing to be performed in a short time.
- (4) It is necessary to prepare an individual special gasket for each engine bore diameter.
- (5) It is possible to observe only the 2-dimensional flame propagation structure on the cross-sectional face of the gasket.

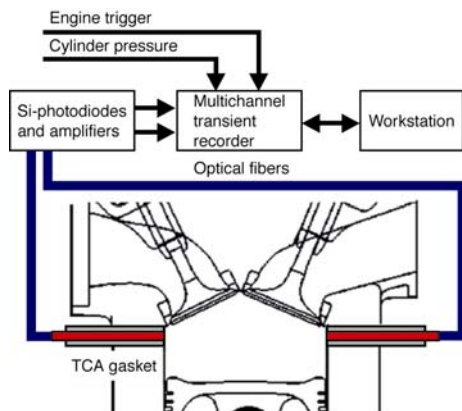


Fig. 7 Schematic diagram of TCA measurement

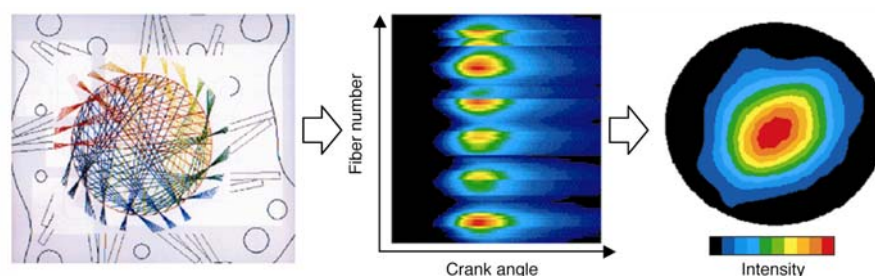


Fig. 8 Schematic flow of TCA measurement

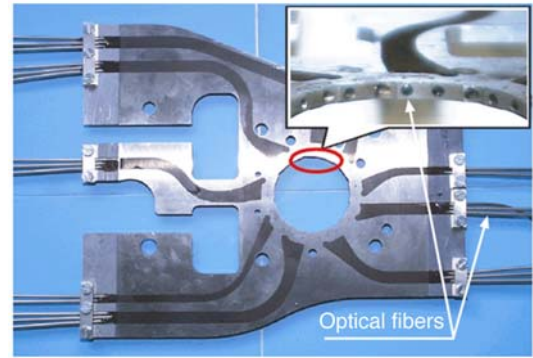


Fig. 9 Photograph of TCA gasket

Compared to conventional visualizing methods, the TCA method has many advantages. However, it is considered to be disadvantageous compared to visualization by CFD, because of the relatively small amount of data obtained by analysis and the high cost of analysis; therefore, in this research it was used only for the purpose of performing a correlation investigation.

3.4.2. Comparison of results

Figure 10 shows the spread of the flame surface obtained by simulation. The operating conditions were an engine speed of 4 900 r/min, an equivalence ratio of 1.13, a brake mean effective pressure of 135 kPa at a partial load, and an ignition timing of 20° BTDC. From the calculation results, it can be seen that the expansion of the initial flame surface at 10° ATDC agrees closely with the experimental results. At 20° ATDC where the combustion reaction increases, the experimental result of the flame speed in the direction of the inlet valve was high, whereas the calculated result was high in the direction of the center of the squish area, thus indicating that there were also parts where the calculated and experimental result were not consistent with each other. At 30° ATDC where the flame surface reaches the squish area furthest from the ignition plug, the phenomenon, whereby the flame propagation to the squish center is fast and conversely the flame propagation toward both ends of the squish is slow, was well simulated. Overall, it was confirmed that the macroscopic spread of the flame surface was consistent with the experimental results obtained using TCA.

Next, the comparison results of the cylinder pressure history (Fig. 11) and the mass fraction burned (Fig. 12) are shown below. As comparison conditions, the engine load was changed in several patterns, and the test was carried out under different combustion conditions. From the calculation results, the early combustion duration tended to be shorter under all load conditions. This tendency was seen especially at a high engine speed and a high load. This is due to the fact that the eddy dissipation model does not include the

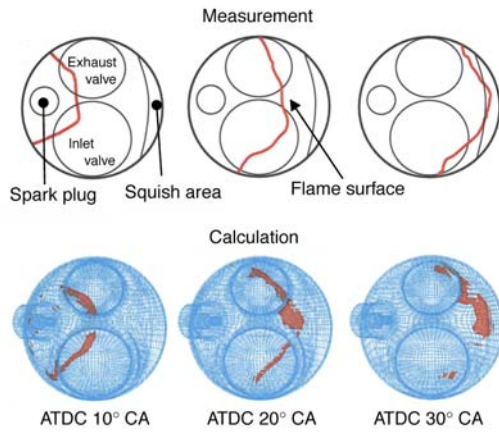


Fig. 10 Comparison of flame propagation between experiment and calculation (4 900 r/min, POT, $\phi=1.13$, Ignition timing=20° BTDC)

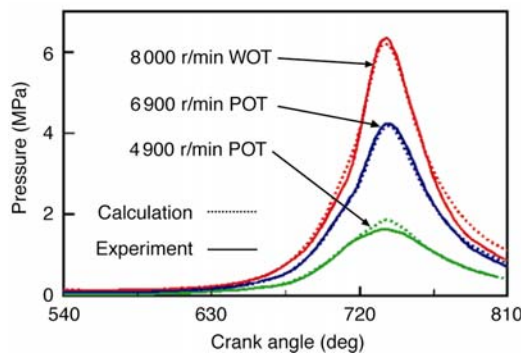


Fig. 11 Comparison of cylinder pressure between experiment and calculation

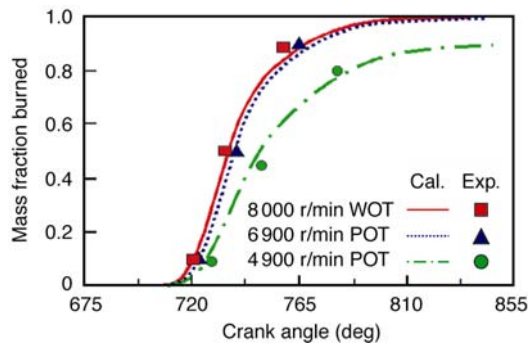


Fig. 12 Comparison of MFB between experiment and calculation

generation of activated chemical species, which occurs at low temperature during the ignition delay period, and the subsequent abrupt high temperature oxidation reaction. It is considered necessary to re-select the combustion model under combustion conditions such that the reaction under delayed ignition greatly affects the main combustion formation. However, it can be said that the selection of this model is appropriate if the main purpose of the model is visualizing flame propagation. Overall, the maximum generated pressure and the crank angle are roughly equal, and there is good correlation between the experimental results and the calculated results.

From the above verified results, it was confirmed possible to estimate the flame propagation behavior in a small displacement motorcycle engine by performing a simulation based on CFD.

4. Case Study Applicable to Engine Development

4.1. Improvement of the Lean Limit

Generally, cylinder pressure cycle fluctuations are evaluated according to their deviations. When the mixture ratio is made leaner, combustion becomes unstable and the pressure deviation becomes larger. As a result of the vibration and noise levels increasing, the lean limit is assessed by the combustion coefficient of variation, which is the deviation divided by the mean value of the cylinder pressure. This sub-section describes a case study in which CFD is used to visualize the flame propagation in order to analyze the cause of combustion fluctuations and to improve the lean limit.

The engine used in the test was a 4-valve, small displacement engine that had a pent roof type combustion chamber and a center plug layout. It was designed under the concept of lean combustion by means of a strengthened tumble flow, however as shown in Fig. 13 the combustion fluctuation of the prototype engine deteriorated abruptly when the equivalence ratio was 0.7 or less, which prevented the lean limit from being expanded (Fig. 14). For this reason, CFD was used to visualize the flame propagation so the cause could be investigated.

Figure 15 shows the results of analyzing the flame propagation both under partial load and under full load. It can be seen that when the engine is operating under full load, the flame spreads concentrically in a circle centered about the ignition plug, and conversely when the engine is operating under partial load the initial flame elongates in the direction toward the exhaust valve, thus indicating that the flame propagation is not uniform.

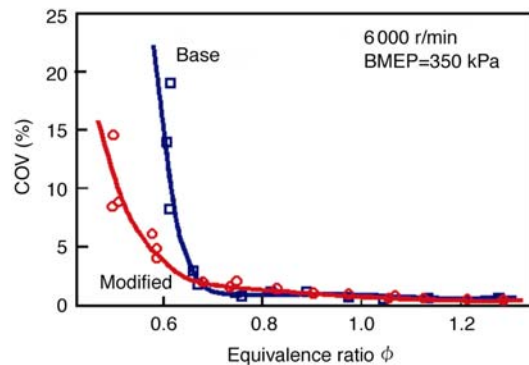


Fig. 13 Comparison of coefficient of variation between base engine and modified

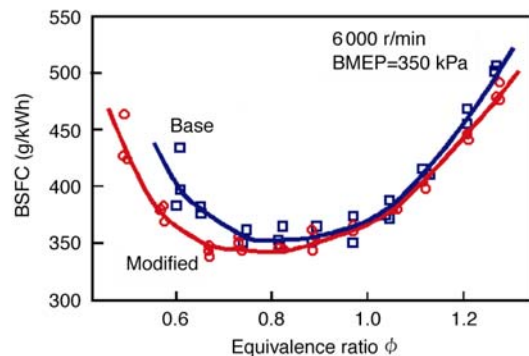


Fig. 14 Comparison of fuel consumption between base engine and modified

Figure 16(a) shows the distribution of the flow gas velocity vector in the cylinder before and after the ignition timing when the engine is operating under partial load. Figure 16(b) shows the distribution of the equivalence ratio. The tumble flow does not collapse even when the ignition timing (BTDC 45°) is reached, and the ignition plug is exposed to a bulk flow. Also, the gas mixture forms a rich band along the bulk flow, so it is thought that this causes the flame to propagate in the direction of the exhaust valve. This situation is extremely unstable for the early flame, and is likely to result in a misfire, so it was judged to be the main cause of combustion fluctuations. It is thought that a misfire does not occur under a full-load because the ignition delay period is shorter than that for partial load operation and also the ignition timing is 15° retarded, so that disintegration of the tumble flow proceeds and the flow velocity falls down.

Next, a study for a countermeasure was carried out using CFD. As a means of restraining the flow velocity around the ignition plug, which is the cause of flame elongation, a method⁽¹¹⁾ of slanting the axial center of the tumble flow has been reported by Kaneko and others. However, in this research, a study of a countermeasure proceeded without a control device. The flow velocity of the vertical eddy remaining at the ignition timing was high along the wall face, however it was slow around the center of the eddy. Attention was focused on these facts, and the ignition plug was made to protrude 2.5 mm, aiming at reducing the flow velocity.

However, the closer one gets to the center of the eddy, the leaner the gas mixture becomes, which makes a misfire more likely to occur. To overcome this, the initial timing of the fuel injection was advanced, and an adjustment performed so that an ignitable gas mixture band would be formed in the vicinity of the ignition plug. Figures 17 and 18 show the comparison of the calculated history of the flow velocity and the equivalence ratio around the ignition plug before and after the countermeasure was adopted. It was found that the flow velocity at ignition timing could be reduced from 18 m/s to 15 m/s, and also obtained the same equivalence ratio as before the countermeasure by advancing the start of the fuel injection timing. From these results, it was confirmed in advance that a countermeasure by which misfire did not readily occur could be investigated by using simulation. Next, physical validation was performed on an actual engine with the specifications of the proposed countermeasure. As a result, it was found that the lean limit could be expanded to about 1 in terms of an equivalence ratio, and the fuel consumption rate improved by about 3%.

4.2. Study of the Injection Direction

It is well known that the emission values vary according to the installing position and angle of the injector. Particularly, it is known empirically that exhaust hydrocarbon (HC) can be reduced by directing the fuel spray at the port wall to collide with the wall. Previously, a study of these

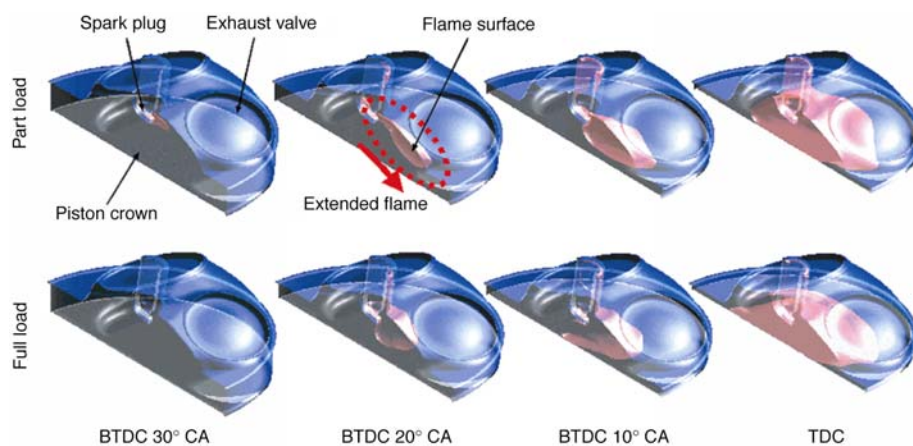


Fig. 15 Calculation results of flame propagation

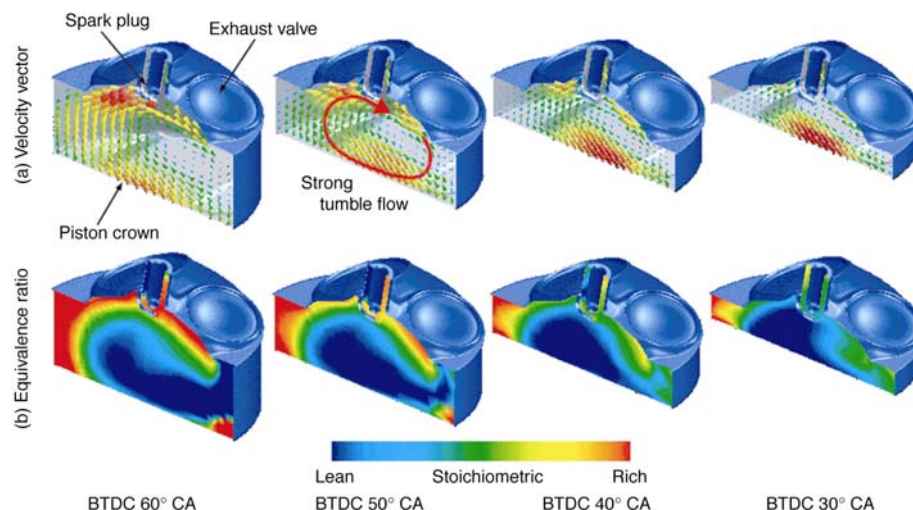


Fig. 16 Calculation results of gas behavior: (a) Gas velocity vector (b) Equivalence ratio distributions

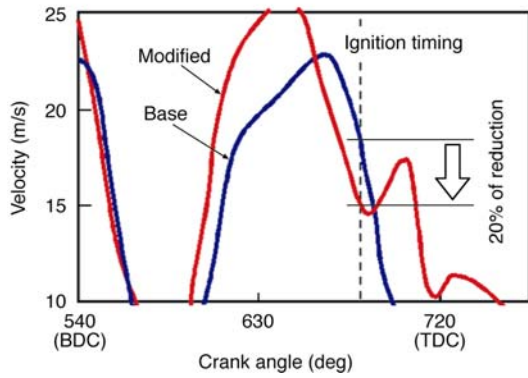


Fig. 17 Time history of mean velocity around spark plug

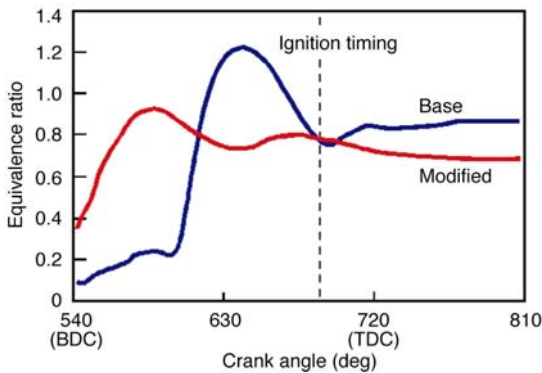


Fig. 18 Time history of equivalence ratio around spark plug

detailed specifications was carried out each time a new engine was being developed. This sub-section describes an example in which CFD visualization was used to verify the mechanism of reduction of unburnt HC, thereby facilitating the selection of the specifications.

The test engine was a 4-valve engine with an injector at an intermediate position on the inlet port. The injection angle of the injector was initially set at 48° (Type 1), aiming at the center of the inlet valve head, then to Type 2, which is 10° higher, Type 3, which is 20° higher, and so on, while keeping the collision of the spray with the port wall to a minimum, and the amount of HC at each angle was measured. The results of the measurements are shown in Fig. 19.

It was found that HC decreased as the injection angle was increased and became a minimum at Type 3, and then it started to rise again gradually as the injection angle was increased further. From this trend, it was found that the difference in the blow-out behavior of the gas mixture when the valves overlap was more dominant than the worsening of HC resulting from the inflow into the combustion chamber of the fuel liquid film at the bottom wall of the port, which increases as the injection angle is raised. In order to validate this trend more qualitatively, CFD was used to visualize the blow-out behavior of the gas mixture and to enable a detailed understanding of the phenomenon to be obtained. As the analysis condition, a partial load equivalent to a constant speed of 50 km/h was applied. Figure 20 shows the distribution of the equivalence ratio for each specification obtained by calculation. In case of Type 1, there is a rich gas mixture near the valve seat before the inlet valve opens (ATDC -26°). Part of this gas mixture is blown off to the exhaust port after the inlet valve opens, and is exhausted as unburnt HC together with the burnt gas.

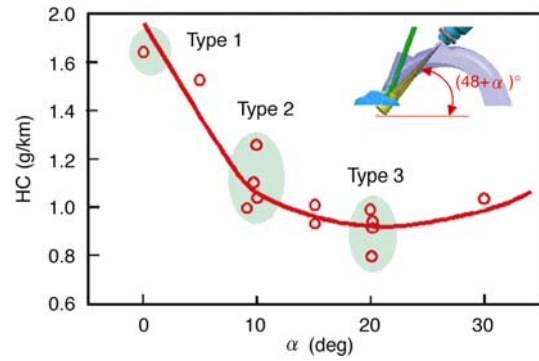


Fig. 19 Effects of injection angle on emission

In the case of Type 2 and Type 3, all of the injected droplets collide with the bottom wall of the port, and are either reflected or become a wall-film. Consequently, almost no droplets directly reach around the valve seat. The fuel that becomes a liquid film is warmed and gasified by the wall temperature of the inlet port, however a delay in reaching around the valve seat occurs. Consequently, it was confirmed that when the inlet valve is open, a rich gas mixture does not exist around the valve seat, and the amount of HC blown off to the exhaust port is less. From the above analysis, it was possible to understand the characteristics of the HC according to the direction of the injector, and it became easier to perform the specification selection.

4.3. Application to the Development of the First 4-stroke 50 cm³ PGM-FI Engine

This technology was used in the development of the first 4-stroke 50 cm³ PGM-FI engine in the world. The resulting engine was installed in a scooter for the Japanese market in the spring of 2004. The new engine (Fig. 21) has 8% higher maximum output and 10% better fuel economy in the actual running mode (in-house test value comparison that assumes actual conditions of use) than the 2-valve, twin carburetor specification engine of the Crea SCOOPY, on which this machine is based. Table 2 shows the main specifications of the engine.

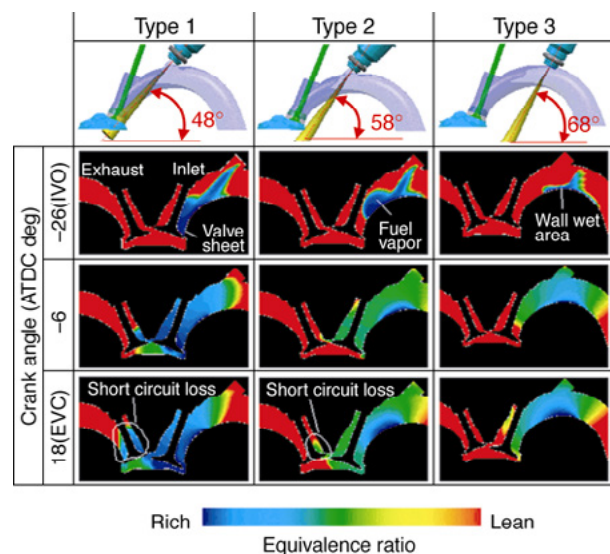


Fig. 20 Calculation results of equivalence ratio distributions



Fig. 21 Cutaway model of 50cm³ four stroke engine with PGM-FI system

Table 2 Developed engine specifications

Engine type	4-stroke single cylinder, spark ignition
Bore × Stroke (mm)	38 × 44
Displacement (cm ³)	49.9
Valve layout	2 inlet and 2 exhaust valve
Compression ratio	12.0
Combustion chamber	Pentroof type

5. Conclusion

A combustion simulation based on CFD was carried out in order to visualize the inside of the cylinder of a small motorcycle engine that features small displacement and high engine speed. Experimental investigation of correlation was performed and the effectiveness of this simulation was set forth below.

- (1) The results of simulating the steady-state flow in the cylinder and the behavior of the fuel spray generally were consistent with the measured results.
- (2) Under the common-use operating load conditions of a small 4-stroke motorcycle engine, the macroscopic flame propagation behavior can be estimated by using an Eddy Dissipation Model as the combustion model. In order to estimate the microscopic combustion behavior including a cool flame, which occurs during the ignition delay period, it is necessary to re-select the combustion model appropriate to the particular phenomenon.
- (3) As a result of visualization of in-cylinder behavior using CFD, it was possible to improve the fuel economy and also expand the lean burn zone, thus indicating that it can be applied effectively to the development of an actual mass-production engine.

References

- (1) Sasaki, T., Yoshida, K., Urata, Y., Ohno, T., Ishii, K.: Introduction of Optically Access Engine to Investigate in Cylinder Phenomena, HONDA R&D Technical Review, Vol. 10, p. 78-89 (1998)
- (2) Yamada, T.: Current Status of Development and Application of CFD, ENGINE TECHNOLOGY, Vol. 04, No. 05, p. 10-11 (2002)

- (3) Sadakane, S., Abe, S.: Present and Future of Direct Injection Gasoline Engine, Journal of Society of Automotive Engineers of Japan, Vol. 57, p. 49-50 (2003)
- (4) Naitoh, K., Itoh, T., Takagi, Y., Kuwahara, K.: Large Eddy Simulation of Turbulent Premixed-Flame in Engine by Using the Multi-Level Formation, JSME, B, 59-559, p. 289-295 (1993)
- (5) Miyagawa, H., Kojima, S., Katsumi, N., Ueda, T., Okumura, T.: Numerical Analysis of the Effects of Squish Geometry on a Newly Developed 4-Valve Gasoline Engine Combustion Process, International Symposium COMODIA 98, p. 227-232 (1998)
- (6) Dokowicz, J. K.: Particle-Fluid Numerical Model for Liquid Spray, J. of Computational Physics 35, p. 229-253 (1980)
- (7) O'Rourke P. J., Amsden A. A.: The Tab Method for Numerical Calculation of Spray Droplet Breakup, SAE Paper 872089 (1987)
- (8) Magnussen, B. F.: On the Structure of Turbulence and a Generalized Eddy Dissipation Concept for Chemical Reactions in Turbulence Flow, 19th AIAA Sc. Meeting (1981)
- (9) Takahashi, Y., Murakami, Y.: Establishment of an Intake Port Design Support Tool Using CFD, Honda R&D Technical Review, Vol. 15, No. 1, p.104 (2003)
- (10) Philipp, H., Plimon, A., Femitz, G., Hirsch, A., Fraidl, G., Winklhofer, E.: A Tomographic Camera System for Combustion Diagnostics in SI Engines, SAE Paper 950681 (1995)
- (11) Kaneko, M., Kato, S., Saisyu, Y.: Combustion Control Technologies for SI Engines by Controlling Tumble Generation and Dispersion, The 17th Internal Combustion Engine symposium, Japan (2002)

■ Authors ■



Yasuo MURAKAMI



Hitoshi KUROSAKA