

Research on Prediction of Cooling Air Flow in Small Air-cooled Utility Engines

Toshiro KIURA*

Takashi KUSHIBIKI*

Kazuyuki SHIOMI*

ABSTRACT

This report describes research on the prediction of the cooling air field for air-cooled engines using Computational Fluid Dynamics (CFD). The design of the cooling air field is very important for engine cooling, a factor which has a significant effect on engine performance, fuel consumption, noise, reliability, etc. However, the flow field is very complex because it is developing turbulence caused by a large number of components are located in the flow passages and the cooling air is generated by a centrifugal fan. In order to understand this flow field, calculations were performed using CFD and measurements were taken in an actual engine.

Given practical considerations such as computing time, modeling time, correlation between calculation and test results, formulation of a database, etc., the fan P-Q (pressure vs. volume flow rate) characteristics curve and the flow field were calculated separately when adopting CFD. In addition to visualization using Particle Image Velocimetry (PIV), pressure measurements and measurements of the P-Q characteristics of the cooling fan were done to measure the flow field in the actual engine. The correlation between the results of CFD calculations and measurements was studied. Almost suitable prediction results were achieved by setting the P-Q characteristics at the boundary surface of the rotational area, as obtained from the calculation of P-Q characteristics, as a fan boundary condition for CFD analysis of the cooling air field. This research has enabled the qualitative and quantitative prediction of the cooling air flow of air-cooled engines.

1. Introduction

Small utility engines are used as power sources for construction machinery, agricultural machinery, industrial machinery and so on. It is estimated that approximately 35 million units of spark ignition engines alone are annually produced around the world (according to Honda's survey). They are required to be compact, lightweight and easy to install, and most of them are air-cooled single-cylinder engines. In the industry of utility engines, approaches to

environmental issues have also come under closer scrutiny just as they have with engines for automobiles and motorcycles. As a result, a change from 2-stroke to 4-stroke engines has become widespread⁽¹⁾⁻⁽³⁾. The mainstream of valve driving systems has also evolved from SV to OHV or OHC, resulting in greater power output per unit displacement volume.

The cooling system of these engines must be of simple and lightweight structure. Mainly for this reason, a forced air-cooling system with a rotational fan mounted on a crankshaft is commonly

* Asakahigashi R&D Center

adopted. It is, however, hard for this design to appropriately distribute the required air quantity into various parts of an engine. If the size of a fan were enlarged just to increase the air quantity for cooling, it would result not only in the increase in mechanical loss and the decrease in power output and fuel efficiency, but also in the increase in aero-acoustic noise. Therefore, a more efficient cooling system is required.

Engine cooling capability, namely, heat transfer, is an extremely complex phenomenon which consists of heat conduction, heat transfer and heat radiation. They are due to physical properties, structure and geometry of cylinders and cylinder heads, cooling flow field and surface treatment of die-cast respectively. Moreover, the cooling air rises from a centrifugal fan, whose geometry is restricted by engine performance, design, and other limitations, through a flow passage in which many components in a variety of forms are located, resulting in a flow field with developing turbulence. As the phenomenon is extremely complex as mentioned above, a greater part of the development of a new engine depends on designing based on experimental knowledge and performance tests of prototype engines. It is obvious that the prediction of the cooling capability of air-cooled engines at the design stage would be an effective technique in the development of engines.

Some experiments have been carried out using CFD to clarify the cooling capability of water jackets for automobiles⁽⁴⁾ or cooling air flow in engine compartments of working vehicles⁽⁵⁾ in order to assist in the development of engines. The advantages in applying CFD is the ability to analyze in detail the flow field or heat transfer in locations that have been difficult to measure by the conventional method, and the possibility to provide a guideline for design.

However, there have been relatively a small number of reports on efforts to use CFD to predict the flow field in forced air-cooling systems for small utility engines.

So, the object of this research is to create a design technique that can predict air cooling capability and to optimize engine structure. To achieve this aim, CFD was applied to quantitatively predict cooling flow field of small air-cooled utility engines and measurements were taken to verify the adequacy of such predictions.

2. Calculation Method

2.1. Calculation of Fan P-Q Characteristics Curve

To achieve the calculation of fan P-Q characteristic, the calculation space has to be discretized and at its each element, the conservation laws for the mass and momentum are satisfied, and

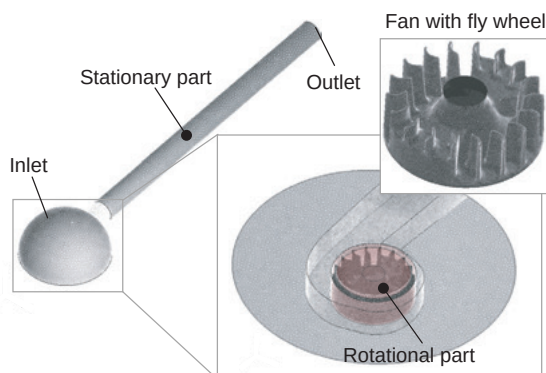


Fig. 1 Model geometry and mesh for calculation of fan P-Q characteristics

the moving boundary problem for rotating region has to be solved. SCRYU/Tetra (Software Cradle Co., Ltd.), which is unstructured grid computational fluid dynamics software, was used for this calculation.

The simulation geometry is shown in Fig.1, which is modeled three-dimensionally to the internal geometry of measurement equipment (refer to section 3.1.) based on JIS Standards⁽⁶⁾. There are two differences between the simulation model and the real measurement equipment. One difference is the inlet. This part is modeled as the finite section instead of the space open to the atmosphere, which is a hemisphere space with an enough length of diameter compared with fan diameter. The other difference is the outlet. This part is replaced with the butterfly valve located at the outlet of pipe with the pressure boundary condition. Therefore, the model produces as a form up to the front of the valve. As for the grid making, triangle meshes of a size appropriate to each region were adopted on the model surface. Then prism meshes were applied for the near wall regions and tetra meshes were applied for the inner space of the model, respectively (see Fig. 1). The purpose of applying this prism mesh is to adjust the wall function parameters (y^+ value). The number of cells were determined to be not more than 1 million cells and as small as possible. This is due to the time limitations of fan performance calculation for practical development.

The phenomenon was regarded as steady state, and Arbitrary Lagrangian Eulerian (ALE) was adopted as the calculation method of the moving boundary problem for the rotating motion of the fan. The model consisted of two parts: the static region and the rotating region. Additionally, the standard k- ϵ model and wall function was adopted as the turbulent model from the viewpoint of convergence, which is based on the turbulence kinetic energy k and the rate of the turbulence energy dissipation ϵ . Pressure boundary conditions were used for both the inlet and the outlet. In actual measurements, the P-Q characteristics curve was measured by raising the fan backpressure using the butterfly valve located at the pipe outlet, while in calculation, the rise of back pressure is treated as the increase of input pressure boundary value.

Calculations were done by use of the above-mentioned models, grids and boundary conditions. To evaluate the calculation results, the values, which are static pressure and flow rate, obtained from measurements and calculations were directly compared. From the results of evaluation, a method was constructed to make the calculation results asymptotically closer to the measured results by controlling the wall function parameters (y^+ value) and adjusting the thickness of prism mesh.

2.2. Calculation of Flow Field

To calculate the flow field, the calculation space has to be discretized and at its each element, the conservation laws for the mass and momentum have to be satisfied, and the moving boundary problem for the rotating region has to be solved. VECTIS (Ricardo plc), which is commercially available structured grid computational fluid dynamics software, was used for this calculation. This is because one of the important abilities of CFD software in the engine development are to ensure to read and modify directly the 3-D model used at the design. This software has higher potential compared with other software for importing geometry, and Cartesian grids make it easy to control the cell size locally.

For calculation of the flow field, the three-dimensional model of an actual engine was used as shown in Fig. 2. A point that the model is different from the measurement equipment was the fact that the outer space open to the atmosphere was modeled on the

finite section (a rectangular parallelepiped that has a sufficient area against the size of the engine). The meshes consisted of a required minimal number of meshes, and the sizes are determined to be able to sufficiently express the geometry of the engine, and in other external spaces, gradually larger mesh sizes were applied.

Steady state condition was applied for this calculation. At the fan part, a fan model was adopted, which could produce a suitable swirl flow by inputting the P-Q characteristics curve. Calculations were performed by substituting the P-Q characteristics curve on the boundary surface of the rotating region, which was obtained from the previous calculation, for this fan model. For the turbulent model, the standard k- ϵ model and wall function were adopted.

Calculations were done based on the above-mentioned models, mesh, and boundary conditions. Additionally, verification was done by use of both the velocity vector obtained from flow visualization via PIV (refer section 3.3.) and the value of total pressure measured as supplementary data (refer section 3.4.).

3. Measurement Method

3.1. Measurement of Fan P-Q Characteristics Curve

Figure 3 shows the measurement equipment. It consists of a motor, a fan, a scroll, a pipe, a butterfly valve, a volumetric flow rate meter, and a static pressure measurement point. Specifications of the measurement equipment are shown in Table 1.

Measurements were taken constantly set at 3 000 rpm continually. The fan P-Q characteristics curve was measured by adjusting backpressure with the butterfly valve installed at the outlet of the pipe. Each point of data was measured when the pressure and flow rate became stable.

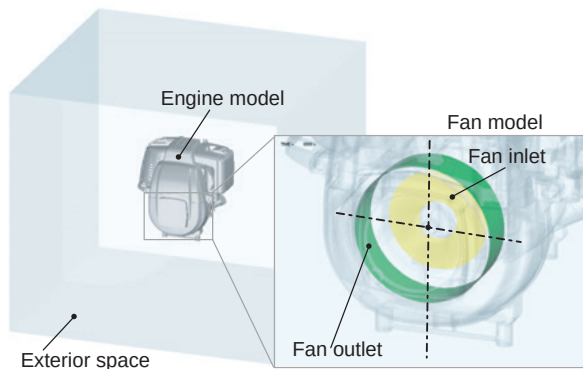


Fig. 2 Model geometry for flow field

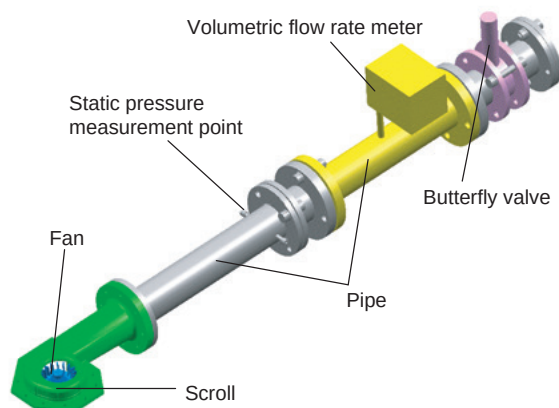


Fig. 3 System for measurement of fan P-Q characteristics

Table 1 Measurement system specifications

Measurement standard	JIS B 8330
Motor	ORIENTAL MOTOR VEXTA FBLM 5120
Pipe	JIS B2238 80A
Volumetric flow rate meter	Japan Flow Control FOX 10A (Range: 15 - 300 Nm ³ /h (Accuracy $\pm 2\%$))
Pressure meter	YOKOGAWA JP208

Table 2 Test engine specifications (GX22)

Engine type	4-stroke single cylinder (OHV) spark ignition
Bore $\times$ Stroke (mm)	33 $\times$ 26
Displacement (cm ³)	22.2
Dry weight (kg)	3.3
Dimensions L $\times$ W $\times$ H (mm)	210 $\times$ 251 $\times$ 249

3.2. Test Engine and Drive Unit

As the test engine, a small air-cooled utility engine, GX22 (Fig. 4) was selected. This engine was developed as the world's first 4-stroke spark ignition engine capable of freely inclining 360 degrees, and as a power source for handheld tools with clean emissions and superior economical efficiency. The engine specifications are shown in Table 2. The testing equipment is also shown in Fig. 5. As the object of this test is only to measure the flow field, the output shaft of this test engine is driven by a battery-powered motor, and a transparent cover through which laser beams



Fig. 4 Test engine (GX22)

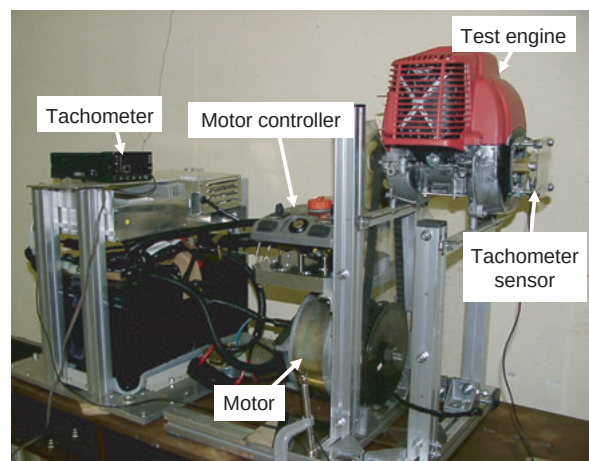


Fig. 5 Test engine bench

can pass is provided. The drive unit consists of a driving motor, a control unit, batteries and an optical rotational sensor.

3.3. Measurement of Flow Field

The flow visualization for the cooling flow field was performed by PIV system. PIV technique uses two special instantaneous images, captured at quite short intervals, of the flow field illuminated by laser. Cross-correlation of the images resulting from two shots was analyzed to calculate the length and direction of displacement of the seeding particles. To obtain two images, two independent laser units, a CCD camera, a synchronizer, and a seeding generator are needed. As an illuminant, two coaxial laser units are used. These laser beams are expanded in a sheet form through the cylindrical lens mounted on the oscillating unit outlet, and they are pulse-emitted twice by controlling the pulse separation (interval of double pulses). Laser oscillation is synchronized with the CCD camera through the synchronizer. In addition to the equipment, “Frame-Straddling”, which is a technique to position the two laser pulses on sequential CCD frames, is adopted for capture at quite short intervals. Seeding particles, which are mixed into the flow upstream, are captured on two images. Spatial velocity vectors of instantaneous flow field are obtained by analyzing with cross-correlation, for each displacement of particles on the pair of images. The specifications of the measurement equipment used are shown in Table 3.

In this experiment, several visualization windows were created on the engine cover (fan blowing area and exhaust side) in order to visualize the flow within the cover. Figure 6 shows the three PIV sections. All the measurements were taken at 3 000 rpm continually. 100 pairs of captured images were taken and each spatial velocity vector of instantaneous flow field was used for averaging. The frame straddling interval was set at 10 μ s and cross-correlation was performed on the interrogation area with 32 pixels in size. To enhance the precision, images captured at PIV measurement section A shown in Fig. 6, were divided into four parts and combined to one flow field after calculation of averaging.

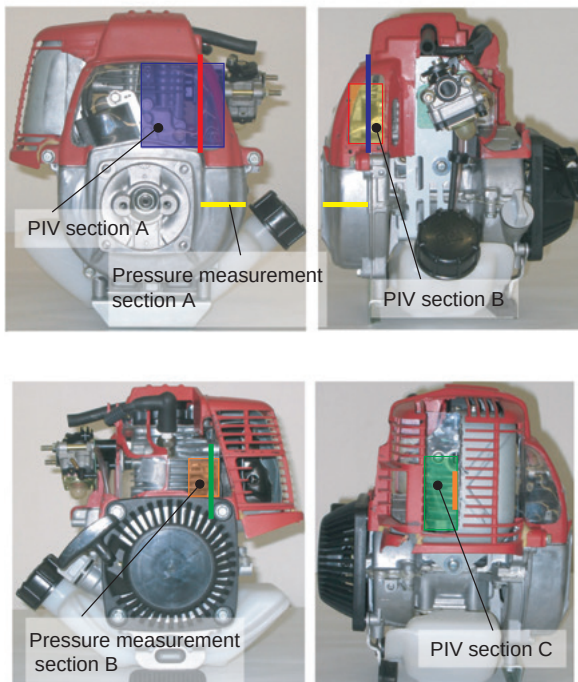


Fig. 6 PIV and pressure measurement sections

Table 3 PIV system specifications

Camera

TSI PIVCAM 10-30 Cross/ autocorrelation CCD Camera	
Light sensitive pixels	1 008 (H) $\times$ 1 018 (V)
Frame rate (MAX) (frame/s)	30
Frame straddling time (MAX) (μ s)	0.4

Optional system components

TSI LASERPULSE™ Y12-15	
Energy (per pulse) (mJ)	12
Repetition rate (max.) (Hz)	15
Wavelength (nm)	532

Others

Seeding generator	DLR Seeding Generator LSG-500 (Particle diameter 1 μ m)
Analysis software	INSIGHT™ TSI

Table 4 Pressure measurement system specifications

Pressure probe

Diameter (internal/external) (mm)	0.8/1.2
-----------------------------------	---------

Pressure transducer

YOKOGAWA JP208	
Range (Pa)	0 - 100 (Accuracy $\pm$ 1%)

3.4. Measurement of Pressure

As it is related to an inside flow of a small utility engine, the area where PIV measurements can be adopted is restricted spatially. For this reason, pressure measurements were performed at two positions, which are the scroll outlet and the space between exhaust fins shown in Fig. 6, as the supplementation for PIV measurements.

For the measurements, a total pressure probe, a traverse unit and a differential pressure transducer were used. Their specifications are shown in Table 4. The engine operating conditions here were the same as those of the flow field measurements.

4. Results

4.1. Fan P-Q Characteristics Curve

Figure 7 shows the results of the fan P-Q characteristics curves at 3 000 rpm obtained from calculation and measurement. The fan P-Q characteristic curve expresses the relationship between static pressure values and volumetric flow rates of the fan. The solid lines in the figure show the values at the measurement points shown in Fig. 3; the black one shows the measured results and the red one shows the CFD results. The broken line shows the CFD results relationship between the pressure drop between the inlet and the outlet boundary of the rotational region (see Fig. 8), and the flow rate. The latter P-Q characteristics curve is adopted as the input data for the flow field calculation. As an example of the results of P-Q characteristics curve calculations, Fig. 8 shows the velocity vector at the center section of the blade height and pressure distribution on the boundary surface between the rotational and statistic region at 80 Pa of backpressure.

According to Fig. 7, the calculation results correctly predict the measurement results within the low to medium backpressure range (pressure 0 – 90 Pa). This is because the relationship between the wall function parameters (y^+ value) and the wall friction loss is examined in advance, in the solver. Based on the pre-examined results, the thickness of boundary layer mesh has been controlled to make calculation results become asymptotically closer to the measured results. In actual practice, 30 to 100 were aimed as the target of the y^+ value, and the suitable thickness of prism mesh was locally determined. In addition, the same mesh was used on all backpressure conditions of this calculation.

On the other hand, Fig. 7 shows that the difference of each result tends to expand at the high back pressure range, this is due to imbalance of pressure between the backpressure set at the outlet and the pressure generated with the fan as increasing the backpressure. Under such imbalanced conditions, the backward flow against the normal vector occurred locally on both the inlet and outlet. And finally the calculation convergence and stability

got decrease resulting in lowering in accuracy. This is a limitation of applying this simulation model. Generally, the application range of backpressure for centrifugal fans is within the low to medium, which is highly efficient. Thus this method can be applied for the prediction of fan P-Q characteristics curves.

4.2. Flow Field Visualization Results

Figure 9 shows the comparison between the flow fields visualized with PIV and calculated with CFD under the same condition, in two sections inside the fan cover and in one section vertical to the cooling fins on the exhaust side.

According to Fig. 9, the comparison at each section shows that the CFD qualitatively predicts the flow field well. While the quantitative comparison of the flow field has a difference of approximately 10% in terms of the absolute velocity, it is considered to be that one of the main reasons is that it was affected twice by the geometry of fan scroll in the fan P-Q characteristics curve analysis and the flow field analysis. In this way of analysis, this difference is unavoidable. Thus there is room for more work to be done in this area.

4.3. Flow Field Pressure

Figures 10 and 11 show the comparison of total pressure results which were obtained from the measurements and CFD. These comparisons are done at the sections indicated in Fig. 6 as the pressure measurement section A and B respectively (fan blowing area and exhaust side).

According to Figs. 10 and 11, in comparison of pressure field, CFD made an accurate forecast of pressure distribution of each section in the same way. However in terms of the absolute values, a difference of about 10% occurred in the same manner as in the flow field.

5. Discussion

The cooling fan for the test engine dose not have cyclic symmetric geometry for each of the blades, because the fan is united within the flywheel, magnet and the counterweight. The configuration of the fan produces the unsteady and intermittent flow with cyclic tendency depending on the angular position of fan. In addition, it is assumed that the complex cooling air passage induces flow separation in various areas, and develops the turbulence of the flow field. For such strongly fluctuated and fully developed turbulence as this, PIV performed space-averaging in the inspection area and time-averaging for a certain period of time in its measurement method and processing. In addition, in CFD, an assumption of steady state was applied for flow field and a time-averaged formula, which is Reynolds Averaged Navier-Stokes Equation (RANS), was used as the turbulent model. Thus it can be easily presumed that many delicate issues are involved in the correlation between this experiment and the calculations. It is, however, highly significant that these phenomena established a generally good correlation both qualitatively and quantitatively in the numeric calculations.

On the other hand, the predictions, which are the fan P-Q characteristics curve, the flow around the fan and the flow in the cooling air passage, make it possible to analyze the flow field for the reduction in the mechanical loss of the fan and in aero-acoustic noise etc. Also, it goes without saying that additional consideration needs to be given to the heat transfer issue at the viewpoint of predicting the cooling capability. Prediction of temperature and radiation of heat from the surface still remains to be ascertained, it

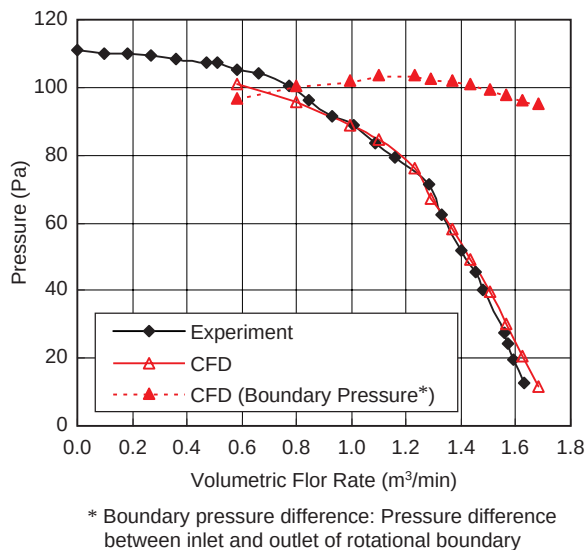


Fig. 7 Results of calculation and measurement of P-Q characteristics curve (3000 rpm)

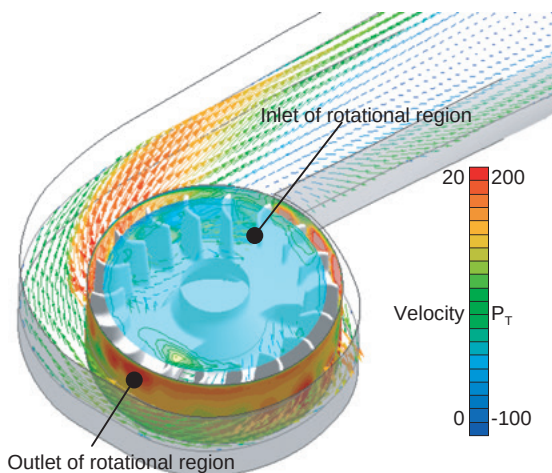


Fig. 8 Velocity vector at 80 Pa of back pressure

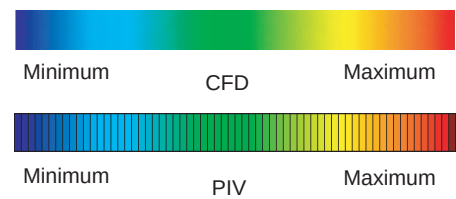
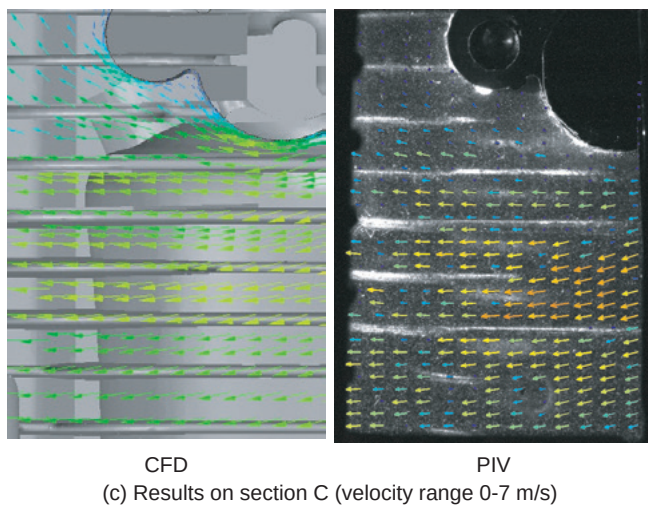
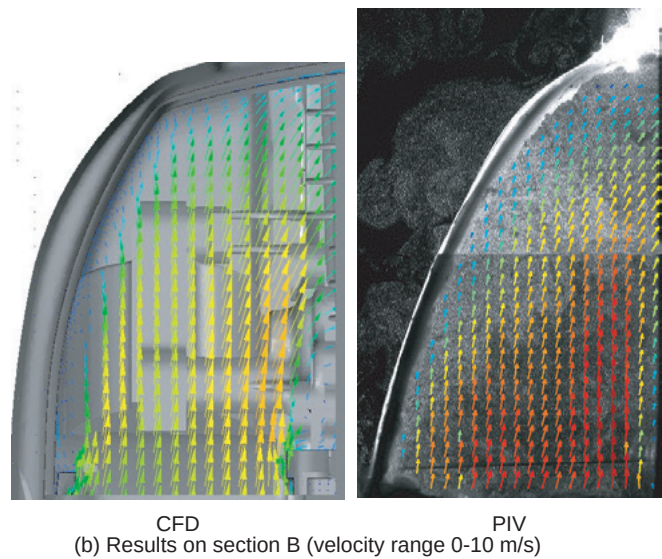
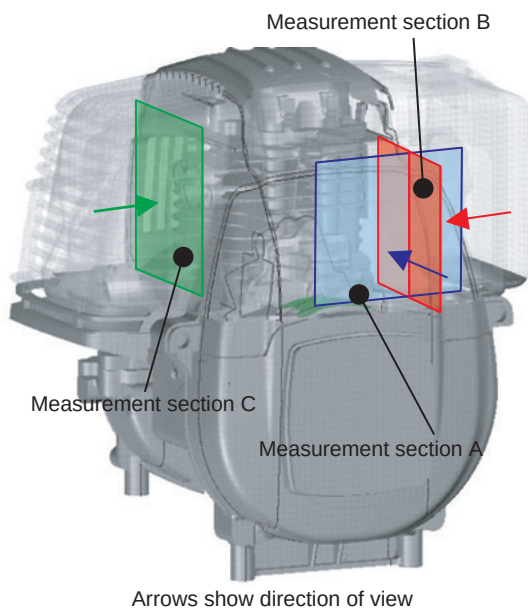
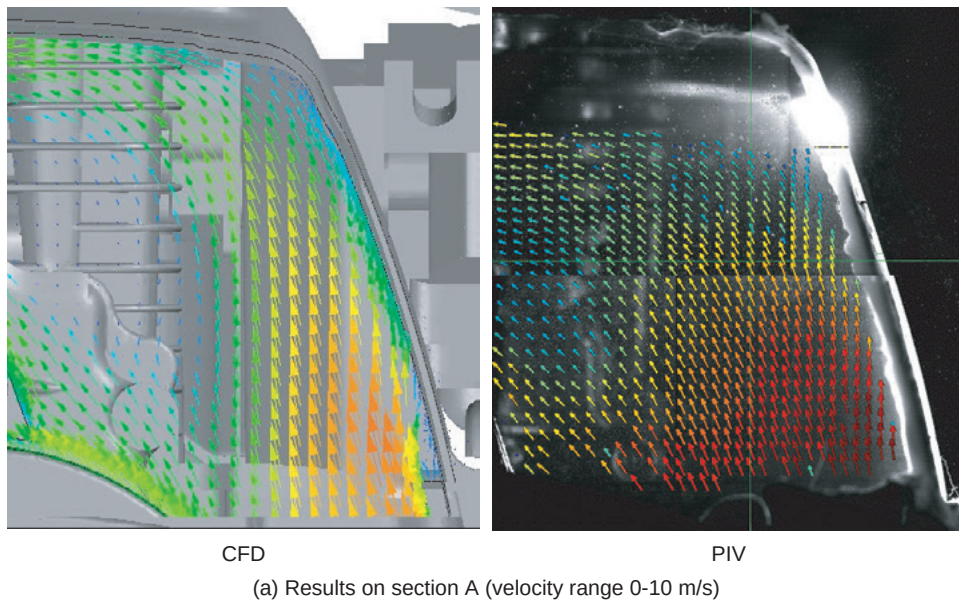


Fig. 9 Comparison of CFD and PIV results for cooling air flow

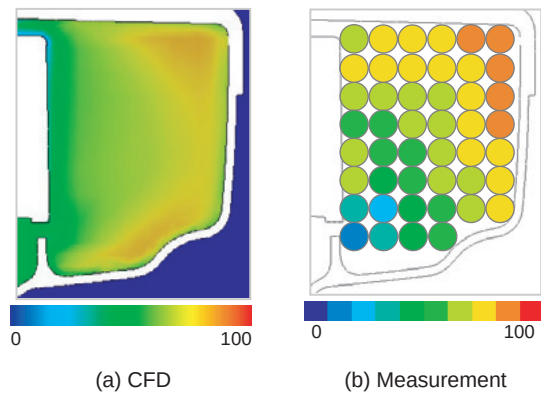


Fig. 10 Pressure distribution at pressure measurement section A (fan - scroll outlet)

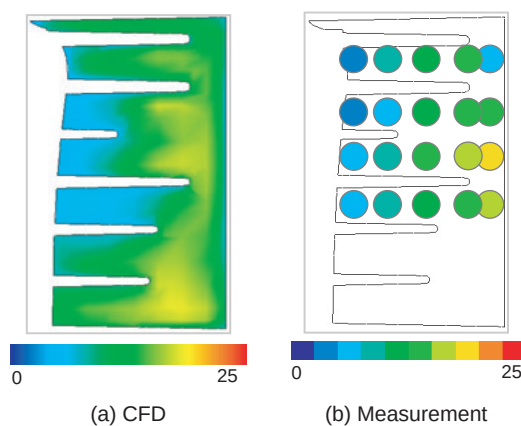


Fig. 11 Pressure distribution at pressure measurement section B (Exhaust fins)

is clear that more research is necessary to establish the technique which can predict the heat flux distribution from in-cylinder.

6. Conclusions

The object of this research was to create a design technique capable of predicting the engine cooling capability and optimizing engine structure. As a first step, the technique of predicting the flow field of the cooling system of an air-cooled engine qualitatively and quantitatively using CFD was studied by using a small utility engine for hand-held power equipment as test engine. The following were learned from these results:

- (1) In the calculation of the fan P-Q characteristics curve, a calculation method to make the calculation results closer asymptotically to the measured results was established. By using the calculation, it became possible to examine the performance of a fan unit only.
- (2) A good coincidence was regarded as the results of the flow field obtained with CFD, and of the measurements by PIV etc. In this calculation method, first, the P-Q characteristics curve of the fan alone was calculated. And the calculations for flow field were performed by substituting the P-Q characteristics curve on the boundary surface of the rotating region which was obtained from the previous calculation, for this fan model.

- (3) It became possible to qualitatively and quantitatively predict and calculate the cooling flow field using the technique created through this research.

It writes in addition that the calculation method created here has been put into practical use in the subsequent development of air-cooled engines; for example, it has enhanced the accuracy of prediction made at the design stage, and contributed to enhancing the development quality.

Acknowledgements

The authors would like to thank Dr. Yasuaki Kohama and Mr. Fukuo Ohta of Tohoku University Institute of Fluid Science, Dr. Satoshi Kikuchi of Gifu University Faculty of Engineering Department of Mechanical and Systems Engineering, and all other related persons. They assisted us in the flow visualization using PIV.

References

- (1) Shiga, M., Nishida, T., Ryu, Y., Tsugekawa, T., Higashi, K.: Development of Multipositional Small Four-stroke Engine, HONDA R&D Technical Review, Vol. 9, p. 44-50 (1997)
- (2) Ito, K., Ryu, Y., Yoneda, S., Warashina, M., Edamatsu, S.: Development of GX25 4-Stroke Engine for Handheld Equipment, Honda R&D Technical Review, Vol. 14, No. 2, p. 1-8 (2002)
- (3) Maeda, T., et al.: Development of All Position-Operable Small Four-Stroke Engine, Small Engine Technology Conference '97, 972104, (1997)
- (4) Xin, J., et al.: Theoretical Consideration to Improve Engine Cooling and Application of Coupling 3D Combustion Simulations with Heat Transfer in Water Jacket and Components, Honda R&D Technical Review, Vol. 15, No. 2, p. 117-126
- (5) Saito, M., et al.: Development of Low Noise Working Machine based on CFD Technique, Small Engine Technology Conference 2002, 20024295, 2002-32-1802, (2002)
- (6) JAPANESE INDUSTRIAL STANDARD B 8330:2000 Testing methods for turbo-fans, Japanese Standards Association, (2000)

■ Authors ■



Toshiro KIURA



Takashi KUSHIBIKI



Kazuyuki SHIOMI