

Estimation Technology Establishment for Temperature Distributions of Engine Based on Thermal Balance under Vehicle Running Conditions

Tomokazu NOMURA*

Koichiro MATSUSHITA*

Yoshihiko FUJII*

Hirofumi FIJIWARA*

ABSTRACT

The estimation of the temperature distributions of an engine requires consideration on the process of releasing heat generated through combustions to the atmosphere. In this study, the heat inflow/outflow balance was calculated using one-dimensional thermal simulation tool. Using those results as boundary conditions, three-dimensional simulation on the structures of an engine was conducted to establish a method to estimate the heat distributions of an engine. An air-cooled engine, whose cooling conditions are highly susceptible to its configuration and shape, was selected as the target engine for investigation.

The 1D simulation for calculating the heat balance was structured with four specific methodologies. The first one was an engine cycle simulation method that calculates the heat transfer from the combustion gas to the engine structures. The second one was a heat transfer simulation method in which engine components were replaced with simple shaped objects. The third one was a CFD coupled with an empirical formula to calculate the heat transfer to oils situated in various regions in an engine. The last method was a CFD simulation applied to the running conditions of a complete vehicle to calculate the heat transfer to the atmosphere.

Utilizing the results yielded through this one-dimensional thermal simulation as the boundary conditions, the three-dimensional temperature distributions were estimated. As the result, it was confirmed that the temperature distributions of an engine could be estimated with a high accuracy under running conditions of a complete vehicle. Moreover, it was verified that the temperature distributions can be estimated accurately for a variety of vehicles with engines of different specifications and mounting conditions.

1. Introduction

The growing demands for fuel economy in recent years call for development of light and compact engines having a high thermal efficiency. While the thermal efficiency of engines is improving due to various innovations, approximately 30% of the fuel's energy is released to the atmosphere, which is referred to as "cooling loss".

The heat in an engine is released to the atmosphere via the engine components, coolant and engine oil. When the cooling is inadequate, the engine temperature rises excessively. As a result, lowering of the volumetric efficiency, knocking and lowering of material strength may occur, which causes loss of engine performance and/or durability. For air-cooled engines in particular, lightness/

compactness and the retention of heat dissipation capacity are two contradicting demands. It is therefore necessary to pay attention to the balance between the heat entering the engine and that leaving the engine in order to design an engine while ensuring performance, durability and compactness. Furthermore, in motorcycles equipped with air-cooled engines, the cooling conditions vary significantly depending on the external shapes of the engines and the airflow around them. It is therefore necessary to analyze the engine temperatures of a complete motorcycle under running conditions.

In the meantime, the so-called "digital development", in which no prototype engine is produced in the early stages of development, is becoming the mainstream of recent engine development in an effort to reduce the development period.

* Motorcycle R&D Center

It is necessary to accurately estimate the temperature distribution in the engine in the early stages of development in order to accurately simulate the intake air temperature, which affects volumetric efficiency and knocking, as well as the durability of engine parts.

There are many examples of past research on engine cooling and temperature estimation⁽¹⁾⁻⁽⁵⁾. The conventional ways include the following.

- (1) A method that assesses heat dissipation from the engine outer surface to the atmosphere and from the water jacket to the coolant by the heat transfer coefficient using the 3 Dimensional Computational Fluid Dynamics (3D-CFD)
- (2) A method that estimates temperature distribution using a thermal conductivity simulation with measured temperatures and heat fluxes of a part of the engine as the boundary conditions
- (3) A method that estimates coolant temperature when operating in test mode using a 1D thermal fluid simulation

As such a method is unable to directly estimate the temperature or requires measured temperatures and/or heat flux data from various parts of the engine, it cannot be used to get an accurate temperature estimate in the design stage. Although the heat balance of coolant is taken into account, the estimate cannot be extended to the temperature distribution in the various parts of the engine.

Accordingly in this project we developed a method for estimating, in an early stage of design, engine temperature distribution in an air-cooled engine under the running conditions of a complete motorcycle. In the first place on this article, estimation methods are explained for the total heat balance and for the temperature distributions of an engine. Subsequently, the results of validations on the accuracies of the temperature estimations are described for the cylinder and the cylinder head whose temperatures are high in operation.

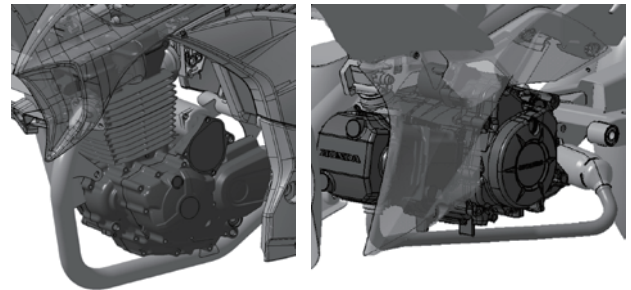
2. Test Motorcycles

The specifications of the two test engines used in this study are shown in Table 1, and the overview of the engine in Fig. 1. An air-cooled, 300 cm³ engine with its single cylinder canted forward 15 degrees was used for the development and validation of the temperature estimation method (Vehicle A in Table 1). The test motorcycle is equipped with an oil cooler. The air guide plates, or “shroud,” direct air flow to the spark plug for cooling.

A motorcycle equipped with an air-cooled, 110 cm³ engine with its single cylinder canted forward 80 degrees was used for the additional validation of the temperature estimation method in this project (Vehicle B in Table 1). This motorcycle is not equipped with an oil cooler. This motorcycle was selected as having significant differences in engine displacement, power output, engine configuration and airflow around the engine compared to Vehicle A.

Table 1 Engine specifications of test motorcycles

	Vehicle A	Vehicle B
Engine type	4 stroke engine	4 stroke engine
Number of cylinders	1	1
Cooling system	Air-cooled	Air-cooled
Bore [mm]	79	50
Stroke [mm]	59.5	55.6
Displacement [cm ³]	291.6	109.1
Compression ratio [-]	9	9
Maximum brake power [kW]	17.9 (7,500 r/min)	6.2 (7,500 r/min)
Maximum brake torque [Nm]	25.2 (6,000 r/min)	8.6 (5,500 r/min)
Maximum vehicle speed [km/h]	128	89



(a) Vehicle A

(b) Vehicle B

Fig. 1 Engine shapes of test motorcycles

3. Overview of Temperature Estimation Method

Figure 2 shows an overview of the temperature estimation method developed in this project. To calculate the heat balance in an air-cooled engine, it is necessary to take into account all the routes from the heat source up to the heat dissipation such as the heat input from the combustion to the engine, the heat transfer within the engine, the heat transfer from engine inner surface to the engine oil, the heat dissipation from the oil cooler to the atmosphere, and the heat dissipation from the cooling fins to the atmosphere. A 1D thermal simulation model is used for the comprehensive calculation with all the above factors coupled. The formulas coupled in the model are as per the following.

- The 1D engine cycle simulation is used to calculate the processes through which the combustion heat is transferred to the structural engine components via the combustion chamber walls.
- The three-dimensional shapes of the cylinder head and the cylinder are converted to the 1D thermal simulation model and incorporated into the simulation for calculation of heat transfer within the engine.

- The 1 Dimensional Computational Fluid Dynamics (1D-CFD) for oil flow rate is used to calculate the heat transfer from the internal walls of the engine to the engine oil. It should be noted that a combination of the 3D-CFD for oil behavior and the empirical equation of the heat transfer coefficient is applied for calculation of the heat transfer in the part where the engine oil flows along the wall such as in the camshaft chamber in the cylinderhead as well as for the heat transfer by the oil from the piston-cooling jet.
- Another 3D-CFD for airflow is used to calculate the heat transfer processes, in which the heat is dissipated to the atmosphere when the complete motorcycle is running.

The heat balance in the engine is determined by coupling the above formulas. Using the accordingly determined temperatures and heat transfer coefficients of the working gas and the engine oil as the boundary conditions, the temperature distribution in the various parts of engine is determined by the 3D thermal conductivity simulation.

4. Details of Temperature Estimation Method

4.1. Calculation of Heat Transfer to Engine

One of the key techniques in temperature estimation is the calculation of heat input from the combustion gas to the structural engine components. 1D engine cycle simulation is

a technique commonly applied to engine development⁽⁶⁾⁻⁽⁸⁾, and recently allows a very accurate calculation of power outputs and in-cylinder pressures.

Accordingly, detailed explanations should be sought from other information source. In an engine cycle simulation, the temperature of the wall that contacts the working fluid such as in the combustion chamber and port has to be set as a boundary condition. In the temperature estimation in this study, the wall temperature is obtainable as a calculation result rather than a boundary condition by incorporating the engine cycle simulation into the 1D thermal simulation that calculates heat balance in the engine.

4.2. Calculation of Heat Transfer in Engine Components

The heat in the engine components (e.g., cylinder head, cylinder) is transferred via three-dimensional routes. To convert such a route to the 1D thermal simulation model, we tried discretization to a number of zones as Fig. 3 shows. The thermal conductivity of each component, such as the cylinder head, cylinder, and valves, are also defined in the model. In our study, the specific heats of individual materials are not counted because the estimation is to be conducted for the temperature that has already been saturated state of the heat transfers with a constant vehicle speed condition.

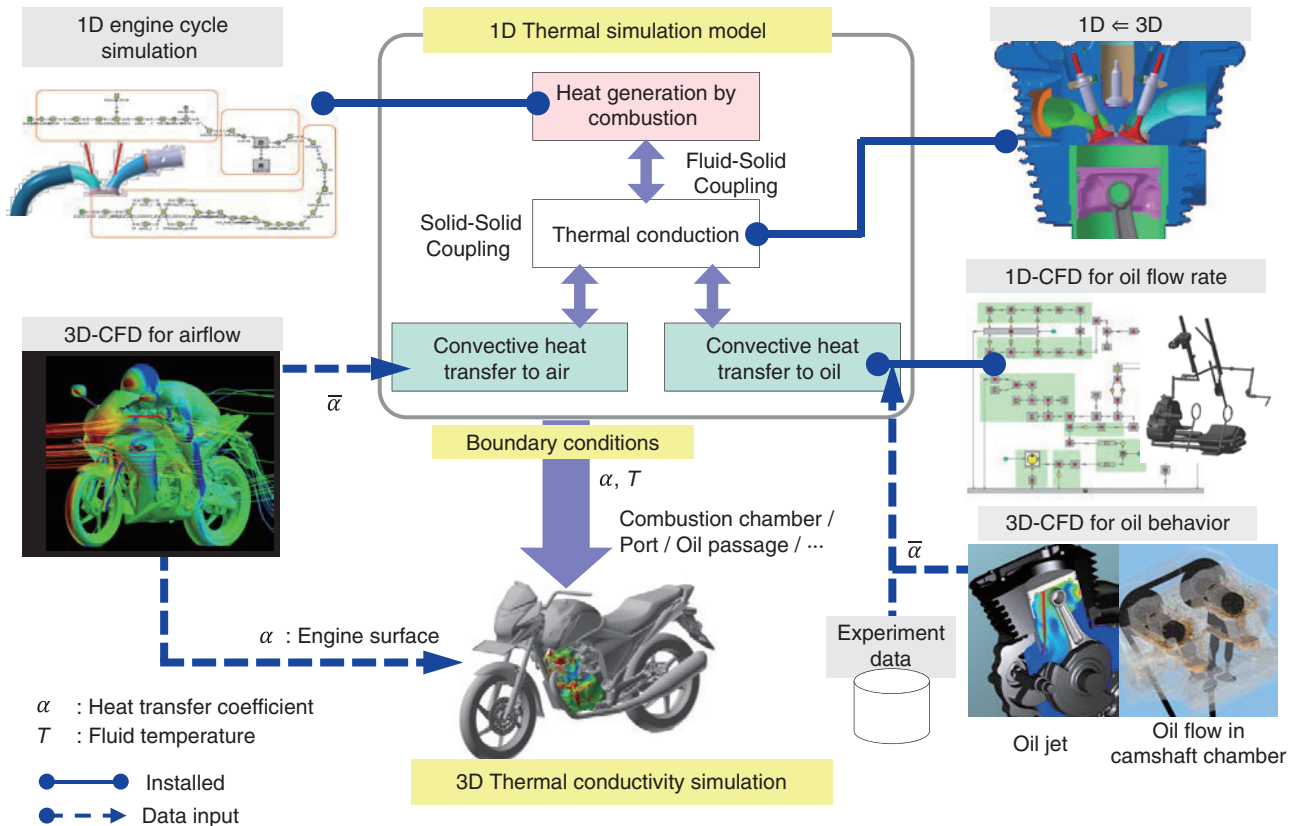


Fig. 2 Overview of temperature estimation method for air-cooled engine

Moreover, the heat transfer coefficients are defined on each boundary surface where the different materials are in contact. In places where a part such as the valve guide and valve seat is press-fit, an infinitely large heat transfer coefficient (i.e., a thermal contact resistance of 0) is set.

As an example other than in the press-fit part, explanations on the heat transfer coefficient between the cylinder and the sleeve are explained in the following section.

4.2.1. Heat transfer coefficient between cylinder body and sleeve

The predominant cylinder construction in small displacement, air-cooled engines is a cast-iron sleeve molded in the aluminum body. It is known that in this type of cylinder, a small clearance occurs between the cylinder body and the sleeve when the engine is operating under high temperature conditions due to the difference in the thermal linear expansion coefficients of aluminum and cast iron. The decrease of contact surface between the two parts due to the clearance interferes with the heat transfer from inside the cylinder to the cooling fins. To incorporate the said conditions into the 1D thermal simulation model, we created test equipment to measure the heat transfer coefficient between the cylinder body and the sleeve. As the test equipment in

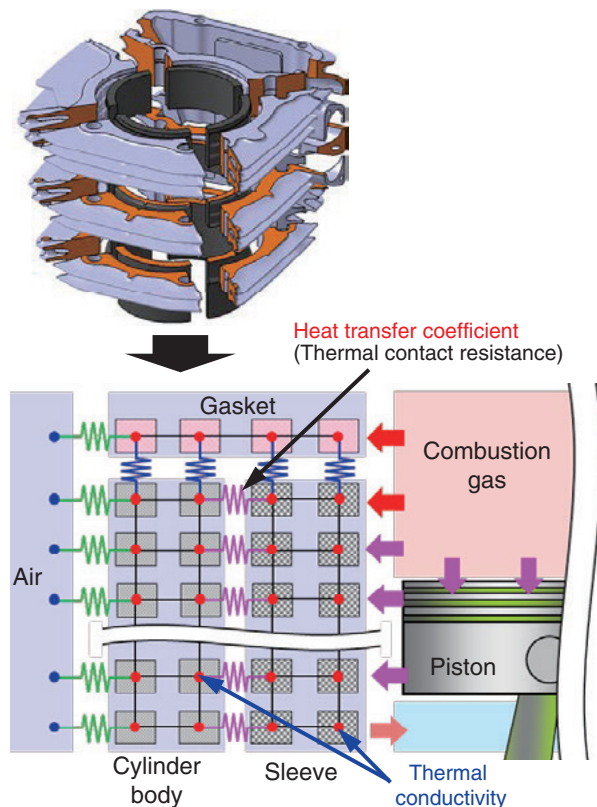


Fig. 3 Outline of heat transfer model in 1D thermal simulation

Figure 4 shows, a temperature gradient was created in the cylinder by heating the cylinder bore surface uniformly with high-temperature fluid (oil) and cooling the outside with low-temperature fluid (water). All the cooling fins were removed from the cylinder body for uniform cooling of the external wall of cylinder. In these tests, a uniform temperature in the axial direction of the cylinder was created by increasing the flow rates of the high-temperature and the low-temperature fluids as much as possible.

Under these conditions, the temperature gradient exists only in the radial direction of the cylinder as Eq. (1) shows.

$$\frac{\partial^2 T}{\partial r^2} + \frac{1}{r} \frac{\partial T}{\partial r} = 0 \quad (1)$$

Where T is the temperature in the cylinder, and r is the distance in the radial direction of cylinder. Under the above assumption, the temperature distribution in the radial direction of the cylinder can be represented by the Eq. (2).

$$T(r) = C_1 \ln r + C_2 \quad (2)$$

where C_1 and C_2 are constants.

Also, the heat flux, q , in the radial direction can be represented by Eq. (3).

$$q(r) = -k \cdot \frac{\partial T}{\partial r} = -\frac{C_1 \cdot k}{r} \quad (3)$$

Where k is the thermal conductivity of cylinder.

The heat transfer coefficient between the cylinder body and the sleeve can be determined by dividing the heat flux on the boundary surface by the temperature difference. However, as the cylinder body and the sleeve of an actual engine are thin, it is difficult to measure the temperature distribution. We therefore decided to calculate

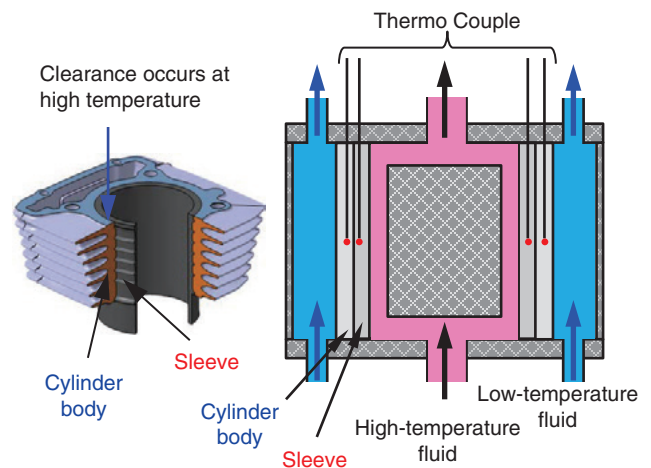


Fig. 4 Test equipment for measurement of heat transfer coefficient between cylinder body and sleeve

the temperature distribution in the cylinder body and sleeve using the heat transfer coefficient between the high-temperature fluid and the sleeve, and between the low-temperature fluid and the cylinder body. Accordingly, these heat transfer coefficients were acquired through a separate experiment using a dummy cylinder. Here, we presumed that the heat transfer coefficient between the fluid and the solid does not depend on the solid material.

Under the above-mentioned conditions, the heat transfer coefficient between the cylinder body and the sleeve is determined by the following steps.

- (1) As Fig. 5(a) shows, using the dummy cylinder made of a single material having a known thermal conductivity, the temperature distribution in the radial direction in the dummy cylinder, and the fluid temperatures on the heating side and the cooling side were measured.
- (2) Using the outcomes from Step 1, the temperature and the heat flux of the dummy cylinder were determined from Equations (2) and (3).
- (3) Using the outcomes from Step 2, the heat transfer coefficients α_{Hot} and α_{Cool} between the high-temperature fluid and the wall, and between the low-temperature

fluid and the wall, respectively, were calculated using Equations (4) and (5).

$$\alpha_{\text{Hot}} = \frac{q(r_{\text{Hot}})}{T_{\text{Oil}} - T(r_{\text{Hot}})} \quad (4)$$

$$\alpha_{\text{Cool}} = \frac{q(r_{\text{Cool}})}{T(r_{\text{Cool}}) - T_{\text{Water}}} \quad (5)$$

- (4) As Fig. 5(b) shows, using the actual cylinder body and the sleeve, the temperatures of the sleeve, cylinder body, the high-temperature fluid, and the low-temperature fluid were measured.
- (5) Using the temperature measured in Step 4 and the heat transfer coefficients determined in Step 3, the radial distributions of the temperature and the heat flux in the cylinder body and the sleeve were determined, and lastly the heat transfer coefficient between the cylinder body and the sleeve was calculated.

$$\alpha_1 = \frac{q}{T(r_{\text{Sleeve}}) - T(r_{\text{Cylinder}})} \quad (6)$$

Also, taking into account the deviation of heat transfer coefficient in the circumferential direction of the cylinder, each heat transfer coefficient was determined at some positions along the circumference of the cylinder, and the averaged value of these results was used as the representative value. Also, the heat transfer coefficient determined by this measurement method is a value for a certain surface area that is determined by the radius r without considering the surface area that depends on the shape of sleeve surface.

Figure 6 shows the outer shapes of the two types of sleeves commonly used in motorcycles. The heat transfer coefficient between the cylinder body and each type of sleeve was determined to cope with either type in the engine design stage. The determined heat transfer coefficients are shown in Fig. 7. It can be seen that the heat transfer coefficient of the Type A sleeve is three times larger than that of the Type B sleeve. It can be supposed that the Type A sleeve has a high heat transfer coefficient because it has many small protrusions on the contact surface. These heat transfer coefficients were applied to the 1D thermal simulation model.

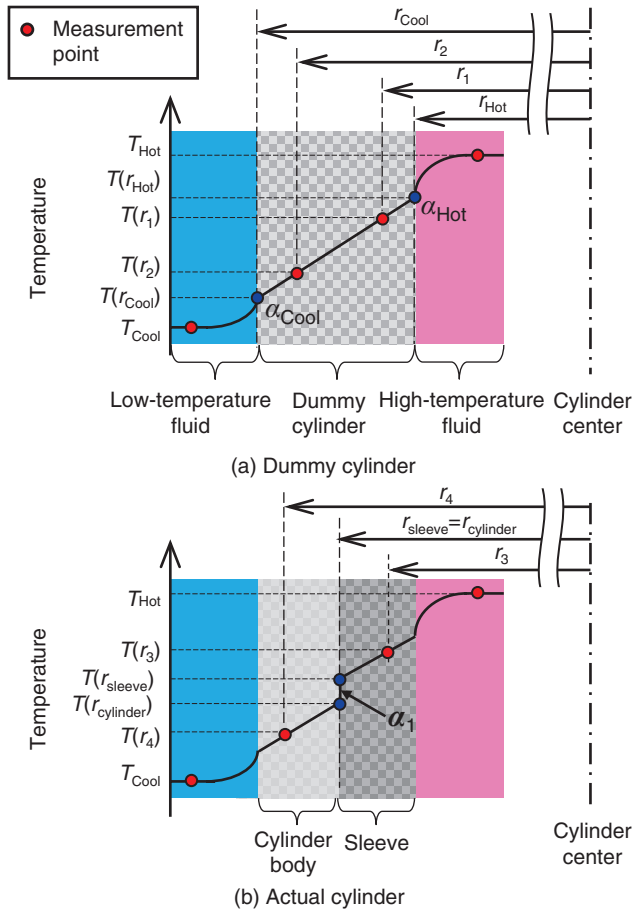


Fig. 5 Calculation technique of heat transfer coefficient between cylinder body and sleeve

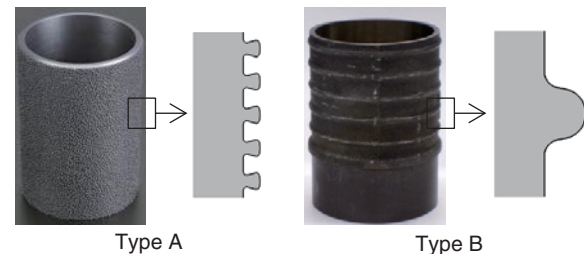


Fig. 6 Outer shapes of sleeves

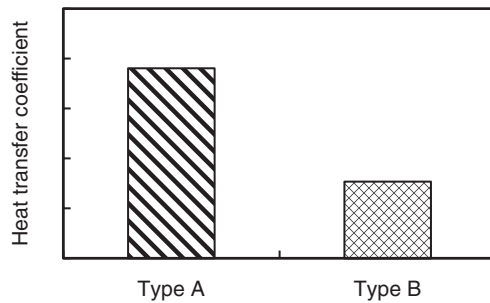


Fig. 7 Measurement results of heat transfer coefficient between cylinder body and sleeve

4.3. Calculation of Heat Transfer to Engine Oil

The heat transfer to the engine oil can be classified into the following three cases.

- (1) Heat transfer on the wall in a passage filled with oil such as the channel that feeds oil to various parts of engine
- (2) Heat transfer when the oil flows on the wall such as in the camshaft chamber, cam chain chamber
- (3) Heat transfer when the oil impinges on the wall such as the oil from the piston-cooling jet

Regarding Case (1), the heat transfer at the walls of the oil supply passages can be calculated without difficulty and this calculation can be incorporated relatively easily into the 1D thermal simulation model. For Case (2) and Case (3), however, as the flow of oil in the camshaft chamber and the behavior of oil from the piston-cooling jet impinging on the back surface of the piston crown are complex, it is necessary to make specific considerations to incorporate the heat transfer calculation into the 1D thermal simulation model. Accordingly, it was decided to determine the size of the area where the oil flows in the camshaft chamber, and the area on which the oil from the piston-cooling jet impinges using 3D CFD, and chose to calculate the heat transfer coefficient between the oil and the wall by basic experiments. Using both calculation results, the heat transfer to the oil was calculated in the 1D thermal simulation.

4.3.1. Calculation of oil flow rate in oil supply passages and calculation of heat transfer

For accurate calculation of cooling by oil, the flow rates of the oil supplied to various parts of the engine should be accurately known. In this calculation of oil flow rate, the 1D-CFD^{(8), (9)} in which the oil passage and the orifice are converted to a model by the diameter or the length of each was used. Also, regarding the oil pump, relief valve, oil cooler, etc., the P-Q characteristics of each were incorporated into the 1D-CFD model. As the 1D-CFD is also a technique commonly used in the product development, detailed explanations therefore are omitted here.

As shown in Fig. 2, the 1D-CFD calculation of the oil flow rate is incorporated into the 1D thermal simulation,

and is also used for the calculation of heat transfers in the engines occurring via the oil.

4.3.2. Calculation of heat transfer in camshaft chamber

In the camshaft chamber, the oil flowing out of the small holes in the rotating camshaft runs to the bottom surface of the chamber and returns to the crankcase. In this process, a heat transfer takes place between the oil and the cylinder head.

This oil behavior is analyzed by the particle method using Moving Particle Semi-implicit (MPS) method⁽¹⁰⁾, which is an algorithm for calculation of flow of incompressible fluid. The particle method is a numerical simulation method for a continuum using particles, and unlike the widely-applied finite difference method or finite volume method, it does not require meshing. Therefore, the particle method is a technique that has been recently applied to a flow field having a free surface such as the behavior of oil in an engine⁽¹¹⁾, the sloshing of fuel in a fuel tank, and so on.

Here, the processes through which the oil flowing from the camshaft returns to the crankcase via the cam chain chamber were calculated. Figure 8 shows the calculated behavior of the oil in the camshaft chamber. The calculation is for an engine speed of 8000 r/min, at which the camshaft rotates at 4000 r/min. The oil flow rate, which was calculated using the previously-mentioned 1D-CFD, was applied as a calculation condition. Also, for simplification, the motions of the valves and the cam chain were excluded from the calculation. Obtaining a time average of the calculation results, the zone where more than the prescribed numbers of particles exist is used as the area on which the oil flows.

In the analysis using the particle method, the heat transfer coefficient between the oil and the wall depends on factors such as the particle diameter and its influence radius in the calculation. It is therefore difficult to maintain a certain degree of calculation accuracy when applications to a large number of engines are considered. The heat

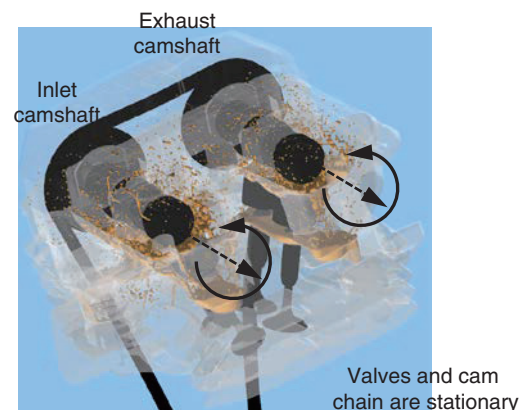


Fig. 8 Calculation result of oil flow in camshaft chamber

transfer coefficients between flowing oil and a flat plate were also acquired experimentally, and these values were used to calculate the heat transfer in the camshaft chamber. Figure 9 shows an overview of the test equipment. It allows a supply of oil on the evenly heated flat plate, in which the flow rate and the temperature are controlled. The flat plate is made of an aluminum alloy for which the thermal conductivity is already known. Also, to prevent heat dissipation from the heated flat plate to the ambient air, the outside of the flat plate is covered with an insulating material.

Using the test equipment, the temperatures of the oil and the inside of flat plate were measured, and the heat flux was calculated using the temperature gradient in the flat plate. By dividing the heat flux by the difference between the surface temperature of the flat plate and the oil temperature, the heat transfer coefficient between the flowing oil and the flat plate was determined. Note that the surface temperature of the flat plate was extrapolated from the temperature gradient in the flat plate.

$$\alpha_2 = \frac{q}{T_{\text{Oil}} - T_{\text{Wall}}} \quad (7)$$

$$q = -k \cdot \frac{\partial T}{\partial x} \quad (8)$$

where,

α_2 : Heat transfer coefficient between flowing oil and flat plate

T_{Oil} : Oil temperature,

T_{Wall} : Flat plate surface temperature,

q : Heat flux in flat plate,

k : Thermal conductivity of flat plate

The calculation of the heat transfer coefficient between the flowing oil and the flat plate was conducted using the oil flow rate, oil temperature, and the flat plate temperature as parameters.

Figure 10 shows the measured heat transfer coefficient between the flowing oil and the flat plate. As Fig. 10(a)

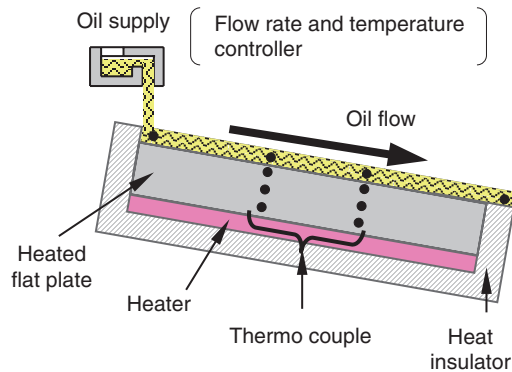


Fig. 9 Test equipment for measurement of heat transfer coefficient between flowing oil and flat plate

shows, the heat transfer coefficient is virtually constant irrespective of the oil temperature or the flat plate temperature within the range where the measurements were conducted. In the meantime, as Fig. 10(b) shows, such a tendency is seen that along with an increase of oil flow rate, i.e., an increase of oil flow speed, the heat transfer coefficient increases.

Accordingly, as seen in Eq. (9), the heat transfer coefficient between the flowing oil and the plate can be expressed as a function of the oil flow rate only, and is applied thus to the 1D thermal simulation model.

$$\alpha_2(Q) = A_1 \cdot Q^{A_2} \quad (9)$$

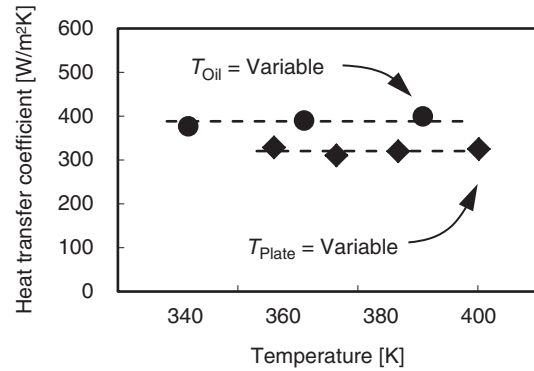
where,

Q : Oil flow rate

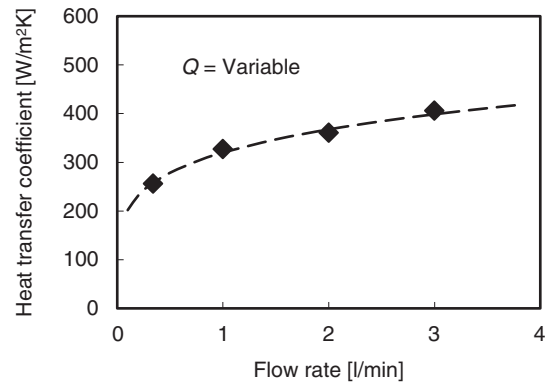
A_1, A_2 : Empirical constants

4.3.3. Calculation of heat transfer from back surface of piston crown

The region where the oil impinges and the heat transfer coefficient are required for calculation of heat transfer between the oil from the piston-cooling jet and the piston in the same manner as in the camshaft chamber.



(a) Influence of oil and flat plate temperature



(b) Influence of oil flow rate

Fig. 10 Measurement results of heat transfer coefficient between flowing oil and flat plate

Table 2 Calculation conditions of CFD for oil jet

Solution type	Unsteady state
Working fluid	Oil and Air
Compressibility	Incompressible
Pressure velocity coupling	SIMPLE (Semi Implicit Method for Pressure Linked Equation)
Mesh type	Hexahedron
Mesh size [mm]	0.15 - 2.4
Engine speed [r/min]	2,000
Flow rate of oil jet [l/min]	1.0

Firstly, how to calculate behavior of oil from the piston-cooling jet and where on the back surface of piston crown the oil impinges is explained hereafter. As the parts such as pistons and connecting rods are in motion during actual engine operation, the impingement position and the velocity of the oil jet change from moment to moment. Keeping that in mind, using the 3D-CFD taking into account the mutual influence of the airflow and the oil in the crankcase, we attempted to calculate the area where the oil impinges on the back surface of reciprocating piston. The 3D-CFD mentioned here means a gas-fluid multiphase flow calculation of the air and oil. The Euler-Euler approach was adopted and both the gas phase and the fluid phase were treated as continua^{(12), (13)}. In this calculation, the motions of the piston, the crankshaft and the connecting rod were also taken into account using the moving boundary. Figure 11 shows the outlines and mesh diagram of this calculation. Table 2 shows the calculation conditions. For the region in the proximity of the piston-cooling jet nozzle and in the direction of oil jet, the numerical diffusion of oil is restricted by laying fine meshes.

The oil impingement area was measured to verify the calculation results through recorded images of oil impingement taken with a high-speed camera through a transparent window made from acrylic resin located at the piston crown. The measurements were conducted at 2000 r/min with the engine driven by an electric motor.

Figure 12 shows a comparison of the measured and calculated oil impingement. The measured profile of the area where the oil impinges is shown by green lines, and the calculated profile of the area where the oil impinges is shown by red lines. In the calculation results, the zone on the piston surface where the oil volume ratio exceeds a certain level is deemed as the impingement zone, and the measured profile of the area where the oil impinges is also shown for comparison. The calculated result corresponded to the measurement result at each crank angle.

Accordingly, it was decided that the average value of

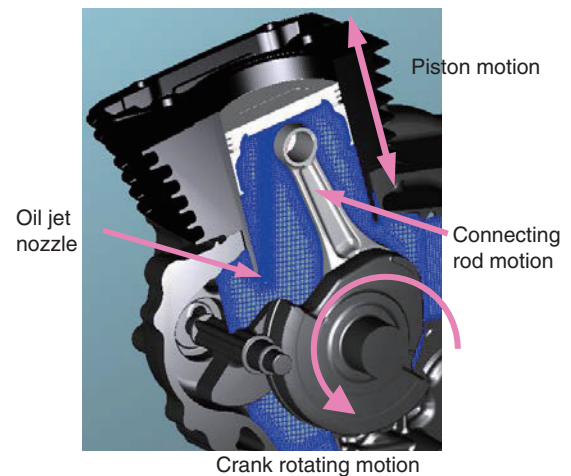


Fig. 11 Outline of CFD model for oil jet and mesh diagram

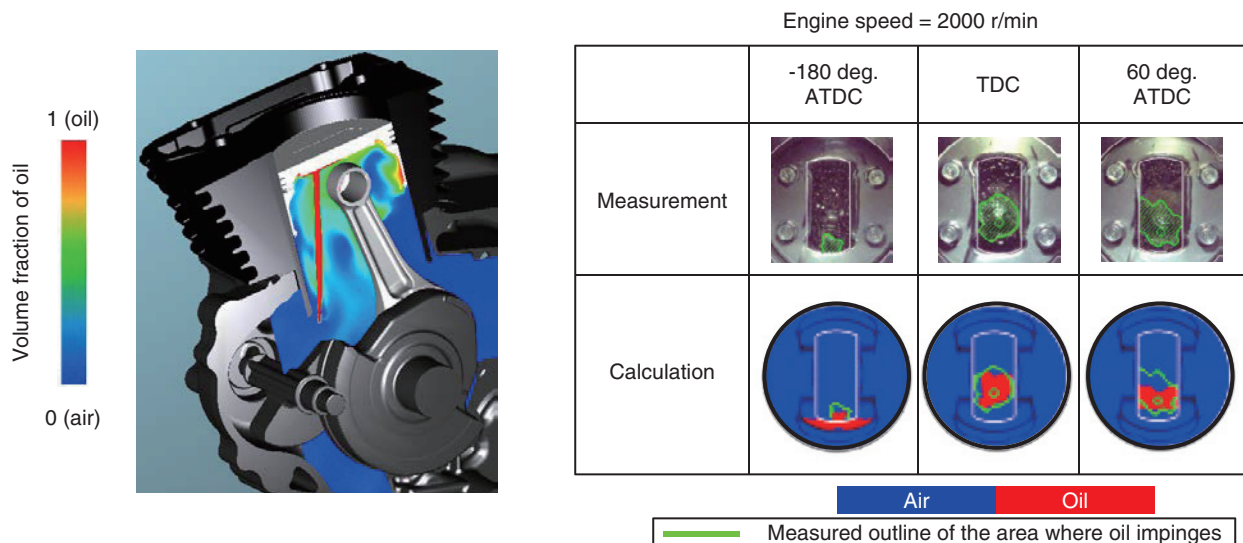


Fig. 12 Comparisons of oil impingement area on back of piston

those calculated results for a single engine cycle was used for the 1D thermal simulation.

As the heat transfer coefficient between the oil and the piston is mesh-dependent also in this CFD, it is decided to use the heat transfer coefficient calculated based on the parameters yielded from measurements.

To obtain an empirical equation for the heat transfer to the oil jet which cools the piston, the heat transfer coefficient was determined using the test equipment shown in Fig. 13. The basic principle of the test equipment is the same as that for the equipment used for determination of heat transfer coefficient of the oil flowing on the flat plate. The heat transfer coefficient was determined by having the oil jet impinge on the heated flat plate and measuring the temperature in the flat plate and the oil temperature at the outlet of nozzle. The heat transfer coefficient between the impinged oil and the flat plate was determined while changing the oil flow rate, piston-cooling jet nozzle diameter and the distance between the nozzle and the flat plate.

Figure 14 shows the measured heat transfer coefficient between the impinged oil and the flat plate. As Fig. 14(a) shows, while a slight variation is seen in the heat transfer coefficient depending on the piston-cooling jet nozzle diameter, the heat transfer coefficient is almost proportional to the flow rate. It is generally known that the heat transfer coefficient in the area where jet flow impinges depends significantly on the flow velocity, and has a radial distribution from the center of impingement⁽¹⁴⁾. However, in this project, we omitted the measurement of the radial distribution of the heat transfer coefficient as our purpose was to get a typical value representing the heat transfer coefficients in the area where oil is impinged. It is assumed, because of this measurement setting, that the heat transfer coefficient is influenced more by the oil flow rate than the nozzle diameter (i.e. flow velocity, effectively).

Figure 14(b) shows that the heat transfer coefficient increases as the distance between the nozzle and the flat plate is reduced. This tendency is maintained despite differences in oil flow rates. From these findings, the heat transfer coefficient between the impinged oil and the piston

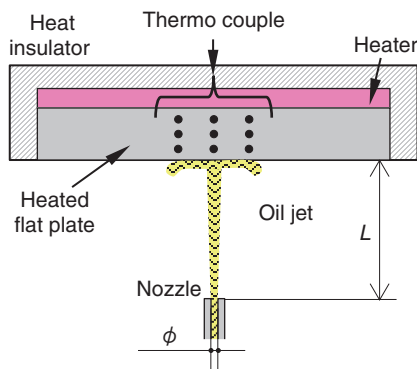


Fig. 13 Test equipment for measurement of heat transfer coefficient between impinged oil and flat plate

is expressed in Eq. (10) as a function of the oil flow rate and the distance from the nozzle to the piston.

$$\alpha_3(Q, L) = B_1 \cdot Q \cdot L + B_2 \cdot Q + B_3 \cdot L + B_4 \quad (10)$$

where,

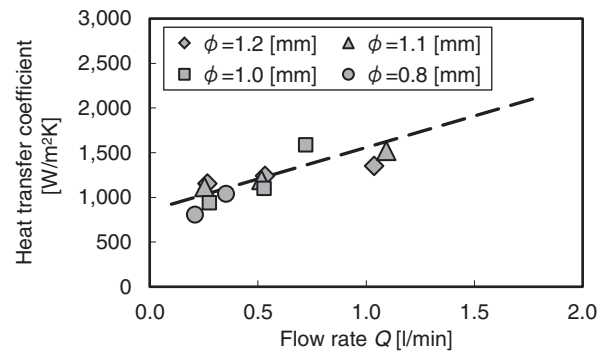
L : Distance between nozzle and piston

B_1 – B_4 : Empirical constants

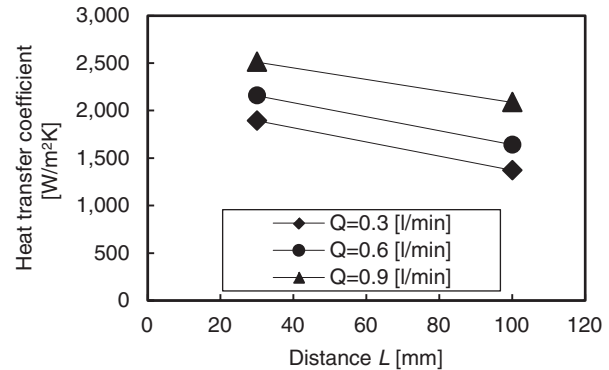
The oil impingement areas determined by this calculation and the average heat transfer coefficient on the back surface of piston crown in a cycle of the engine determined empirically from Eq. (10) were applied to the 1D thermal simulation model. Thus, the heat transfer processes in the cylinder head and the cylinder were taken into account in the temperature estimation method of this project.

4.4. Calculation of Heat Dissipation to the Atmosphere

As the flow of air around the engine in a running motorcycle must be taken into account in order to calculate the heat dissipation from the engine surfaces to the atmosphere, an airflow simulation was conducted using the 3D model of a complete motorcycle. On the basis of the findings from the 3D-CFD for the airflow, the engine surfaces were split into a number of zones, and the surface area and the average heat transfer coefficient in the zone



(a) Influence of oil flow rate and nozzle diameter



(b) Influence of distance from nozzle to flat plate

Fig. 14 Measurement results of heat transfer coefficient between impinged oil and flat plate

were calculated and used as the boundary conditions of the 1D thermal simulation model. The distribution of heat transfer coefficients was also used as a boundary condition of the detailed temperature distribution estimation.

As an example, a list of CFD calculation conditions in the vehicle A is indicated in Table 3, and the mesh diagram in the proximity of the engine is shown in Fig. 15. The mesh size near the engine surface is 1 mm, and the mesh size is set to a coarse size in accordance with the increase of the distance from the engine. The total number of meshes is approximately 25 million. Recently, unsteady calculation using an LES (Large Eddy Simulation) is widely applied to the estimation of aerodynamic effects and/or noises from turbulence⁽¹⁵⁾. However, it can be considered that the time scale of temperature changes on the engine surface is significantly larger than the time scale of the vortices occurring on the engine surface. Accordingly, we judged that the 3D-CFD for air flow under the steady conditions is adequate in this project.

4.5. Estimation of Detailed Temperature Distribution in Engine

For estimation of detailed temperature distribution in the engine, the heat balance in the engine shown in Fig. 16 is determined by the calculation in the 1D thermal simulation in which a combination of the technical options explained so far is applied. The diagram is the amount of heat transfer from various walls inside and outside of the engine, indicating one-cycle average of the engine. It indicates that the amount of heat transfer into the engine is very large from the combustion chamber [(1)-(3) in Fig. 16] and from the exhaust port, which is constantly exposed to hot gas. Conversely, the amount of heat dissipation from the engine is largest to the atmosphere [(9), (10)], followed by the heat transfer to the oil from the piston and camshaft chamber. It can also be understood from the aforementioned that in an air cooled engine, the heat transfer to the atmosphere and to the oil is very essential and cannot be ignored.

Next, using the findings from the 1D thermal simulation as the boundary condition, the 3D conductivity simulation

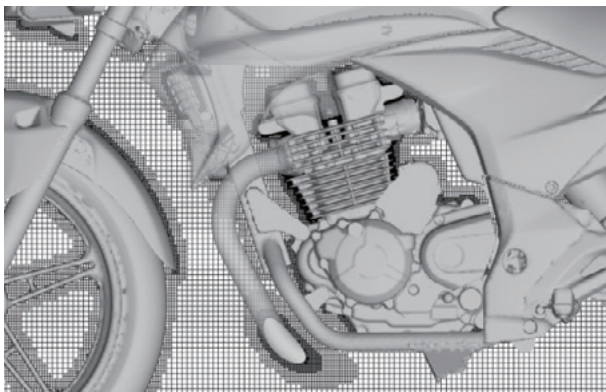


Fig. 15 CFD calculation mesh diagram for airflow around engine

Table 3 Calculation conditions of CFD for airflow

Solution type	Steady state
Working fluid	Air
Compressibility	Incompressible
Turbulence model	Realizable $k-\varepsilon$
Mesh types	Hexahedron Tetrahedron Pyramid
Mesh size around the engine [mm]	1 - 8
Total number of meshes	25,000,000
Vehicle speed [km/h]	128

was conducted in the actual engine configuration. The overview of the calculation model of vehicle A is shown in Fig. 17, and the calculation conditions in Table 4. Regarding the thermal conductivity of engine components and the heat transfer coefficient at an interface between varying materials, the same value as in the 1D thermal simulation model was used. The total number of meshes in the calculation is approximately 15 million, requiring approximately 40 boundary conditions for the entire walls. In this method, as a substitute for the omission of piston shape, the boundary condition taking into account the piston travel obtained from the 1D thermal simulation was incorporated in the cylinder bore wall. The temperature and the heat transfer coefficient applied as the boundary condition are shown in Fig. 18. As shown in the picture,

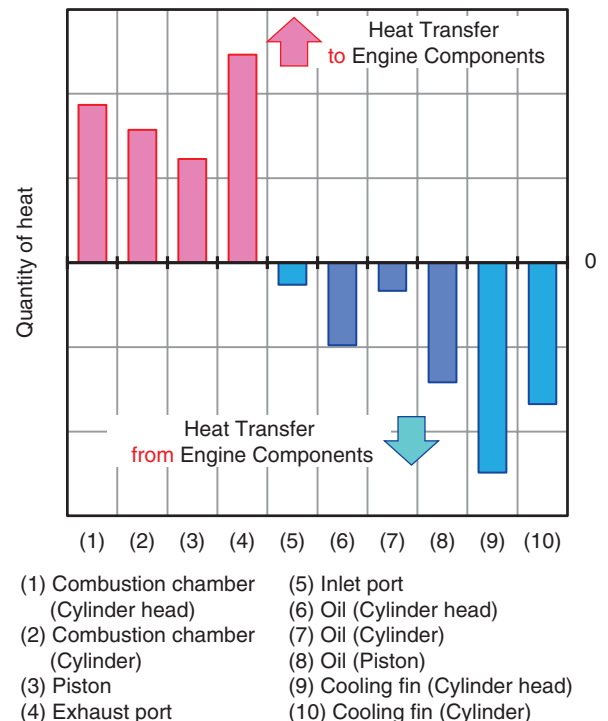


Fig. 16 Heat balance in engine of vehicle A

Table 4 Calculation conditions of 3D thermal conductivity simulation

Solution type		Steady state
Mesh type		Tetrahedron
Mesh size [mm]		0.7 – 1.4
Total number of meshes		15,000,000
Number of boundary conditions		40
Boundary conditions	Internal surface of engine	T and α (1D thermal simulation)
	Outer surface of engine	Atmosphere temperature, α distribution (3D-CFD for airflow)

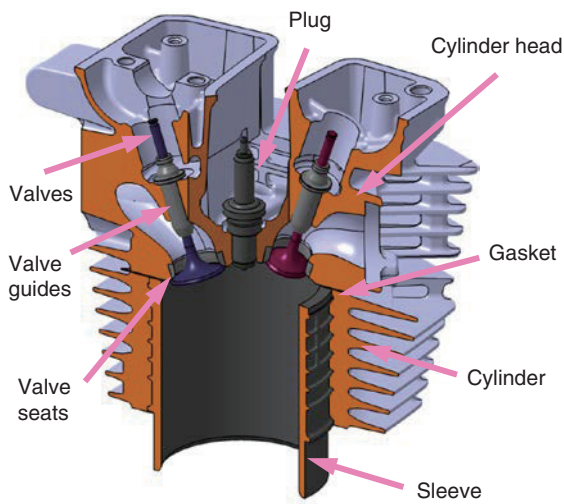


Fig. 17 Outline of 3D thermal conductivity simulation model

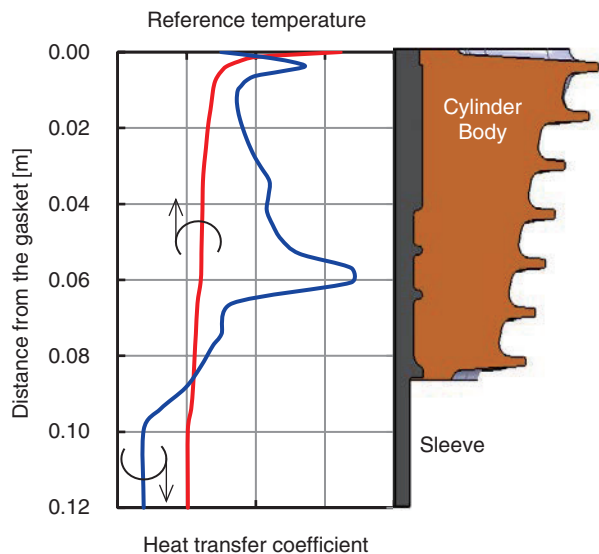


Fig. 18 Boundary conditions on cylinder wall

the reference temperature is high in the region close to the gasket surface that is significantly affected by the combustion gas. Also, the heat transfer coefficient is distributed in such a manner that it is high in the proximity of the TDC (Top Dead Center) and BDC (Bottom Dead Center) of piston.

Moreover, the distribution of heat transfer coefficient calculated by the 3D-CFD under running conditions is mapped on the engine surfaces.

5. Validation of Engine Temperature Estimation Accuracy in Running Motorcycle

5.1. Validation of Temperature Distribution Estimation Accuracy in Vehicle A

To validate developed method, we measured the temperatures of various parts of the engine mounted on vehicle A placed in a wind tunnel equipped with a dynamometer. The measurements were conducted at the maximum speed of the test motorcycle, 128 km/h, and an engine speed of 8150 r/min.

Figure 19 shows the estimated temperature distribution on the engine surfaces. Figure 23 shows a comparison of the temperature distribution in various parts of the cylinder head. The estimated temperatures and the measurements are in good agreement at the four spots in the proximity of the combustion chamber and on the spark plug seating surface. Good agreements between the estimates and the

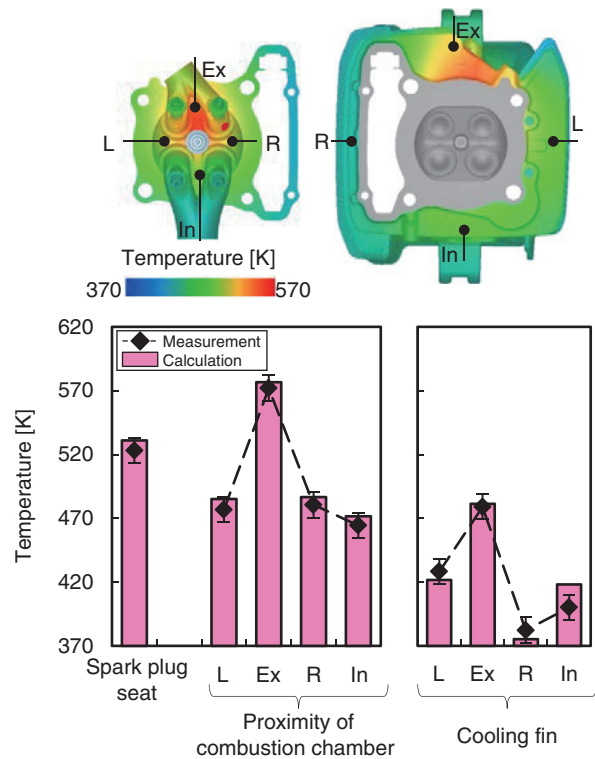


Fig. 19 Comparisons of temperature distribution in cylinder head

measurements are also seen in the cooling fins. In most points, the estimations are within 10 K of the measurements.

Figure 20 shows a comparison of the cylinder temperature distributions at a position 5.5 mm away from the cylinder head mating surface and 2 mm from the bore surface. The estimated and the measured temperature distributions around the cylinder are in very good agreement with a variance less than ± 10 K. The measured temperature is higher on the exhaust port side than on the intake port side. The temperature on the right side of engine is higher than on the left side. While the high temperature on the exhaust port side is due to the exhaust gas, the reason why the right side becomes higher in temperature can be considered due to poor heat dissipation to the atmosphere because the cam chain chamber is on the right side and that side of the cylinder is far from the cooling fins. The estimation of this trend is also in a good agreement with the measurement. The variance from the measurement is within ± 10 K at any measurement point.

Figure 21 shows a comparison of the temperature distribution in the direction of cylinder axis. The same trend is seen in the estimates and the measurements at any spot around the cylinder. In the conventional method of estimating the temperature distribution using just a 3D thermal conductivity simulation, the temperature gradient along the cylinder axis direction cannot be estimated accurately. In the newly developed method, however, the temperature distribution is accurately estimated by using the 1D thermal simulation model and taking into account the amount of heat transfer from the combustion gas to the cylinder wall in the direction of the cylinder axis.

5.2. Influence of Exterior Specifications

Tests were then conducted with and without a shroud for cooling the area near the spark plug. As Fig. 22(a) shows, the airflow around the spark plug and the heat transfer coefficient vary significantly with and without the shroud. Also, as Fig. 22(b) shows, it has been confirmed that the estimation can adequately represent the changes of temperature in the spark plug seating surface.

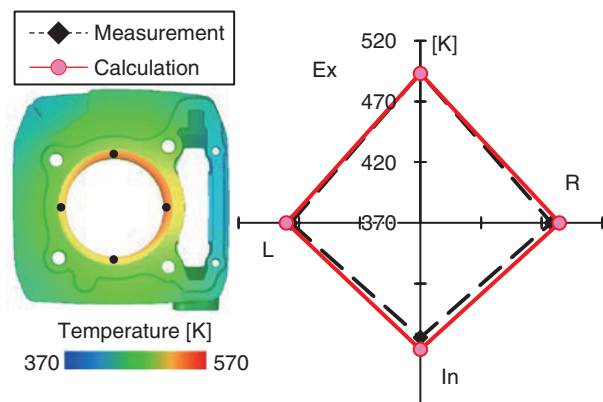


Fig. 20 Comparisons of temperature distribution in cylinder

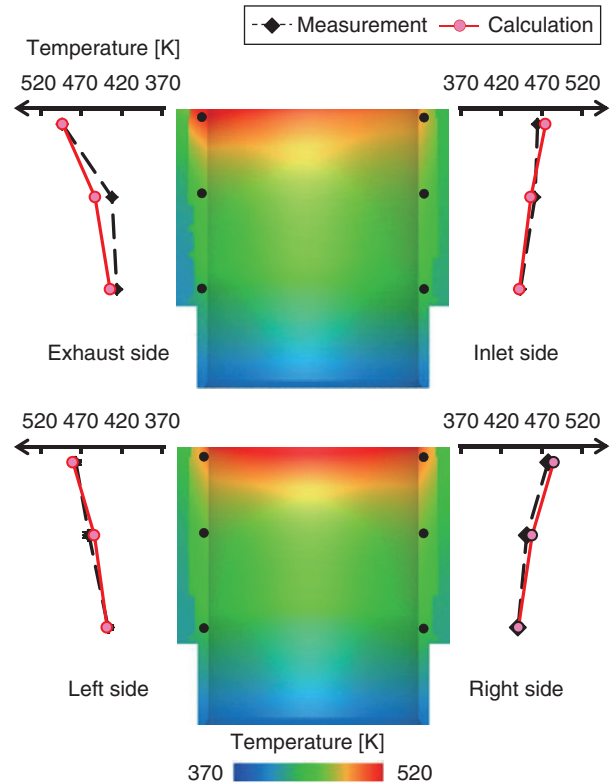
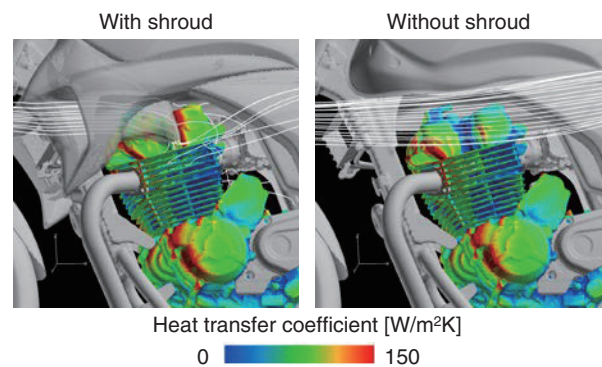


Fig. 21 Comparisons of temperature distribution in the direction of cylinder axis



(a) Stream line and heat transfer coefficient on engine surface

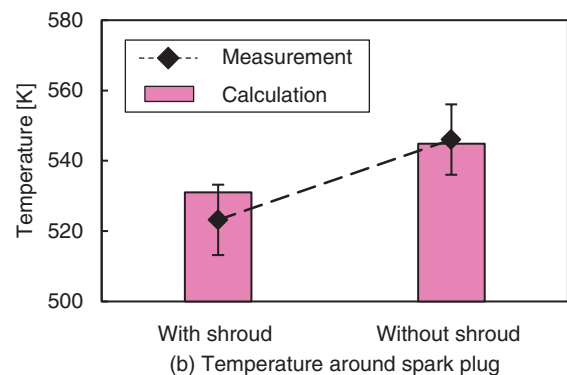


Fig. 22 Comparisons of heat transfer coefficient and temperature around spark plug

5.3. Validation of Temperature Estimation Accuracy in Vehicle B

This method was validated using a second motorcycle. The specifications of the engine in the test motorcycle are as per vehicle B shown in Table 1. The temperature estimation validation was run at the maximum speed of the test motorcycle, 89 km/h, and an engine speed of 8300 r/min. Figure 23 shows a comparison of engine temperature distributions from the motorcycle. Measurements were taken from the cylinder at a position 23.8 mm away from the cylinder head mating surface and 2 mm from the bore surface. The temperature distribution estimation by this method is in good agreement with the measurement both at the spark plug seat and near the cylinder. In the temperature distribution of this motorcycle, the differences between the estimates and the measurements are less than 10 K.

This proves the high accuracy of this temperature estimation method, since the two motorcycles have significantly different engine displacements, power output, engine configurations, airflow, and so on.

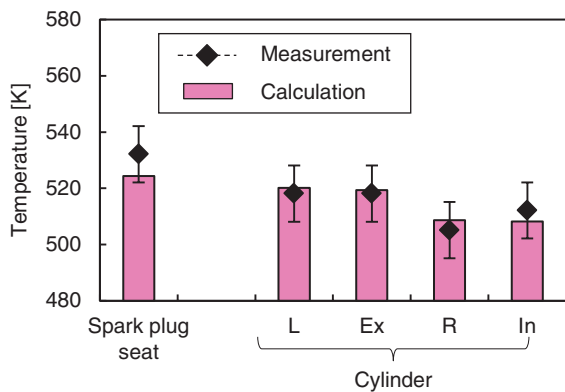


Fig. 23 Comparisons of engine temperature of vehicle B

6. Conclusions

We have developed a new method for estimating the temperature distribution in an air-cooled engine in a running motorcycle at an early stage of product development. The features of this method are outlined in the following.

- (1) To determine the heat balance of engine, we created a 1D thermal simulation model in which the heat input from the combustion, the heat transfer in the engine, and the heat dissipation to the atmosphere are coupled.
- (2) The heat transfer coefficient between the cylinder body and the sleeve, which has not been taking into account in the past, was empirically-determined and applied to the 1D thermal simulation model
- (3) A detailed process of heat transfer to the oil is applied to the 1D thermal simulation model by combining the empirically determined heat transfer coefficient between the oil and the wall with the 3D-CFD for oil behavior.

- (4) The proportion was clarified by using 1D thermal simulation between the transferred heat to the various portions of the engine and the heat dissipated to the oil and the atmosphere. Based on the result, the temperatures and the heat transfer coefficients of inside and outside of the engine were obtained. Eventually, those data were implemented to the detailed temperature distribution estimation.

Thus, our method makes it possible to accurately estimate the engine temperature distribution, with a deviation less than ± 10 K, in an engine on a running motorcycle without using data measured from the engine. We also confirmed a high correlation even when the flow of air to the engine changes significantly. Moreover, it has been proven that the method provides an accurate estimation of temperature even when the cooling air flow to the engine is significantly altered or in another motorcycle equipped with a different engine. The new method has been thus developed for estimation of temperature distribution in an air-cooled engine in the design stage.

References

- (1) Takahashi, Y., Gokan, Y.: CFD Analysis of Air Flow of Air-Cooled Motorcycle Engines, SAE Technical Paper, 2006-32-0005 (2006)
- (2) Baird, A., Thornhill, D., Cunningham, G., Troxler, P., Price, B.: CFD Simulations of Heat Transfer from Air Cooled Engines, SAE Technical Paper, 2006-32-0004 (2006)
- (3) Gokhale, A., Karthikeyan, N.: Optimization of Engine Cooling Through Conjugate Heat Transfer Simulation and Analysis of Fins, SAE Technical Paper, 2012-32-0054 (2012)
- (4) Mahmoud, K., Loibner, E., Krammer, J.: Integrated 1-D Tools for Modeling Vehicle Thermal Management, SAE Technical Paper, 2004-01-3406 (2004)
- (5) Moffat, J.: Coupling of 1-D and 3-D CFD Models to Predict Transient Hydraulics in an Engine Cooling Circuit, SAE Technical Paper, 2002-01-1285 (2002)
- (6) Horikawa, H., Kido, H., Iijima, S., Murakami, Y.: Prediction Technology of Engine Power and Intake/Exhaust Noise Using 1D-Simulation for Small-Displacement Motorcycles, SAE Int. J. Engines 3(2), p.428-437 (2010)
- (7) Onorati, A., Ferrari, G., Cerri, T., Cacciatore, D., Ceccarani, M.: 1D Thermo-Fluid Dynamic Simulation of a High Performance Lamborghini V12 S.I. Engine, SAE Technical Paper, 2005-01-0692 (2005)
- (8) Nomura, T., Matsushita, K., Fujii, Y., Fujiwara, H.: Development of Temperature Estimation Method of Whole Engine Considering Heat Balance under Vehicle Running Conditions, SAE Int. J. Engines 8(1), p.120-134 (2015)
- (9) Klingebiel, F., Kahlstorf, U.: Simulating Engine Lubrication Systems with 1-D Fluid Flow Models, SAE

- Technical Paper, 2000-01-0284 (2000)
- (10) Koshizuka, S., Oka, Y.: Moving-Particle Semi-implicit Method for Fragmentation of Incompressible Fluid, Nuclear. Science and Engineering, Vol. 123, p.421-434 (1996)
 - (11) Muto, K., Sakai, I., Ozaki, N.: Prediction of the Fluid Resistance in Stirred Fluid Using The Particle Method, Transactions of Society of Automotive Engineers of Japan, Vol. 41, No. 1, p.147-151, (2010) (in Japanese)
 - (12) Ishii, M.: Thermo-Fluid Dynamics Theory of Two-Phase Flow, Eyrolles, Paris (1975)
 - (13) Basara, B., Alajbegovic, A., Belder, D.: Simulation of Single- and Two-Phase Flows on Sliding Unstructured Meshes Using Finite Volume Method, International. Journal for Numerical Methods in Fluid, Volume 45, Issue 10, p.1137-1159 (2004)
 - (14) Martin, H.: Heat and Mass Transfer between Impinging Gas Jets and Solid Surface, Advances in Heat Transfer, Volume 13, p.1-60 (1977)
 - (15) Tsubokura, M., Nakashima, T., Nouzawa, T., Nakamura, T., Kitoh, K., Zhang, H., Hori, K., Oshima, N. and Kobayashi, T.: Numerical Prediction of Unsteady Aerodynamic Forces on Road Vehicles Using High Performance LES, FISITA World Automotive Congress (2008)

■ Author ■



Tomokazu NOMURA



Koichiro MATSUSHITA



Yoshihiko FUJII



Hirofumi FIJIWARA