CFD Technology for Formula One Engine

Naoki HANADA* Atsushi HIRAIDE* Masayoshi TAKAHASHI*

ABSTRACT

Simulation technology has advanced markedly in recent years, and various types of CFD models have come into use for Formula One engine development.

However, to use such simulation modeling it is necessary to establish simulation technology for the unique conditions for Formula One engines, such as that for high engine speed.

The pressure of each engine part was measured and in-cylinder gas motion and fuel spraying behavior were also measured using a single-cylinder optical engine, enabling CFD technology that can be applied to Formula One engines to be created.

1. Introduction

Simulation technology has advanced markedly in recent years. Even in many fields for Formula One, simulation technology has been established and has come into use for development. CFD requires extensive computer resources and has become more practical as hardware and computing technology have advanced.

CFD modeling uses a variety of sub-models such as turbulence models, and it is crucial to validating whether the phenomenon being calculated can be expressed. For example, in the case of automobile reciprocating engines, which use a wide range of operating speeds, the pressure fluctuation within the intake and exhaust pipes will differ at low engine speeds and high engine speeds. A calculation technique validated for just low engine speeds or just high engine speeds would not necessarily be compatible with the other. Dependable validation of simulations helps make it possible to create CFD technology that is effective for development.

CFD technology for commercial engines[(1)-(5)] has been developed by members of a special team. At first, since the Formula One engine is also a reciprocating engine, it was thought that the CFD technology should also be compatible with the Formula One engine as is, and the attempt was made to apply the technology to Formula One engine development. When actual simulation was performed, however, it became clear that the technology could not express Formula One engine phenomena. Since commercial engines are normally used at low engine speeds, low engine speeds of about 2000 rpm are an important analysis condition of simulations too. Since the highest priority of a Formula One engine is output, it is necessary to conduct evaluation when the engine speeds are high, as high as 20000 rpm. At high engine speeds, several factors affect output: increased piston speed and gas flow rate, fuel spray behavior and cylinder interference in the intake and exhaust systems resulting from pressure fluctuations within the intake and exhaust pipes, the motion of gas coming into the cylinders, and so on. We began work on developing a simulation model that could express these impact parameters.

Using data from pressure measurements for each part and in-cylinder measurements taken with an optical engine, a simulation model usable for the development of Formula One engines was validated.

This paper explains CFD technology for the flow of air during intake and exhaust, in-cylinder gas motion, in-cylinder fuel behavior, and combustion.

2. Intake and Exhaust Systems

The intake and exhaust systems of a reciprocating engine impact volumetric efficiency and as such are important components determining engine output.

Being able to analyze pressure and gas flow of intake and exhaust systems and predict performance in advance provide effective ways of determining specifications in a short period of time when developing Formula One engines, where enhancements are expected on a daily basis. To achieve such predictions of performance, it is necessary to perform simulation studies in advance, and the results should be cross-checked with the measurement data.

Intake and exhaust system simulations were originally performed using one-dimensional gas dynamics and engine performance simulation software WAVE made by Ricardo Software. Subsequently, for the intake system,

* Automobile R&D Center

an in-house software capable of three-dimensional compressible fluid analysis was used, with the aim of enhancing calculation accuracy. For the exhaust system, usable three-dimensional analysis software could not be found and so we continued to use WAVE.

2.1. Intake System

The current Formula One engine intake system is equipped with an airbox, and the pressure fluctuation (pulsation) that occurs within the intake pipe of each cylinder propagates and reflects inside the airbox and is transmitted to other cylinders. As a result, airbox shape is one factor affecting engine output.

On the other hand, the body's aerodynamic performance greatly affects lap time. Therefore, the shape of the body cowl is determined by aerodynamic characteristics. As for airbox shape, the degree of freedom is restricted by the shape of the cowl.

Using WAVE, at first, performance was evaluated with the air intake form only, but in order to achieve both body aerodynamic performance and engine output, a large amount of time was spent during development on selecting the airbox shape. We tackled this issue so that simulations could be utilized for performance evaluation and phenomenon analysis including also the influence of airbox shape.

The simulation software used was Honda's three-dimensional compressible fluid analysis software[(1)] in which a one-dimensional model is coupled.

The simulation model is shown in Fig. 1. The airbox portion was expressed in three-dimensional form, and the portion from the downstream part of the air intake throttle to the cylinder and exhaust pipe was expressed by one-dimensional and scalar models.

The greatest feature of Honda's software is that it solves compressible fluids by not only one-dimensional but also three-dimensional calculation.

Although it is possible to do similar calculations by combining Ricardo Software's three-dimensional fluid analysis software (VECTIS) with their one-dimensional gas dynamics simulation software (WAVE), these software programs did not satisfy our target standards of three-dimensional compressible fluid calculation accuracy.

Based on the measured intake pipe pressure, boundary conditions, mesh density, flow coefficient, filter model and so on were investigated and a series of enhancements were made until a simulation that could be used for the analysis of intake interference in the airbox of a Formula One engine was created.

Figure 2 shows measurement and calculation results of pressure within intake pipes in each of the cylinders in one bank of a V10 engine. Engine speed was 17000 rpm, and the crank angle of 360 deg is top dead center of intake stroke. Intake interference within the airbox causes the different intake pressure in all the cylinders. While the amplitude and phase of pressure varies with

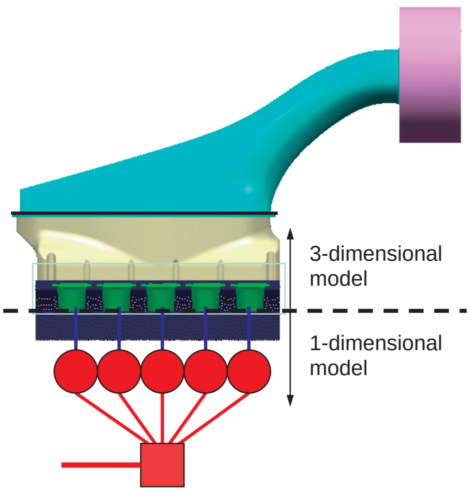

Fig. 1 Simulation model

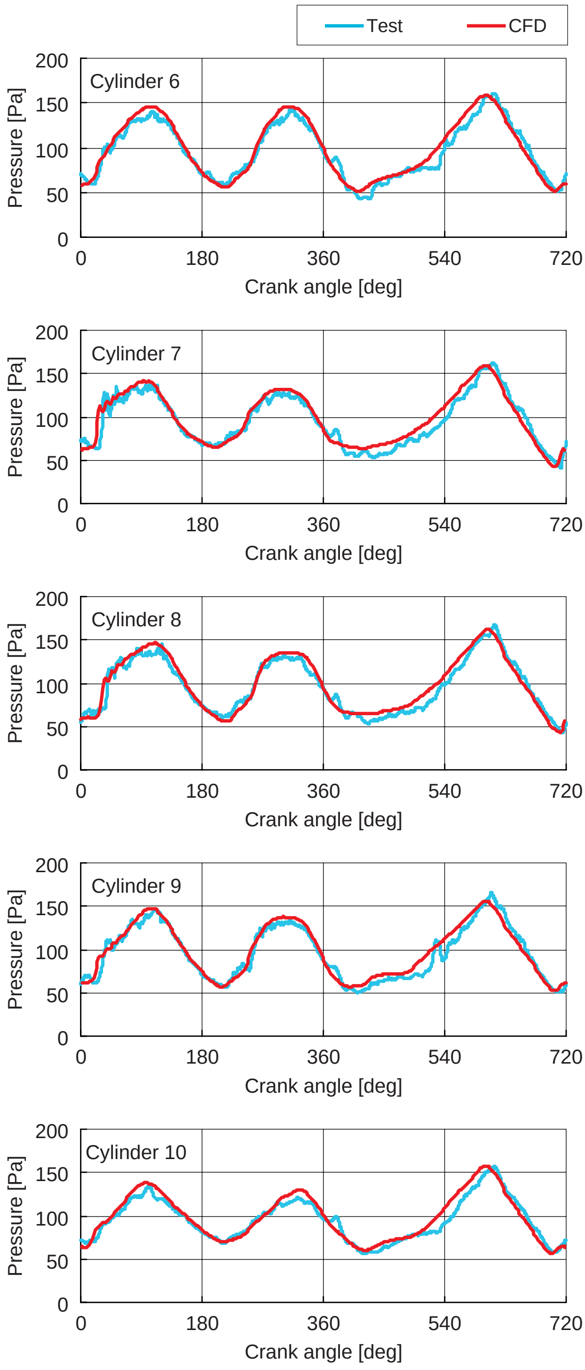

Fig. 2 Intake pressure of V10 engine at 17000 rpm

Table 1 Parameters for WAVE simulation

Parameter	Old value	New value
Flow coefficient at pipe end (inflow)	Auto	0.8
Flow coefficient at pipe end (outflow)	Auto	0.5
Ambient temperature of exhaust side	Atmospheric temperature	400 K
Multiplier of heat transfer of cylinder wall when intake valves are closed	1.0	2.0

each cylinder, there is a particular difference between cylinder 10 and the other cylinders. It can be seen that intake interference causes the volumetric efficiency of each cylinder to change and has a direct impact on engine output.

The calculation results reproduce these characteristics with good accuracy, demonstrating that Honda's software is capable of analyzing intake interference within the Formula One engine's airbox.

The slim and curving airbox shape required for good body aerodynamic performance is due to reduced volume and is characterized by a weak attenuation of pressure fluctuation and lowered volumetric efficiency. However, because practical simulation has been achieved, it is now possible to satisfy both a slim airbox shape which can contribute to body aerodynamics and engine output enhancement by optimizing air flow in the airbox.

2.2. Exhaust System

WAVE was used to analyze the exhaust system and examine its shape.

This explanation uses a V10 engine as an example. A 5-into-1 collector exhaust system, as shown in Fig. 3, was used. The WAVE model is shown in Fig. 4. Because the effect of intake interference in the airbox cannot be expressed in one-dimensional simulation WAVE, a model used for the intake side was designed such that intake pipes are open to the air, with each cylinder being independent. Because the left and right exhaust systems are independent, the WAVE model used one bank with five cylinders.

Figure 5 shows V10 engine validation results.

Exhaust pressure was measured at a downstream position approximately 40 mm from the exhaust valve seat of cylinder 10. Because the calculation model has five cylinders, the data used was of cylinder 5. The initial calculation results (b) differed from the measured results (a).

A parameter study to revise the variables used for simulation shown in Table 1 yielded the results shown

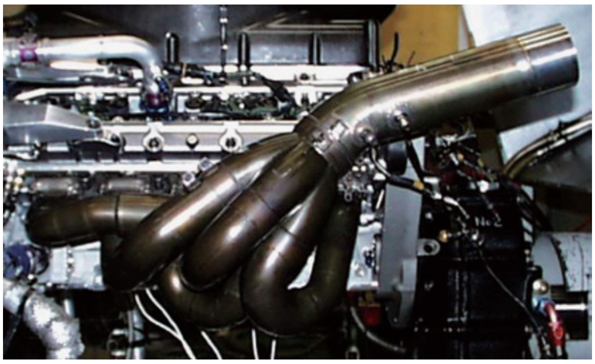

Fig. 3 Exhaust pipe of V10 engine

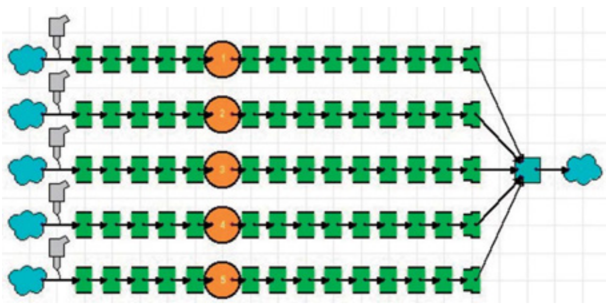

Fig. 4 WAVE model of 5 cylinders for V10 engine

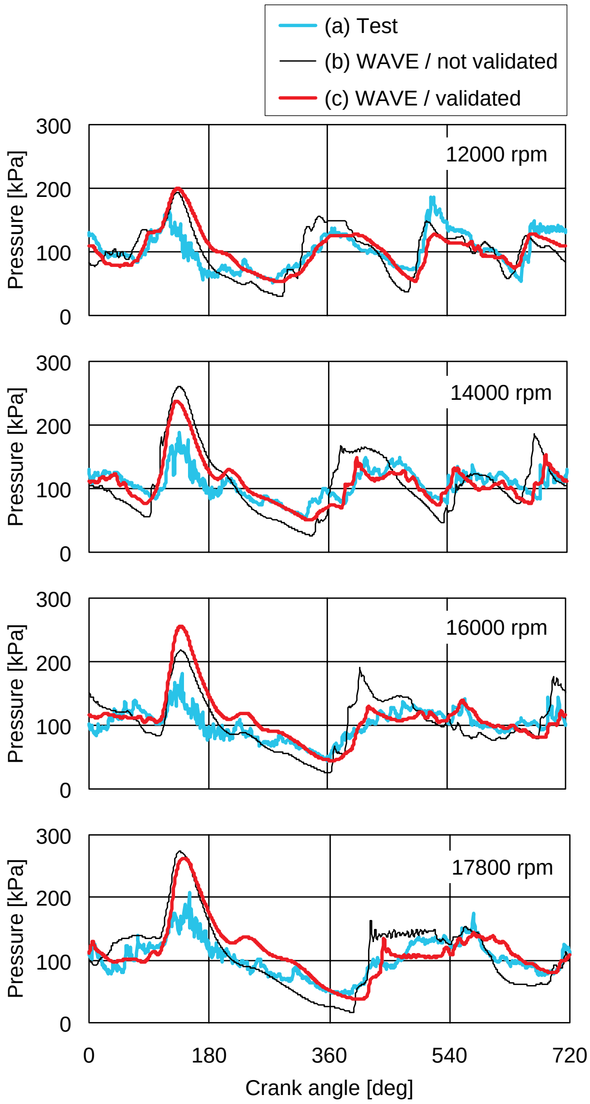

Fig. 5 Exhaust pressure of V10 engine (#10 cylinder)

in (c) in Fig. 5, with enhanced calculation accuracy. The important revised parameter is the flow coefficient for the end of the exhaust pipe. In the WAVE calculation, exhaust pressure changes greatly depending on the value of the flow coefficient, and ultimately volumetric efficiency and output values also change. Estimating the flow coefficient correctly is clearly the most important aspect of one-dimensional simulation.

In addition, this result shows that there are limits to analyzing exhaust systems with one-dimensional simulation. One-dimensional analysis has many subjects to be addressed such as followings. Naturally, it is not possible to estimate pressure loss in each part with one-dimensional analysis. General usefulness of flow coefficient is low for changes in the form of the exhaust pipe or changes in flow velocity. To some extent it is possible to analyze and examine flow within exhaust pipes with one-dimensional simulation, so this can be used in the development of exhaust systems. However, the calculation accuracy is not necessarily sufficient; for example, revision of the flow coefficient is inevitable when engine specifications change. Development of three-dimensional simulation, as used with the intake system, is needed in the future as a way to resolve this issue.

3. In-cylinder Behavior

Enhancing the combustion status is indispensable for achieving engine performance enhancements. Besides that, the current Formula One engine has longer combustion duration than commercial engines because of its high engine speed and big bore. Therefore, shortening the combustion duration is necessary.

To shorten combustion, it was considered necessary to adjust the in-cylinder gas motion and fuel distribution. To do this, it is indispensable to be able to predict in-cylinder gas motion and simulate fuel spray and mixture distribution formation.

3.1. In-cylinder Gas Motion

Validation on in-cylinder gas motion has been performed on commercial engines and an attempt has been made to apply the results to Formula One engines, but no correlation was found to actual engines. Thus we started over again and redid the process from validation of the Formula One engine's in-cylinder motion.

Figure 6 gives an illustration of the system for measuring in-cylinder motion. An optical engine(6) was used. Two high-repetition-rate YAG lasers were used with a YAG 532 nm second harmonic wavelength. The camera used was Vision Research's Phantom V7, and resolution was 656 × 328 pixels. Hollow resin tracers were used, with particle size of ϕ 40 µm and density of 36 kg/m^3. Based on tracer size and density, it is thought that air flow traceability will have a 95% attenuation rate for fluctuations of approximately 300 Hz. The measurement data took the mean for 100 cycles.

The simulation software was three-dimensional fluid analysis software (VECTIS). Figure 7 shows the simulation model. To correctly express intake and exhaust pulsation, the calculation covered the entire single cylinder engine from air inlet chamber to exhaust pipe.

Table 2 Calculation conditions of in-cylinder gas motion

Calculation region	Whole engine
Number of calculation cycle	4 cycles
Mesh density (for combustion chamber)	1st to 3rd cycle: 2.0 mm 4th cycle: 1.5 mm (Half the above size is used near the wall.)

The calculation conditions are shown in Table 2. These conditions were selected to enable calculation results to reproduce PIV measurement results and at the same time cause intake pulsation, volumetric efficiency and in-cylinder turbulence energy to converge. The reason that mesh density is changed with the number of cycles is to ensure both calculation accuracy and reduced calculating time. Except for turbulence intensity, the calculation results showed no differences between 2 mm and 1.5 mm of mesh densities, but for turbulence energy, changes in cycles could not be expressed at mesh density of 2 mm. For that reason, calculating time was saved by using a coarser mesh to calculate the first three cycles, and then a finer mesh was used on the fourth cycle to ensure calculating accuracy.

Figure 8 shows visualization and simulation results in a motored condition at 10000 rpm. The crank angle shown in the figure is the angle after top dead center. Measurements were performed through a cross-section

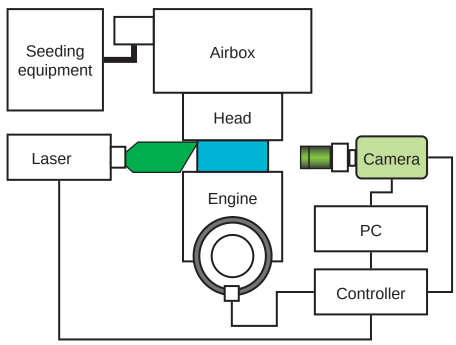

Fig. 6 Schematic of PIV measurement system with optical single cylinder engine

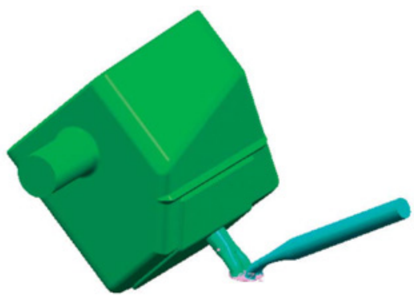

Fig. 7 Simulation model of in-cylinder gas motion

under the intake valve, as shown in the upper right of the figure. Two intake pipe lengths of 200 mm and 150 mm were used to compare differences in gas motions. The left column is from the 200 mm intake pipe, and the right column is from the 150 mm intake pipe.

In-cylinder motion in a Formula One engine consists of two vortexes, as demonstrated by the example of crank angle 216 deg ATDC: a large vortex centered on the bore, and a vortex that forms on the bottom of the intake port. The images show that the twin vortex configuration does not change with the difference in intake pipe length, but the gas inlet velocity, vortex center and vortex speed are affected by intake pipe length. This means, in other words, that intake pulsation has an impact on not only volumetric efficiency but also combustion.

Calculation results showed a flow pattern that closely matches visualization results. This demonstrates that if a model shape and calculation conditions are chosen such that intake and exhaust pulsation can be expressed, it is possible to accurately simulate in-cylinder gas motion at the high speed of 10000 rpm.

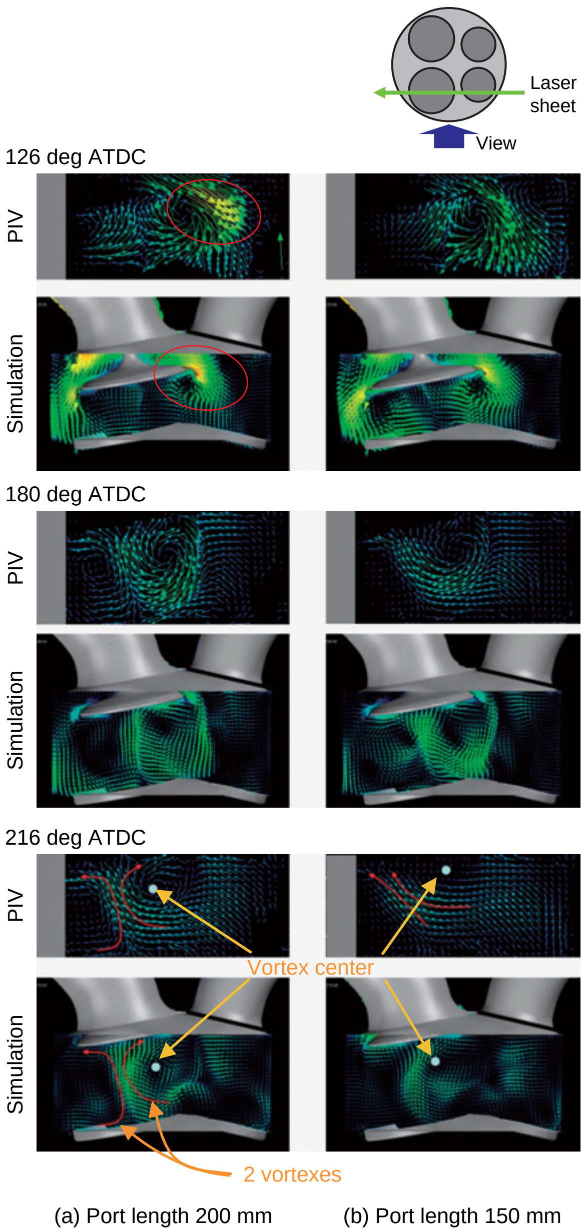

Fig. 8 Comparison of flow vectors between PIV measurement and simulation results at 10000 rpm

These initiatives have made it possible to predict gas motion in a Formula One super-high-speed engine. In addition, this research has renewed the researchers' awareness of the importance of measurement that helps one to grasp phenomena when creating simulation technology. Without PIV measurement results, what to set as the number of cycles and mesh density cannot be determined.

The fact that turbulence intensity is not validated is an issue, and analysis of this, including its impact on combustion, should be performed.

3.2. In-cylinder Fuel Behavior

The distribution of fuel within the cylinder is one factor affecting the quality of combustion, and it is important to predict this in advance with simulation. However, there are issues with measuring fuel distribution within cylinders, and measurements are not easy. For that reason, analysis by simulation is valuable in terms of advancing phenomenon analysis.

Injectors of Formula One engine are typically installed close to the end face of the trumpet. The reason is because volumetric efficiency is increased as a result of charge cooling by evaporation latent heat of the fuel. Because of this installation, fuel injected toward the interior of the intake port is affected by gas motion which fluctuates because of intake pulsation, and then it enters the cylinder. Therefore, to calculate in-cylinder fuel distribution, the necessary calculations must include fuel behavior inside the intake port.

Simulation validation was performed with the behavior of the fuel droplets flowing into the cylinder which was photographed with an optical engine. Using two kinds of injectors, it was evaluated whether the simulation can express the difference produced from the difference in the fuel spray characteristic. One was a pintle form with fuel pressure of 1.2 MPa, and the second was a six-hole plate form with fuel pressure of 10 MPa. These were characterized by respective Sauter's mean diameters of approximately 40 μm and 16 μm, respectively.

Using an optical engine[(6)], spray droplets were filmed directly with a strobe as a light source. Operating conditions were: motored at 10000 rpm, and wide open throttle.

For simulation, VECTIS was used, the same as for in-cylinder motion calculation. An entire single cylinder engine was modeled and a fuel spray calculation was added. To calculate fuel spray, the following method was used. The state of spraying with the injector alone was matched with measurement results in advance. Then, the fuel spray calculation conditions were input into the single cylinder engine calculation.

The fuel spray form of the injector alone is shown in Fig. 9.

This shows that with both the 1.2 MPa and 10 MPa injectors, the calculations were able to sufficiently express the actual spray form.

It was learned that for in-cylinder mixture forms to converge in calculations of fuel spray in the engine, about five cycles are necessary. Figure 10 shows the cycle history of in-cylinder average A/F and A/F variance when calculating sprays in an engine using a 10 MPa injector. Fuel injected from the trumpet end face begins to enter the cylinder on the second cycle after injection, but the amount is small. This figure can be read as showing in-cylinder fuel behavior converging from the fourth to fifth cycle. To give some margin, it was decided to conduct the analysis with five cycles.

Figure 11 shows the results of photographing in-cylinder spraying and the calculation results.

The measurement results are colored to represent the degree of photographed brightness. In the calculation results, spray droplets are represented by dots, and the broken line shows the approximate area in which the droplets occur.

With the pintle, the fuel spray has large particles and the spray has great inertia. That is why, as the photograph taken at crank angle 120 deg shows, the spray entering the cylinder from the intake valve passes below the exhaust valve and collides with the cylinder wall. In contrast, the high pressure injector spray has little inertia, and after it enters the cylinder, it rides the flow of air down the cylinder toward the piston. In addition, the spray entering from the intake valve enters not only from the exhaust side but also from the port bottom side that is opposite it.

To make the fuel pass through the entire combustion chamber, it would be effective to reduce particle size and inertia.

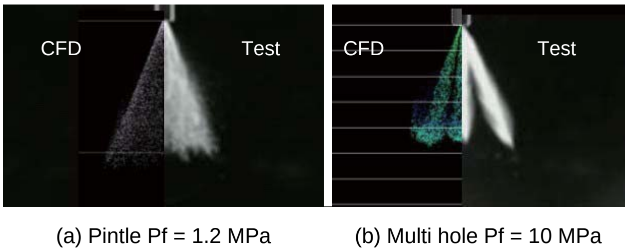

Fig. 9 Fuel spray form

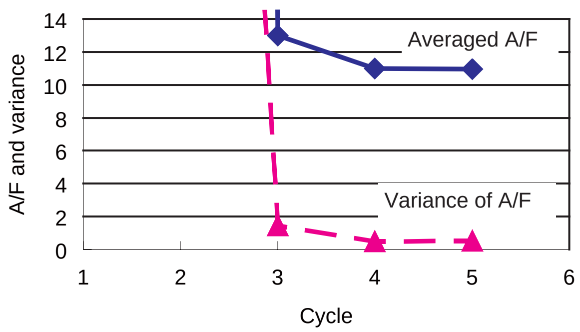

Fig. 10 Simulation convergence of averaged A/F and variance of A/F in cylinder

The simulation largely expressed the measurement results, and by matching the spray with the injector alone beforehand, it was possible to predict in-cylinder fuel behavior. In addition, the fact that gas motion inside both the intake port and the combustion chamber was correctly solved is one factor for accurate calculation.

3.3. Combustion

To directly evaluate the influence of gas motion and fuel behavior on combustion, and to predict amount of residual gas, efforts were focused on combustion simulation as the next step.

The software used was VECTIS, but the combustion simulation module[4] made by Honda was incorporated into VECTIS.

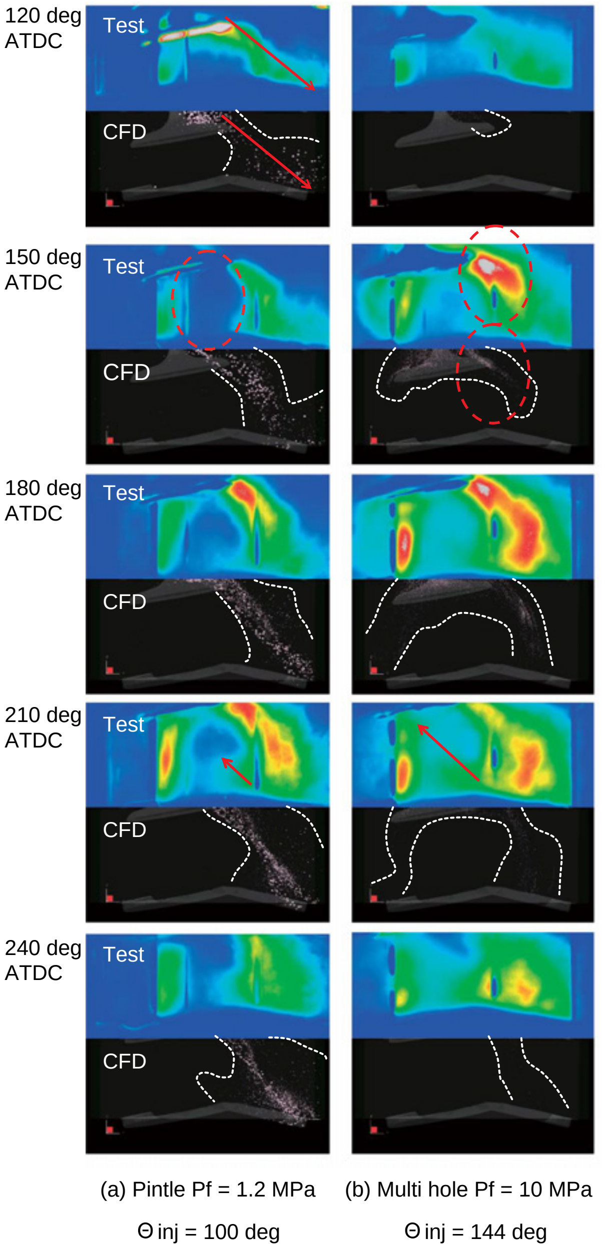

Fig. 11 Comparison of fuel droplets in cylinder between measurement with optical engine and simulation with VECTIS at 10000 rpm

The subject of the simulation evaluation was not the Formula One engine itself, but rather a single cylinder engine with the same specifications as the Formula One engine. The calculation model was the same as the in-cylinder motion and fuel spray model shown in Fig. 7.

As a calculation procedure, at first, in-cylinder motion and fuel behavior in a motored condition were calculated up to the point just before ignition on the fifth cycle for convergences of gas motion and mixture concentration. After that, combustion calculation was performed only by a model of the combustion chamber except intake and exhaust elements.

In-cylinder pressure measurements and calculation results for a single cylinder engine at 15000 rpm and 17750 rpm are shown in Fig. 12. In the combustion simulation used, it is necessary to tune a constant that adjusts combustion speed, but the simulation was able to reproduce in-cylinder pressures at different engine speeds with the same constant. However, this is a different value from the constant used for commercial engines and it was confirmed that it is necessary to tune a unique value for Formula One engines. The outlook is that Honda's own combustion simulation can be applied to Formula One engines as well, and the research is at the stage of confirming whether it has general usefulness for an even greater number of operating conditions.

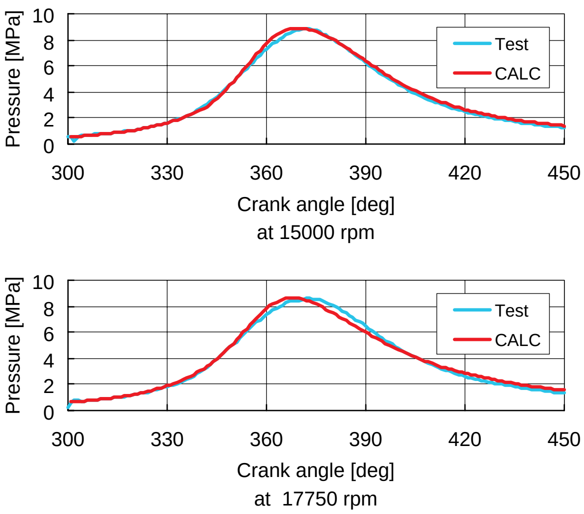

Fig. 12 Comparison of cylinder pressure between measurement and simulation

4. Conclusion

Creating simulation models has been attempted to allow development of Formula One engines with more performance more quickly.

Simulations which use various models cannot demonstrate their ability unless reliable validation is conducted for each subject of calculation.

In the future, we hope to use the model validated and created with Formula One to develop CFD technology for engines that will be friendlier to the global environment.

References

(1) Takabayashi, T., Hotozuka, Y., Tsushima, H.: The Three-dimensional Pulsation Flow Simulation and Its Application for Engine Intake and Exhaust Systems, 2000 FISITA World Automotive Congress, F2000A091 (2000)

(2) Ishikawa, N., Hiraide, A., Takabayashi, T.: Air/Fuel Distribution Simulation in a Port-Injected Gasoline Lean-Burn Engine, SAE Paper, 2001-01-1230 (2001)

(3) Zhu, G., Reitz, R. D., Xin, J., Takabayashi, T.: Characteristics of Vaporizing Continuous Multi-Component Fuel Sprays in a Port Fuel Injection Gasoline Engine, SAE Paper, 2001-01-1231 (2001)

(4) Yang, X., Ohashi, T., Takabayashi, T., Kubota, S., Katsuyama, H., Urata, Y.: Ignition and Combustion Modeling with G-Equation in Spark Ignition Engines, The 13th International Pacific Conference on Automotive Engineering (2005)

(5) Ohashi, T., Yang, X., Takabayashi, T., Urata, Y., Kubota, S., Katsuyama, H.: Ignition and Combustion Simulation in HCCI Engines, SAE Paper, 2006-01-1522 (2006)

(6) Yanagisawa, N., Hanada, N., Hamakawa, T., Ogiyama, k., Goto, T.: Measurement Technologies for Formula One Engines, Honda R&D Technical Review 2009, F1 Special (The Third Era Activities), p. 101-108

■ Author ■

Naoki HANADA

Atsushi HIRAIDE

Masayoshi TAKAHASHI