

CFD Air Flow Analysis for Air-cooled Motorcycle Engines

Yasushi TAKAHASHI* Yoshitsugu GOKAN*

ABSTRACT

In order to develop an air-cooled engine, cooling air flow was analyzed by CFD (Computational Fluid Dynamics). Cooling performance of engine is necessary for precise temperature control. However, stable radiation is difficult for motorcycles because of the driving wind. Cooling fins are thin and complicated; therefore plenty of work time is necessary to create a boundary fitted mesh. In addition, since low Reynolds (Re) number area coexists with high Re number area in a calculation domain, applications of the wall function are limited. Therefore, we employed the Partial Cells in Cartesian coordinate method, which expresses complex shapes according to turbulence boundary conditions in the Cartesian coordinate. Analysis results showed good agreement with wind tunnel test results. Furthermore, useful knowledge in engine developments was obtained by examinations of local heat transfer coefficients calculated by Karman's analogy.

1. Introduction

Recently, even as motorcycle user requirements diversify, the demand for motorcycles using simple air-cooled engines remains strong.

The major issue in terms of cooling performance in these air-cooled engines is the fact that the heat radiation must depend on the airflow that passes over the engine when the vehicle is in motion. Ensuring sufficient overall heat radiation and achieving an appropriate balance between the cylinders in multi-cylinder engines are important factors in cooling performance. The intensive pursuit of cooling performance is simultaneously a pursuit of functional beauty. Previously, when developing air-cooled engines, it has been difficult to quantitatively analyze airflow behavior at the initial stages of the engine design process, and designers have relied on making the airflow visible using wind tunnels and taking temperature measurements. The importance of reducing emissions and enhancing fuel economy has recently increased in the field of motorcycles too, and precise engine temperature control has become an important issue. In addition, an efficient and adaptable development process that can respond to the requirement for large item small volume production is desirable from a business perspective.

The shapes of the cooling fins and other components of air-cooled engines are fine and highly complex, and a considerable

amount of operation time are required to generate normal boundary-fitted meshes such as tetra mesh. For this reason, there is no choice but to simplify detail forms. In addition, significant differences in velocity exist within the same calculation domain; for example, the front of the engine, which is directly struck by the air, is a high Reynolds (Re) number area, and the rear of the engine, where there is virtually no airflow, is a low Re number area. This necessitates a turbulence model which is applicable from laminar flow areas to high- Re areas. Therefore, the Partial Cells in Cartesian coordinate method^{(1),(2)} was used for CFD analysis method. The Partial Cells in Cartesian coordinate method is a computational fluid dynamics method using Cartesian coordinates that involves almost no expenditure of operation time on mesh generation for flow fields around complex forms. This method is provided with a turbulence model applicable to flows with a wide range of Reynolds numbers.

This research was intended to analyze the cooling air behavior around a motorcycle under actual driving conditions. Analyses were conducted using a motorcycle equipped with an in-line 4-cylinder 750 cm³ engine, set in a wind tunnel. The behavior of the airflow between the cooling fins was analyzed; in addition the distribution of heat transfer coefficients on the surface of the engine was calculated to quantitatively determine cooling performance. Furthermore, an optimization of cooling performance was examined based on obtained findings of the analyses.

* Motorcycle R&D Center

2. Air-cooled Engine Design and CFD Analysis Items

Figure 1 shows the relationship between air-cooled engine design and the subjects of CFD analysis.

To enable effective use of cooling air, visualization of the airflow, the shape of the cooling fins and the air channels, and the position of the oil cooler are included as CFD analysis items. Visualization of air flow in a wind tunnel shows only the behavior of the air. By contrast, this CFD analysis aims at the examination under actual driving conditions and the calculation of the heat transfer coefficient of each part of the engine for quantitative study of cooling performance. In addition, possibilities to connect to the calculation of the temperature distribution of the whole engine and the thermal stress analyses were considered.

When designing a new engine, CFD analysis will include the shape of the oil cooling jacket, layout of the oil pathway, optimization of amount of cooling oil, and the shape of the exhaust ports for minimizing heat source.

3. Calculation Method

3.1. Partial Cells in Cartesian Coordinate Method

In terms of design man-hours, the major issue in CFD analysis is the mesh generation, which in particular represents an issue when the analysis subject is a motorcycle. The Partial Cells in Cartesian coordinate method is one of Cartesian coordinate method that was developed to significantly reduce the operating time required for mesh generation when analyzing flow fields around complex forms. Figure 2 compares the concept of input data between the Partial Cells in Cartesian coordinate method⁽¹⁾ and the boundary fitted coordinate (BFC) method. Because the mesh in the BFC method is required to correspond to the shape data, a calculation mesh must be generated every time when the shape is changed. By contrast, the shape data and the mesh data are mutually independent in the Partial Cells in Cartesian coordinate method, and there is therefore no requirement to regenerate and modify meshes when the shape is changed.

When the mesh lines and the surface shape of analysis subject do not match, fluid cells lacking a part of hexahedron are produced in vicinity of the wall. These are called partial cells. By this definition, partial cells can exist in an enormous number of shapes, as shown in Fig. 3 (a). Discretization equations based on the finite volume method (FVM) were derived to allow calculating according to the shape in partial cells of any shape. In addition, because there are no fitted meshes to the boundary, an original turbulence model in vicinity to the wall⁽²⁾ that considered the turbulent flow behavior in low Reynolds number areas close to the wall has been used to calculate the turbulent flow, while the k - ε model and the wall function are fundamental. Cells divided by the wall, as shown in Fig. 3 (b), are not considered. The reason is based on the fact that the state inside the cell is expressed by one value. When this situation is predicted, the nearby meshes are more finely regenerated.

3.2. Calculation Procedure

Figure 4 shows the calculation procedure. The shape data is formulated with numerical wind tunnels by 3D-CAD software (for example CATIA) and written to a file as STL data. The STL data format is commonly used with optical molding devices; it presents shapes as forms resembling triangular aggregates, and is supported by almost all CAD systems. However, because the format was originally used in CAM fields, it lacks precision in the field of numerical simulations, and the content of the data will differ in

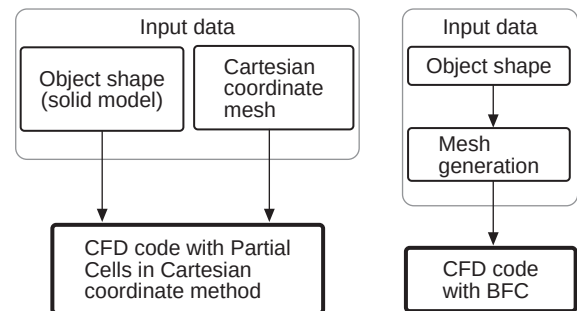


Fig. 2 Comparison of data flow between Partial Cells in Cartesian coordinate method and BFC

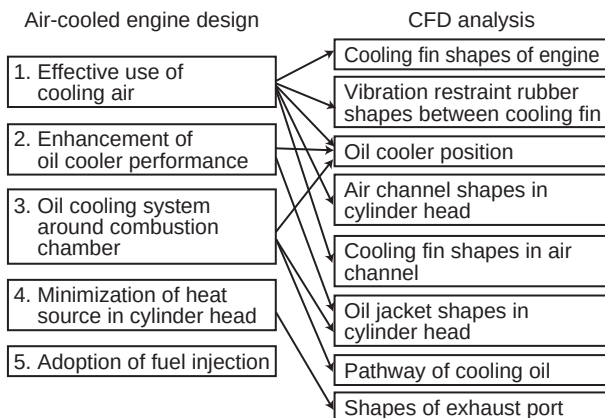


Fig. 1 Relation between air-cooled engine design and CFD analysis

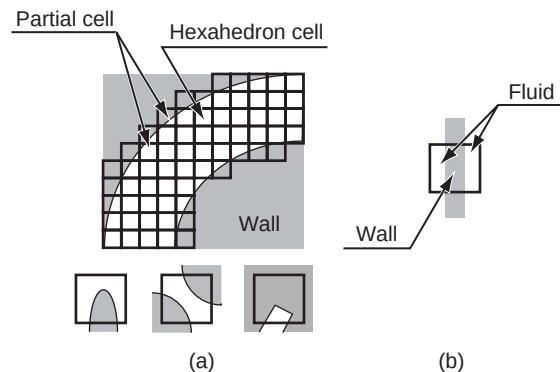


Fig. 3 Examples of partial cells (a), and partial cells divided by wall (b)

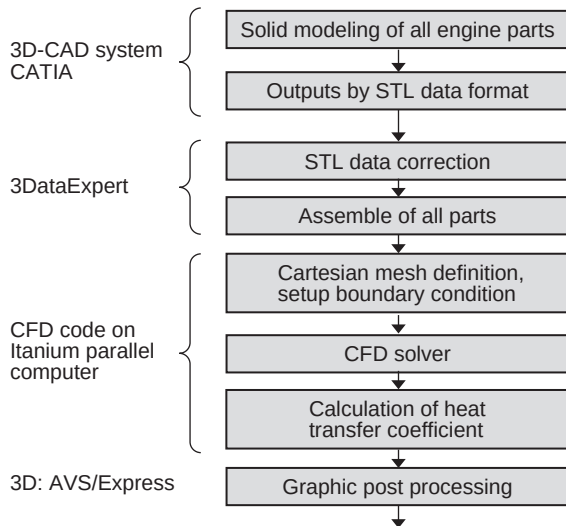


Fig. 4 Procedure of air cooling engine analysis by CFD code

each CAD system. For example, there may be gaps between certain triangles and contiguous triangles, and data correction is therefore required. In this project, a commercially available software program⁽³⁾ was used to correct the problems and also to assemble components. Following parts assembly, the Cartesian mesh was defined, the inlet boundary conditions were established, and calculations were started. When the calculations were completed, the results were used to calculate heat transfer coefficients, and the results of these calculations were represented graphically.

3.3. Formulation of Shape Data

Figure 5 (a) shows the solid model of the engine used in this analysis. Because it is not necessary to simplify forms to generate the mesh when using the Partial Cells in Cartesian coordinate method, the fillets and other shapes around the cooling fins can be created as correct as possible. In addition to engine shape data, frame shape data were created as required to enable analysis of behavior under actual driving conditions. The parts positioned upstream from the engine (the front tire, front forks, headlight, fuel tank, oil cooler, etc.) have a strong effect on airflow around the engine. Because the bottom of the tank has a particularly significant effect on the airflow in the space between the tank and the head cover, small parts and the wire harness were also created in as much detail as possible. The fins of the oil cooler core were simplified to reduce the amount of generated data, and were created with a distance of 1.5 mm separating the fins and a fin thickness of 0.25 mm. Downstream parts such as the rear tire and seat were not absolutely necessary for the purposes of this project, but were created to enable evaluation of aerodynamic performance. Because the Partial Cells in Cartesian coordinate method involves almost no expenditure of time on mesh generation, the formulation of this shape data represented the major commitment of operation time in the analysis.

Figure 5 (b) shows the completely assembled STL data. Fine facets are used for areas with complex shapes, such as the area around the cooling fins, while large, long and/or narrow facets are used for parts like the frame pipes. This has limited the number of facets to 1.8 million, enabling the data to be processed by a computer with 2 GB of memory.

3.4. Calculation Domain and Boundary Conditions

Figure 5 (c) shows the wind tunnel that forms the calculation domain and the boundary conditions. For analyses of aerodynamic performance, it is necessary to use a large wind tunnel in order to eliminate the effect of the side walls. However, because the intent of the present study was only to analyze the airflow around the engine, a small wind tunnel (5.0 m × 2.0 m × 2.0 m) was used to reduce calculation time.

Boundary conditions were set as an airflow velocity corresponding to vehicle speed. A 5% turbulence intensity was used and the length scale was set at 0.05 m, at the entrance of the wind tunnel, and fixed walls were used for the floor and side walls. Calculations were conducted for a maximum vehicle speed of 210 km/h and 60 km/h to correspond to wind tunnel test conditions.

3.5. Calculation Grid

From considerations of installed memory and calculation time, the number of grids in the entire calculation domain was set at 133 × 128 × 139, to total approximately 2.37 million grids. Because a calculation grid size of approximately 2.3 mm was used close to the fins, while a fin pitch of approximately 10 mm and a fin thickness of 1–4 mm were set, two or three grids were used between fins. The thickness of the tubes in the oil cooler was approximately 2 mm with an approximate 10 mm distance between them. Two to three grids were used between the tubes, as in the case of the cooling fins. The type of divided cells shown in Fig. 3 (b) occurred in the core section, but this was allowable because the quantities of the state of multiple flow domains in the same cell can be regarded as identical. Figure 5 (d) shows the grid for the area around the oil cooler and engine fins. A maximum distance between grids was set 40 mm close at the front tire, 85 mm close at the rear tire, and 250 mm close at the entrance and exit of the wind tunnel.

The calculation time was approximately 7 hours using a 16 CPU parallel computer. While the calculation time for the method used here was almost the same as that for tetra mesh CFD software when the total number of meshes was the same, given the fact that complex forms could be calculated with a comparatively lower number of meshes, the actual calculation speed was several times faster. This method makes it possible to complete calculations within one night, and has therefore become an effective design support tool.

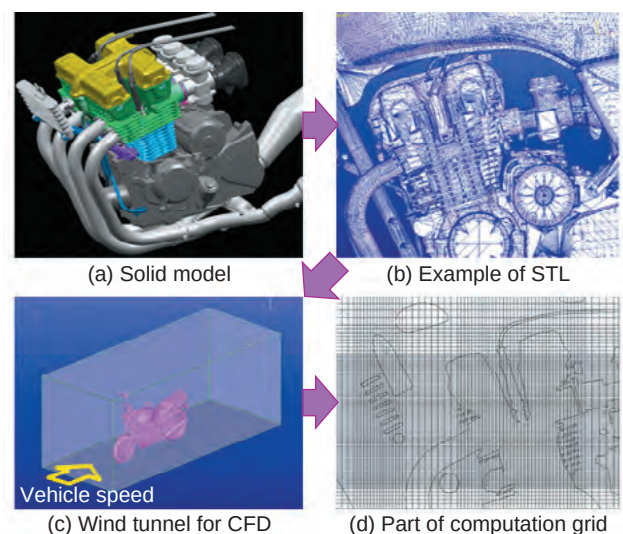


Fig. 5 Calculation flow chart

3.6. Calculation of Heat Transfer Coefficients Using Karman's Analogy

The heat transfer coefficient, α , is defined by the following Newtonian equation:

$$q = \alpha (T_w - T_f) \quad (1)$$

Here, q is the amount of heat transfer per unit area, T_w is the temperature of the wall, and T_f is the temperature of the fluid. The latter parameter is established at a sufficient distance from the wall.

The heat transfer coefficient is a function of an enormous kind of parameters, including all the effects on heat transfer, such as the shape of the solid walls, the type of fluid, and the state and velocity of the flow. It is therefore challenging to determine heat transfer coefficients for complex forms experimentally, and the use of CFD analysis is considered to have particular value in this area. In addition, turbulence has a significant effect on the state of the flow.

Normally, heat transfer coefficients are determined by solving energy equations for the flow field. This makes boundary conditions for heat at the walls necessary. If changes in the temperature of the walls are also considered, thermal conductivity calculations are required for the solids in the simulation model. Because the heat transfer coefficients are determined by the fluid properties and flow field state, when changes in the temperature of the fluid cause changes in these fluid properties and/or flow field state, coupling calculation between fluid and solid able to be solved for the mutual dependence of flow and temperature become necessary, and this requires enormous calculation resources. However, in most cases the fluid temperature changes that result from the cooling airflows in air-cooled engines can be ignored. To increase calculation efficiency, Eq. (2), which uses Karman's analogy⁽⁴⁾ (derived from the relationship between fluid friction and heat transport), is used to obtain the heat transfer coefficient.

$$\frac{1}{\alpha} = \frac{1}{\rho C_p} \sqrt{\frac{\rho}{\tau_w}} \left[\frac{U_s - U_2}{\sqrt{\tau_w/\rho}} + 5P_r + 5\ln(5P_r + 1) \right] \quad (2)$$

where α is the heat transfer coefficient, ρ is density, C_p is isopiestic specific heat, τ_w is the wall shear stress, P_r is the Prandtl number, U_s is the mean velocity at the outer edge of the boundary layer, and U_2 is the mean velocity in the boundary layer between the logarithmic and transition areas.

4. Calculation Results and Discussions

4.1. Rendering Flow Visible

Figure 6 shows the pressure distribution on the vehicle surface and flow lines around the vehicle, providing an overall image of the calculation results. No partial cells were used to display the stream lines because a commercially available software program was used; post-processing using a normal Cartesian coordinate mesh was used. Because of the extremely rough nature of mesh that was used, other than in the area around the oil cooler and the engine, accuracy in calculating flow inclusions to the tires and the back wake was not a consideration. The space between the front fender and tire was blocked in to prevent the generation of divided cells of the type shown in Fig. 3 (b) in the calculations using rough meshes.

Figure 7 (a) shows the velocity magnitude in a cross-section of the center of the vehicle. Most of the airflow over the front fender

strikes the engine block, passes under the oil pan and flows to the rear. Figure 7 (b) shows the flow velocity vector at an offset of 57 mm to the left of the center of the vehicle. While airflow towards the oil cooler is visible, the oil cooler is mounted in relation to the frame, positioning it diagonally to the flow, and there is therefore room for enhancement from the perspective of cooling performance. The airflow passed through the oil cooler is broad in three dimensions, but some flows go over the handles and the tank to the upward. There is little flow over the head cover, and as per predictions no strong flow can be observed at the rear of the engine. Figure 7 (c) shows the turbulence intensity at an offset of 57 mm to the left of the center of the vehicle. As this figure indicates, turbulence is very strong behind the front tire. Because the tire and the floor are both fixed in this calculation, turbulence can be predicted to be even stronger when the vehicle is actually running.

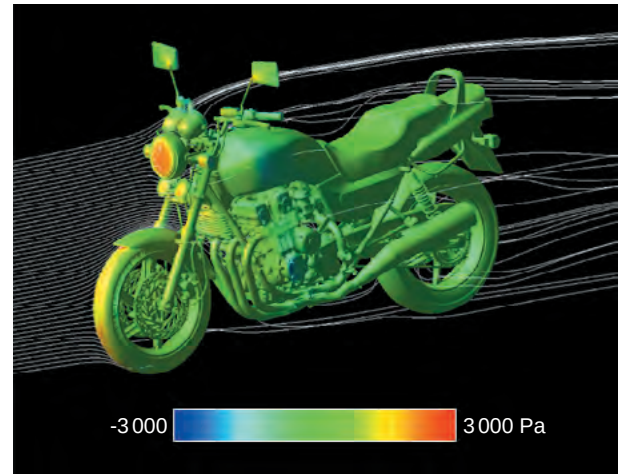


Fig. 6 Stream line around motorcycle

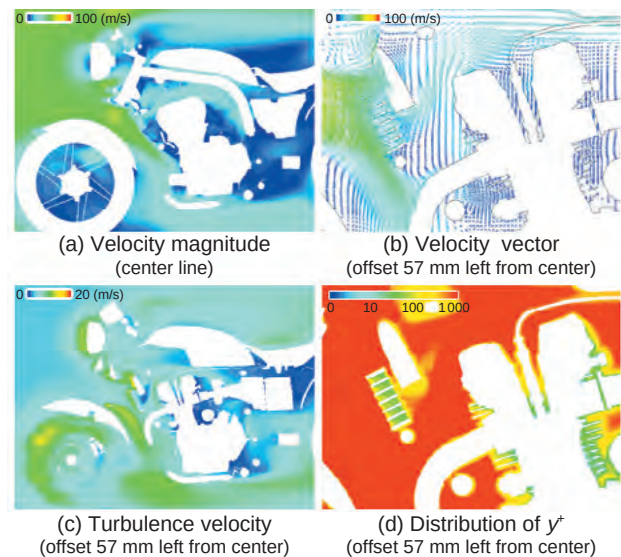


Fig. 7 Section-cut post processing

Figure 7 (d) shows the distribution of the dimensionless distance from wall y^+ , as determined using Eq. (3), to verify the accuracy of the calculating of the turbulent flow close to the walls.

$$y^+ = \frac{\rho l C_\mu^{1/4} k^{1/2}}{\mu} \quad (3)$$

where ρ is density, l is distance from the wall, C_μ is the turbulence model constant (0.09), k is kinetic turbulence energy and μ is the viscosity.

Figure 8 shows the results from the previous figure represented in three dimensions. The two-dimensional representation in Fig. 7 shows the calculation results for the main spaces, but the three-dimensional representation in Fig. 8 enables the calculation results for the areas close to the wall to be read. Figure 8 (a) shows the pressure distribution on the surface of the vehicle. These pressure results are used to determine the hexagonal component force when evaluating aerodynamic performance. Figure 8 (b) shows the flow velocity near the wall, and Fig. 8 (c) shows the turbulence intensity near the wall. Strictly speaking, because the flow velocity on the physical wall is zero, this represents the flow velocity and the turbulence intensity in the partial cells contiguous to the wall. Because the size in each partial cell is different, the distance in millimeters from the wall of the represented values differs according to each partial cell size. For this reason, caution must be exercised

when treating these results as flow analysis results. In addition, cooling air analysis must provide a quantitative estimate of the ultimate magnitude of heat radiation and the temperature that will be reached by each part of the engine. Equation (2) is therefore used to determine heat transfer coefficients for the engine surface from flow velocity and turbulence energy. The results are shown in Fig. 8 (d). The red sections in the figure (the cylinder head, the side of the head cover on the exhaust side, the front of the crank case cover, etc.) represent areas at which the heat transfer coefficient is close to 360 W/m²K and the heat transfer coefficient is high, i.e., the areas displaying excellent heat radiation.

4.2. Airflow around the Cylinder Fins

Post-processing using the particle trace method was conducted for the area around the cylinder fins to visualize the airflow. The results are shown in Fig. 9. The particle trace method places particles in the flow and traces their locus. This method corresponds to using smoke in wind tunnel tests to make the streak visible, and results for the two methods correlate for steady-state flows. The flow velocity at the entrance of the wind tunnel was set at 60 km/h to correspond to a wind tunnel test. Angular fins were used in this engine, and the airflow striking the edge of the cylinder block fin was observed to deflect from the edge without having flown along the fin or to the rear of the engine. At middle of cylinder block, the

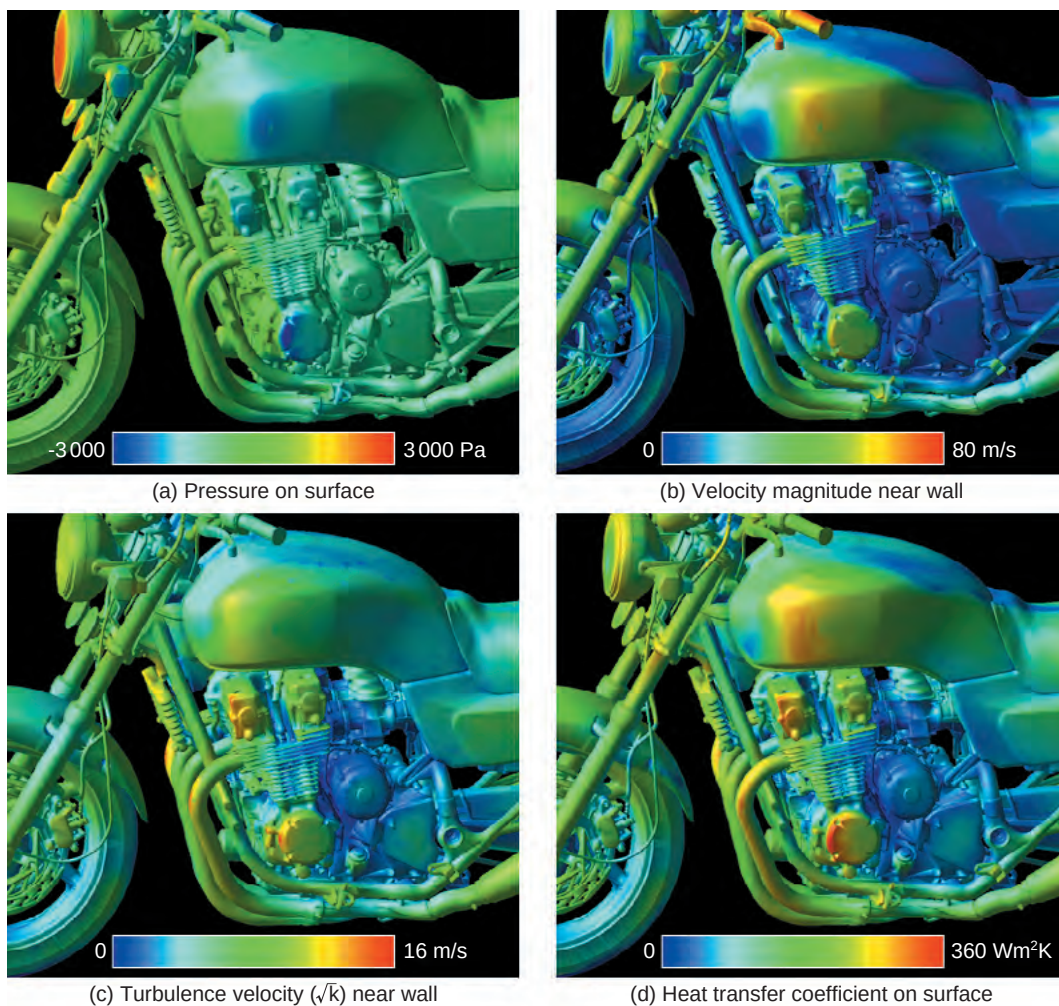


Fig. 8 Post treatment derived from CFD results

place is below ignition plug, Fig. 10 shows velocity vector between cylinder fins by close-up indication. The reverse direction of flow was observed in a backward domain from the cylinder center. The heat transfer coefficients shown in Fig. 8 (d) also indicate that heat radiation from the rear half of the cylinder fins is low. There is therefore a margin for enhancement from the perspective of cooling performance.

4.3. Comparison with Wind Tunnel Tests

Wind tunnel tests were conducted to verify the CFD results. Figure 11 shows the result of these tests. The installation conditions of the wind tunnel resulted in the tests being conducted at a flow velocity of 60 km/h. As Fig. 11 shows, the smoke used in the tests was deflected from the edges of the cylinder block fins and flowed away from the vehicle rather than flowing between the fins. This result corresponded with the results of visualization using CFD, and then the accuracy of CFD was confirmed by actual vehicle tests.

4.4. Study of Optimum Fin Shape

A variety of studies of the fin shapes were conducted. The results indicated that the change from angled fins to the rounded fins shown in Fig. 12 would enable the achievement of the flow characteristics close along the fins shown in Fig. 13. These studies therefore enabled the development of a fin shape displaying enhanced cooling performance.

4.5. Airflow above the Spark Plugs

Precise heat control in the middle cylinders is an issue for in-line 4-cylinder engines. In particular, the temperature of the spark plug seats, which are mounted above the combustion chamber of

the middle cylinder, reaches temperatures that are tens of degrees higher than those at the end cylinders. The airflow around the spark plugs was therefore analyzed. Figure 14 shows the visualization results of the airflow around the spark plugs using the particle trace method. The airflow over the head cover is drawn laterally from the side of the spark plugs by the wind when the vehicle is in operation, but the flow from the head cover to the area around the spark plugs is limited, especially, with virtually no airflow to the area around the spark plugs at the middle cylinders. As a result, and also indicated by the heat transfer coefficients shown in Fig. 8 (d), the heat transfer coefficient at the head surface around



Fig. 11 Flow visualization by wind tunnel with smoke

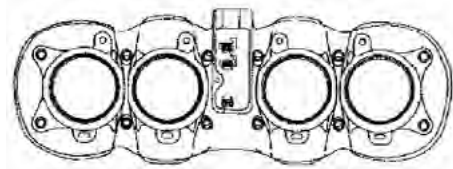


Fig. 12 Overhead view of cylinder block of prototype engine

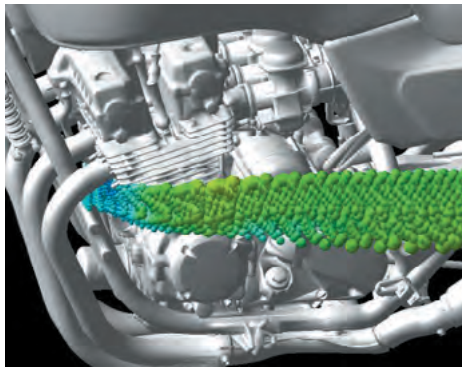


Fig. 9 CFD results by particle trace method

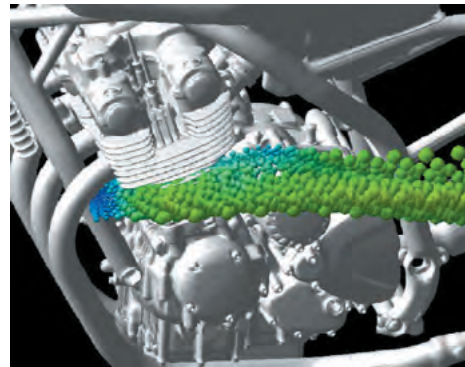


Fig. 13 Flow in vicinity of rounded-fins cylinder block

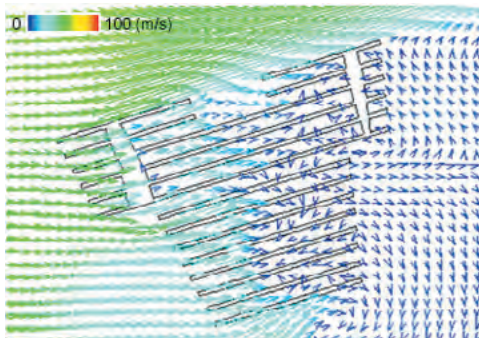


Fig. 10 Velocity vector around fins

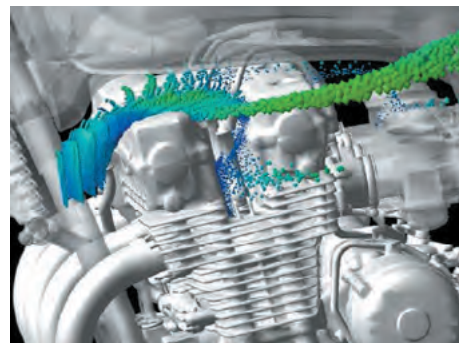


Fig. 14 Flow analysis around ignition plug

spark plugs of the middle cylinder is low. It is necessary to establish an air inlet enhancing the cooling performance of the area around the middle cylinder spark plugs, and this will require a design that is coordinated with the configuration of the interior of the engine.

4.6. Heat Transfer Coefficients in Each Section and Future Prospects

Figure 15 (a) shows the heat transfer coefficients over the entire motorcycle. Figure 15 (b) shows a view of the front of the engine, and Fig. 15 (c) shows a view of the front of the engine from below and diagonally. For greater clarity, the frame parts (tires, exhaust pipe, etc.) have been removed from Figs. 15 (b) and (c), and only the engine is shown. As the figures show, while there is good heat radiation from the sides of the engine, the heat transfer coefficients of the central section of the engine, and particularly the area around the cylinders, are unexpectedly low. By contrast, there are numerous areas displaying high heat transfer coefficients at the bottom of the engine (the oil pan, etc.), indicating the significance of the oil pan's effect on engine cooling. In addition, the heat transfer coefficient of the area around the oil filter that is attached to the bottom of the front of the engine is quite high, demonstrating that the filter also contributes to cool the engine.

Figure 15 (d) shows the frame parts and the engine viewed from the bottom, and gives a good idea of heat transfer in parts other than those on the engine. For example, the figure indicates that

cooling air does not necessarily flow uniformly over the entire core area of the oil cooler. Results for heat radiation from the exhaust pipe will be useful in predicting the exhaust gas temperature in the pipe when the vehicle is running. The figure also indicates that it is possible to study brake disk cooling using CFD.

These analyses of heat transfer coefficients can connect to thermal analysis of the entire engine. Conventionally, thermal analyses link fluids and solids and solve energy equations. However, in cases in which different coolants coexist, such as air-cooled engines that also use oil cooling, the amount of calculation required makes it difficult to link all the elements and solve the energy equations. The method using Karman's analogy discussed here, by contrast, is simple and can be used for analyses during the design process, making it an effective method of examining design proposals. In addition, the method also enables consideration of heat transfer on the inner walls of the intake and exhaust ports, enabling analysis under conditions that resemble those encountered in actual vehicle operation. Moreover, if heat transfer between the combustion gas and the cylinder heads is solved by coupling those in combustion simulations, present-day computer capabilities make it difficult to solve cooling air by additionally coupling it in the simulation. Using Karman's analogy for heat transfer between the cylinder head and the coolant can be considered an effective method of calculating the relationship between combustion and cooling when conducting combustion analyses.

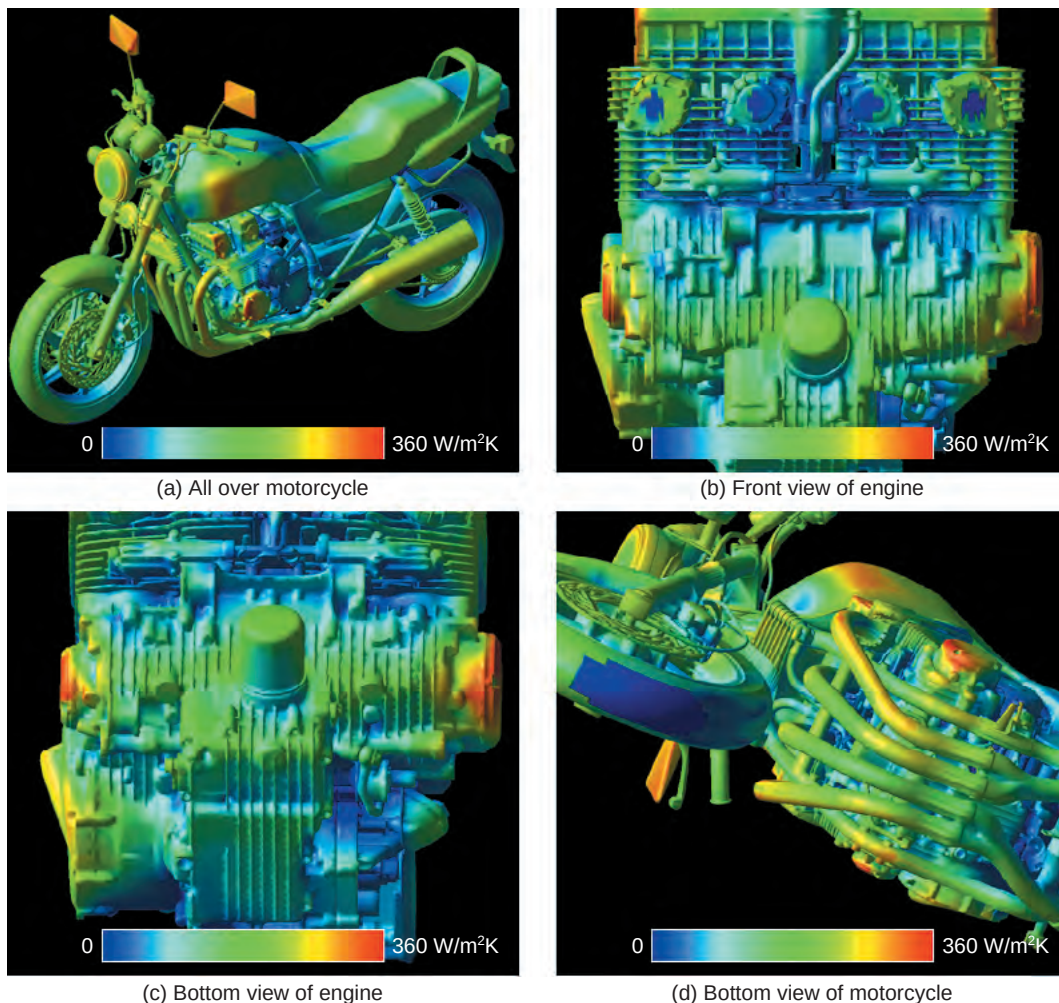


Fig. 15 Heat transfer coefficient at 210 km/h

4.7. Study of Cooling Performance of Auxiliary Equipment

The reason for the high heat transfer coefficients around the oil filter is the fact that the airflow between the front forks is bent downwards and deflected as airflows go over the front fender, and thus a comparatively strong flow strikes the engine, as indicated in the velocity magnitude distribution shown in Fig. 7 (a). Positioning a heat converter where the oil filter is currently located would be one means of making effective use of the airflow generated while the vehicle is running. A proposed engine design is shown in Fig. 16, and an example of analysis results used in studying this design is shown in Fig. 17. The installation angle of the heat converter affects the amount of the airflow through the core, but this effect can also be studied on the desktop by CFD. Due to limited space constraints, a high-capacity heat radiator cannot be mounted, but the method would be effective in preventing the oil cooler from blocking airflow to the area around the spark plugs. These discussions were conducted by mean velocity based on the calculation condition under steady-state flow, but given that strong turbulence was observed at the rear of the front fender in Fig. 7 (c), there is a possibility that an unsteady airflow is being generated.

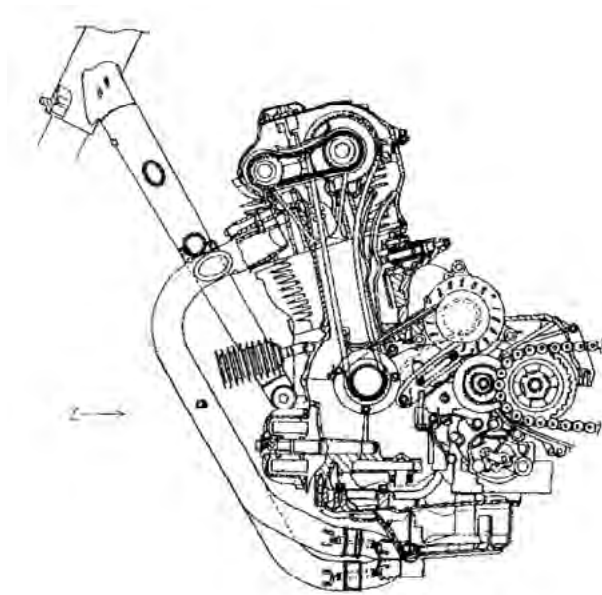


Fig. 16 Cross section of prototype engine

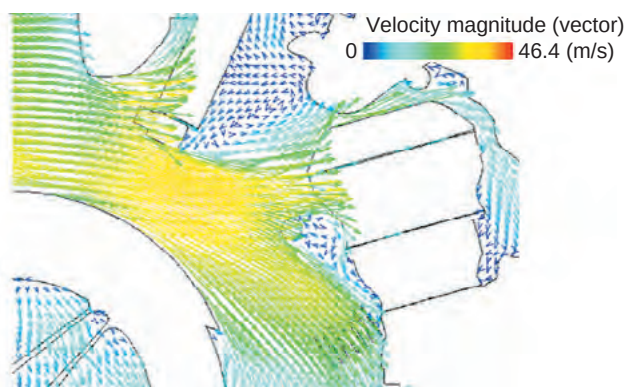


Fig. 17 Velocity vector investigation at oil cooler location

5. Conclusion

An analysis of cooling performance in an air-cooled engine was conducted using the Partial Cells in Cartesian coordinate method. The following conclusions were drawn from this research:

- (1) Analysis of airflow behavior including fine shapes around cooling fins of air-cooled engines became available in a short time using the Partial Cells in Cartesian coordinate method, and practical use of CFD as design support tool became possible.
- (2) The visualization results by the Partial Cells in Cartesian coordinate method correlated well with the visualization results in wind tunnel tests.
- (3) The calculation of the distribution of heat transfer coefficients using Karman's analogy has enabled local studies of heat radiation characteristics, and it has become possible to quantitatively evaluate design specifications. This method can also be used to calculate the temperature distribution in the engine and analyze thermal stress.
- (4) Many and useful knowledge pertaining to cooling fin shapes and the heat transfer coefficient in each part of an air-cooled engine were provided for engine developments.

Acknowledgements

We wish to express our gratitude to all concerned of Honda R&D Sun Co., Ltd. for their cooperation in three-dimensional solid modeling.

References

- (1) Takahashi, Y., Fukuzawa, K., Fujii, I.: Numerical simulation of flow in intake ports and cylinder of multi-valve S.I. engine using PCC method, *Proceedings of International Symposium on COMODIA 94*, p. 529-534 (1994)
- (2) Takahashi, Y., Nomura, T., Ishima, T., Obokata, T.: Numerical analysis on steady state in-cylinder-flow by Partial Cells in Cartesian Coordinate method and verification with PIV measurement, *Transactions of the Japan Society of Mechanical Engineers*, Vol. 71, No. 706, B (2005-6), p. 1694-1701 (2005)
- (3) 3DataExpert, <http://www.deskartes.com>, 2006/1/16
- (4) Tachibana, F., Akiyama, M., Okamoto, Y., Morishita, T.: *Fundamentals of Heat transmission*, Corona Publishing Co., Ltd., p. 44 (1965)

■ Authors ■



Yasushi TAKAHASHI



Yoshitsugu GOKAN