

Establishment of an Intake Port Design Support Tool Using CFD

Yasushi TAKAHASHI*

Yasuo MURAKAMI*

ABSTRACT

An intake port design support tool using CFD (Computational Fluid Dynamics) software based on the PCC (Partial Cells in Cartesian coordinate) computational mesh method was applied during the development of a motorcycle engine. CFD analyses were performed with a relatively quick turn around time due to the reduced mesh generation time. To ensure sufficient accuracy in actual applications, the CFD results have been validated against measurement data. In addition, recommendations for the mesh setup and details of implemented post processing improvement are described. In the present report, it has been confirmed that the developed CFD software is a robust design support tool.

1. Introduction

Since the shape of the intake ports of an engine greatly influences the characteristics of power output and adjacent components, it should be determined at the early stage of engine development. Therefore, developing a design support tool capable of evaluating flow features prior to prototyping has been desired to replace the conventional method employing steady state flow test rig. Rapid improvement of computer performance in recent years has made it possible to perform CFD calculations of port flows with desktop computers. However, most commercial CFD codes require much time for mesh generation and are not suitable as a design support tool for motorcycle engines because such engines are usually developed in an extremely short term. To address that issue, the authors have developed a PCC (partial cells in Cartesian coordinate) method which employs an orthogonal grid of straight lines and enables shortening mesh generation time⁽¹⁾. This work aims to construct a support tool for an intake port design with the PCC method.

2. Overview of Tools

mixture into the engine cylinder, so flow resistance through the port is used as one of the indicators for engine power output. For port flow resistance calculations in most CFD codes, a series of tasks shown in Fig. 1 (a) are necessary and a new mesh must be created for each geometry shape. The general process of mesh generation consists of the following tasks: initial mesh generation → mesh quality check → improvement. Recently, the initial mesh generation has been automated, and not much human labor is required. On the other hand, the mesh quality check and improvement tasks have a profound effect on calculation stability and accuracy, and highly depend on the know-how of an operator. These considerations have been an obstacle to the full automation of mesh generation. In such a situation, by using the PCC method shown in Fig. 1 (b), a great deal of reduction in mesh generation time has been accomplished.

In the PCC method, calculations of fluid dynamics are carried out with an orthogonal grid of straight lines among any geometric shape, for example as shown in Fig. 2. Though many cell shapes other than hexahedron cells appear around a wall surface, mesh update is not necessary because the CFD calculation has been enabled among these cell shapes. Therefore, the grid data also can be applied for other calculations such as similar shapes and each valve lift.

The intake port is the component to introduce the air fuel

* Asaka R&D Center

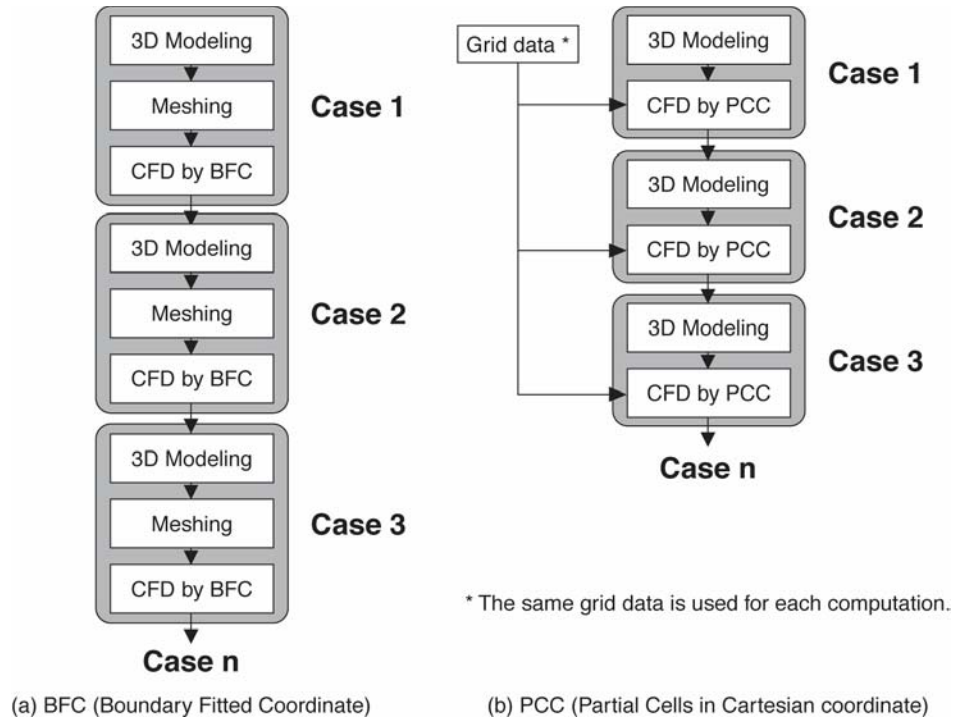


Fig. 1 Comparison of work flow between BFC and PCC

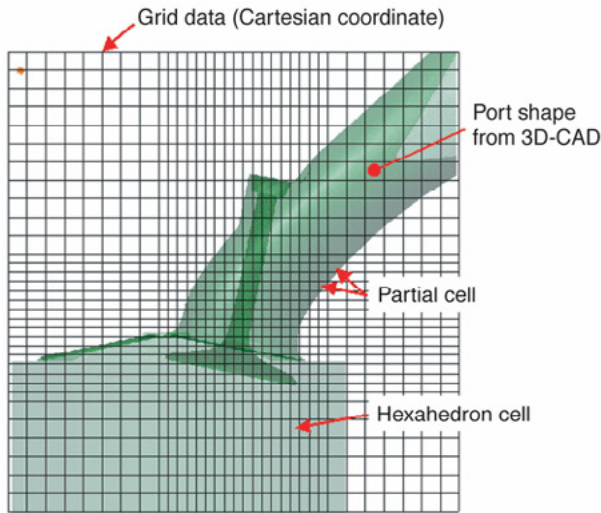


Fig. 2 Example of port shape and Cartesian coordinate

3. Evaluation Parameters for Intake Port Characteristics

Effective valve opening area (Z) defined in Eq. (1) is employed as a parameter indicating ease of air flow through the port.

$$Z = \frac{\dot{m}}{\sqrt{2\rho\Delta p}} \quad (1)$$

Here, $\dot{m}$ is mass flow rate, ρ is density of air, and Δp is pressure drop over the port.

In addition, $SUM(Z)$ defined in Eq. (2) is used to consider a valve lift curve, which affects engine volumetric efficiency.

$$SUM(Z) = \int_{\theta_1}^{\theta_2} Z d\theta \quad (2)$$

Here, θ_1 and θ_2 are the crank angles at valve opening and valve closing, respectively.

Since the power output characteristics are predicted using these parameters during engine development, in this paper the effective valve opening area (Z) has been selected as the indicator to evaluate intake port characteristics.

4. Preparations for Application of Tools

In practical use as a design support tool, the main theme is to achieve both short analysis time and sufficient accuracy. Short analysis time was achieved in the PCC method. However, general orthogonal grid methods by straight lines show considerable concerns in calculation accuracy^{(2), (3), (4)}. Therefore, verifications of the accuracy of the effective valve opening area (Z) were carried out using several types of ports. In addition, to improve the utility of the software as a design support tool, several functions of post-processing were added. Note that the code in this paper is an upgraded version combining “logarithmic law of the wall” and a low Reynolds number $k-\varepsilon$ turbulence model⁽⁵⁾.

4.1. Comparison with Measurement Values

First of all, a verification of the calculation accuracy concerning the details of port geometry was carried out. Figure 3 shows the base port shape, and Figs.4 (a) and (b) show improvement rate of the effective valve opening area (Z) in increasing +2 mm and +3 mm port diameter each, and comparison between measurement and calculation values are presented. The effective valve opening area (Z) improves in the whole valve lift by increasing port diameter. Since the improvement rate between measurement values and

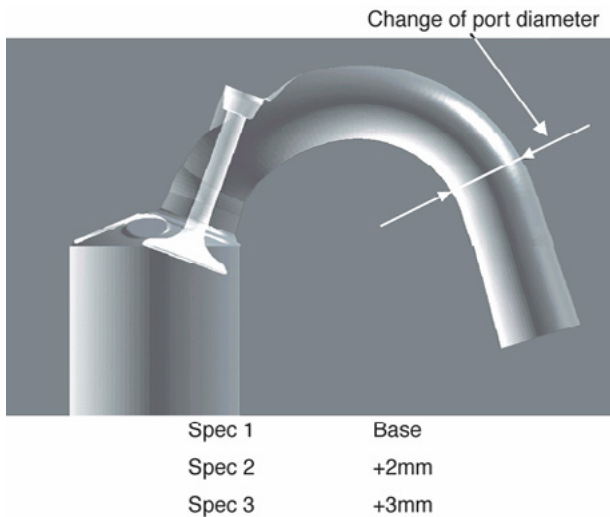


Fig. 3 Port shape for verification by changing port diameter

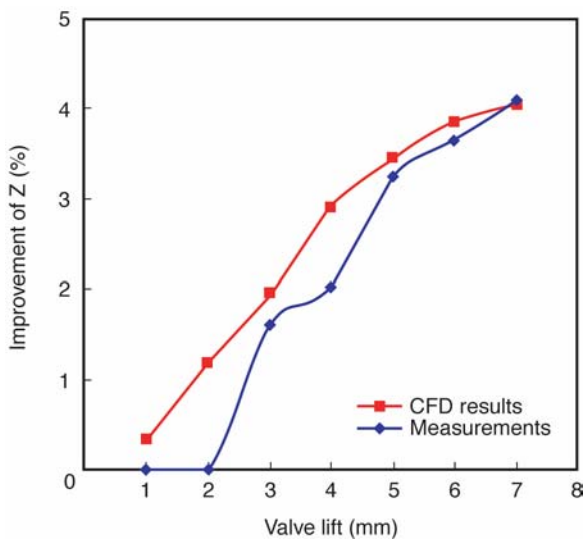


Fig. 4(a) Improvement of Z for Spec 2 port (diameter +2 mm) compared to Spec 1 port

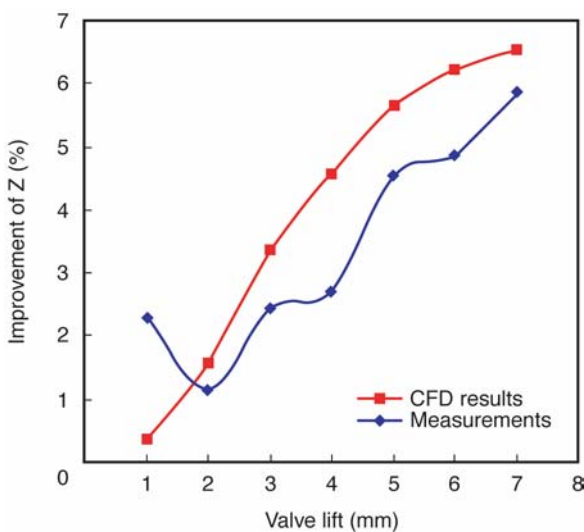


Fig. 4(b) Improvement of Z for Spec 3 port (diameter +3 mm) compared to Spec 1 port

calculation values agreed, it was confirmed to have much potential for examining geometry changes in detail.

Next, the calculation accuracy with a big port geometry change was examined. Figure 5 shows two port shapes used in the verification. Figure 6 shows comparison of measurement and calculation results in terms of the effective valve opening area (Z) at each valve lift. Although the two ports differ greatly in several dimensions such as port length, port bend angle, existence of a hollow around the injector tip, etc., the correlation between measurement values and calculation values was almost good. A difference between the two was observed around 5 mm of valve lift; these are considered due to differences in the scale of separation vortices in the calculations and measurements⁽⁵⁾.

Thus, it was confirmed that the issues of calculation accuracy concerning the orthogonal grid by straight lines were resolved by employing the PCC method.

4.2. Setting of Recommended Grid Data

To guarantee the calculation accuracy as a design support tool, a minimum standardization was needed to provide a fine grid size at places of sudden changes in flow. In addition, it was necessary to use the same grid data to avoid the issue of grid dependence of results⁽⁶⁾. The recommended grid data was provided mainly around the valves.

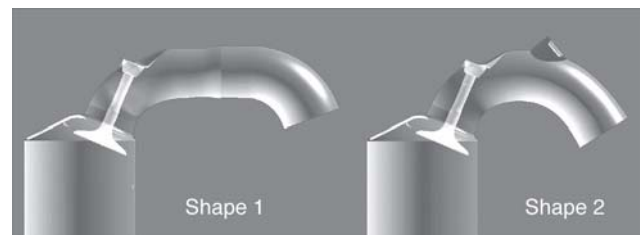


Fig. 5 Port shapes for the following comparisons

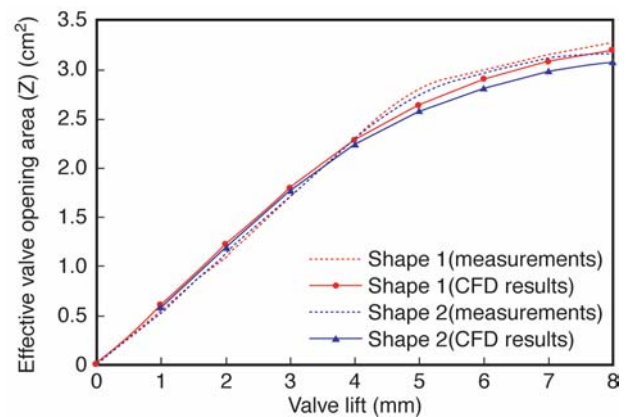


Fig. 6 Comparison between measurement results and CFD results for Shape 1 port and Shape 2 port

Figure 7 shows comparisons between measurement and calculation values of SUM(Z) of 16 engines with displacements from 50 cm³ to 500 cm³ per cylinder. By setting the recommended grid data, the differences between the measurement and calculation values were held within $\pm 4\%$. Thus, it has become possible to guarantee sufficient accuracy as a design support tool.

Some calculation values in the figure are estimated rather high from the correlation line. However, since these were limited cases caused by large separation vortices inside the ports or just under the valves, it appears that an issue remains with the current turbulence model. Note that the recommended grid data established here were two sets for small-displacement engines and for large-displacement engines, and the grid size around the valves was set between 0.8 mm and 1.2 mm.

4.3. Automation of Calculation of Effective Valve Opening Area

Pressure measurement points of the steady state flow test rig are in a chamber with a large volume and at a point open to the atmosphere, so that the static pressure is equal to the total pressure. To make calculation conditions correspond to the measurement conditions, a chamber to imitate the atmosphere was added at the port inlet of the CFD geometry model. By adding a large chamber, the flow velocity in the chamber is ignored, so the conditions correspond to the experiment. However, there is a drawback that large volume of the chamber requires a longer calculation time by increasing total mesh numbers. On the other hand, small size of chamber derives the necessity to calculate the total pressure from the static pressure and the flow velocity. Therefore, both the expressions to calculate the total pressure and Eq. (1) were incorporated into the CFD code to easily obtain correct values of the effective valve opening area (Z), regardless of the size of the chamber.

4.4. Equipment of Function to Indicate Vortex Magnitude

Because only the Z value is provided from the conventional steady-state flow tests, the only way to optimize

the port shape has been to depend on the designer's experience and instincts for finding the place and method. Improving the effective valve opening areas means reduction of pressure drop over the port. However, it is difficult to identify the locations of momentum losses by examining the pressure distribution. In order to make it easier to identify those locations, the vortex magnitude ($|\omega|$) defined in Eq. (3) was considered, and its graphic display function has been equipped in a post processing program. Strictly speaking, the vortex magnitude does not quantitatively express pressure drops, but it smooths the way for further work, since it identifies the locations that need to be optimized.

$$|\omega| = \left[\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right)^2 + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)^2 \right]^{\frac{1}{2}} \quad (3)$$

Above, x, y and z are the coordinate axes, and u, v and w are the flow velocity in each direction.

Next, the utility of the vortex magnitude display will be shown. Figure 8 (a) displays the vortex magnitude distribution of the base shape. Vortices due to flow separation are visible in the red on the lower surface of the curve. Although modifications to the port bend angle are desired to reduce the pressure drop, the separation on the bottom surface of the port curve was controlled by increasing port diameter due to engine layout. As shown in Figs. 8 (b) and (c), vortex magnitude decreases with the increase in port diameter by +2 mm and +3 mm, and as shown in Figs. 4 (a) and (b), the effective valve opening area (Z) was improved.

5. Examples of Application in Motorcycle Engine Development

An example will be described that aims to improve both power output and throttle response simultaneously by intake port modifications in a 4-valve engine. The smaller port volume between the throttle valve and the intake valves help to obtain the higher engine acceleration rate by throttle control. Therefore, 5% reduction of port volume against base ports was aimed while maintaining the effective valve opening area (Z).

Figure 9 presents a cross-sectional view of the base ports, and Fig. 10 (a) shows its vortex magnitude. By the indication of high vortex magnitudes in three locations of A, B and C, it was found that these places should be modified.

For location A, it is well known that small radii of the valve stems are helpful to reduce the pressure drop. However, in this example improvements were attempted only by modifying port geometry. Figures 10 (b) and (c) show the velocity magnitude distributions in section D – D and section E – E in Fig. 9. At the downstream of the bifurcation, it is visible that flow speed of port center becomes rather high. Generally, it is desirable to create a flow velocity distribution in a port as uniform as possible, but flow in a port is also governed by the shape of the combustion chamber and the shape of insulator which connects a carburetor to the ports. Therefore, attempting a uniform flow distribution by only port geometry revisions often results in the increase of pressure drop. As shown in Fig. 11, the change of the port bifurcation position and optimization of port sidelines increased the flow toward the centerline of the ports. Thus, the colliding flow to the valve stems could be reduced.

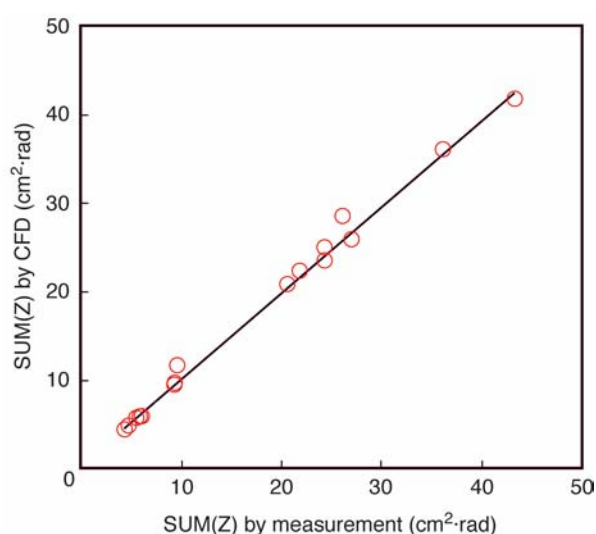


Fig. 7 Comparisons of SUM(Z) between CFD results and measurement results

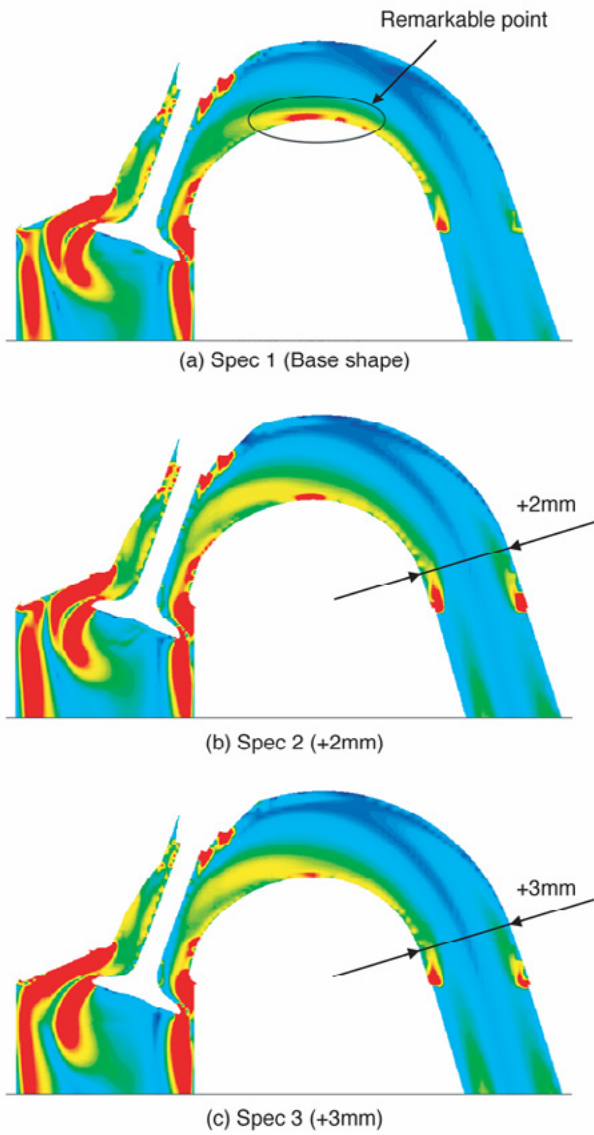


Fig. 8 Display of vortex magnitude for finding defective shape

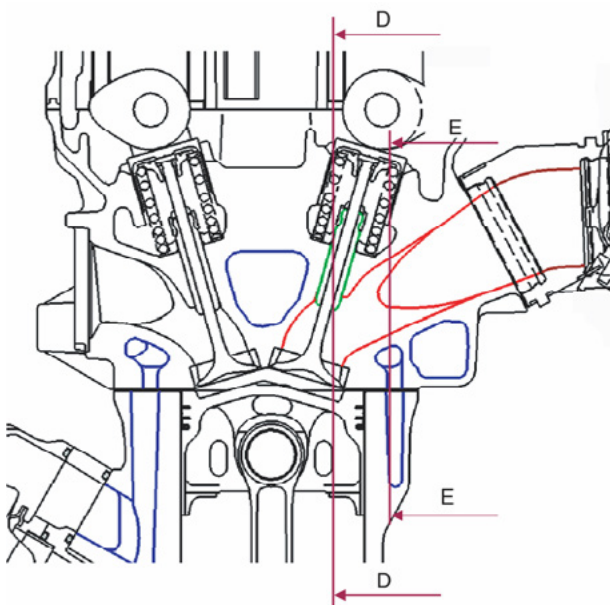


Fig. 9 Drawing of intake port

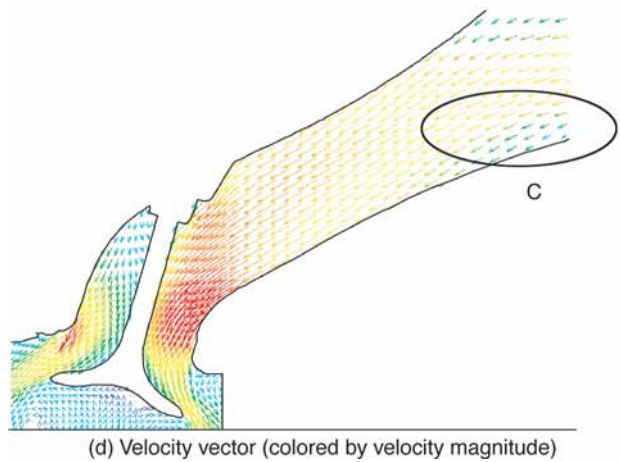
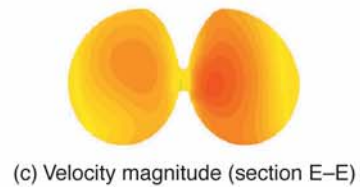
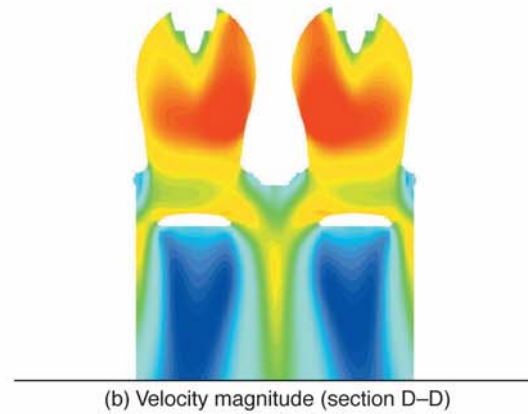
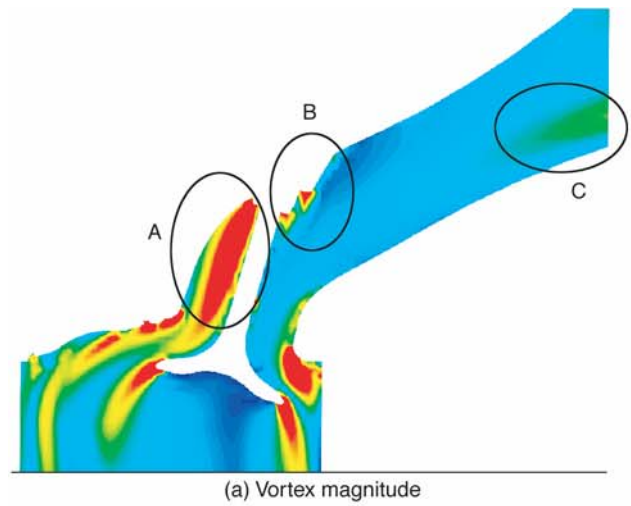


Fig. 10 Display for analyzing port shape

For the location B, a decrease of the inflow angle of the ports was first attempted to reduce the protrusion distance of the valve guides. However, a weak separation occurred at the lower port wall on the curve, and the calculation result of the effective valve opening area (Z) was decreased against expectation. Instead, shapes around the valve guide were smoothed as shown in Fig. 11.

At the location C, the reasons for the vortices were examined together with the velocity vector display shown in Fig. 10 (d). The results indicated separation at the lower wall of the insulator because the ports and the insulator were not smoothly connected; the flow tended toward the upper side of the port (toward the valve guide). Thus, port volume was reduced while optimizing the lower wall of the port to smoothly connect to the insulator.

As mentioned above, countermeasures with CFD allowed a 4.2% reduction in port volume and a 2.4% improvement in SUM(Z). In an engine performance test of the prototype engine, it was confirmed that both the maximum power output and the throttle response characteristics were improved compared with the base port configurations. Thus, the utility of this tool was confirmed.

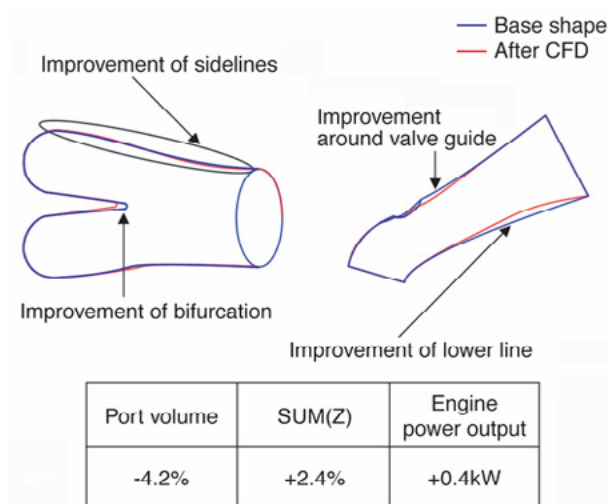


Fig. 11 Examination points and results

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, International Symposium COMODIA, p. 529-534 (1994)
- (2) Ichikawa, O., Fujii, K.: Study of boundary conditions on solving flow field around arbitrary shape by using Cartesian grid (in Japanese), The 13th Japan CFD Symposium, F03-3 (1999)
- (3) Matsunaga, N., Himeno, R.: An approach of finite difference method for general boundary in orthogonal coordinates (in Japanese), The 13th Japan CFD Symposium, F03-5 (1999)
- (4) Ono, K.: Product Design using Cartesian Mesh method (in Japanese), Nagare (Journal of The Japan Society of Fluid Mechanics), 21, p. 16-25 (2002)
- (5) Takahashi, Y., Nomura, T., Ishima, T., Obokata, T.: Analysis on Flow Around a Rectangular Cube by the Numerical Simulation of PCC Method with LDA Verification (in Japanese), Transactions of The Japan Society of Mechanical Engineers, 68-666, B, p. 331-337 (2002)
- (6) Takahashi, Y., Fujii, I.: Numerical Simulation of Gas Flow in Cylinder with PCC (Partial Cells in Cartesian coordinate) Method (in Japanese), The 12th Internal Combustion Engine Symposium, p. 91-96 (1995)

6. Summary

- (1) A design support tool for intake ports was constructed by verifying calculation accuracy of the PCC method, by setting recommended grid data and by improving the post processing functions.
- (2) The utility of the tool was verified for use in development of motorcycle engines and in the tuning of engine specifications prior to prototyping.

By the establishment of a quick and convenient design support tool that significantly reduces mesh generation time, its applications have rapidly spread for various examinations of intake port design at several stages of engine development. Especially, the applications in the initial design stage are increasing, because the tool proves to be very helpful to shorten design optimization time.

■ Authors ■



Yasushi TAKAHASHI



Yasuo MURAKAMI