

Technique for Prediction of Automotive Transmission Lubrication Performance

Daiki SAEGUSA* Shinji KAWAI*

ABSTRACT

A technique was created for rapidly and accurately predicting oil flow within a transmission.

This research employs the finite difference method to perform analysis, and handles wall boundaries by the method of assigning boundary information to cells and reproducing the object shape. This enabled the gear rotation and engagement that shape oil flows to be handled more readily, and also reduced the calculation cost.

This analytical technique was applied to a manual transmission, and the results show that the calculation of oil flows that takes approximately 26 days by the conventional technique was shortened to approximately four days. In terms of accuracy, it was further confirmed that the technique also enabled prediction of the state of lubrication in an actual transmission.

1. Introduction

The purpose of the transmission is to convert the power output from the engine to an optimal driving force and speed and transmit it to the driveshaft. The gears, bearings, and other component parts that form the route for transmitting that power are subject to damage or deterioration from wear and heat generation. In order to limit these factors, a continuous supply of oil needs to be unfailingly delivered to every part while it is in operation. On the other hand, when the amount of oil is increased to enhance the lubrication, the resistance torque increases because of oil agitation and shear. This resistance torque becomes a direct power loss and affects vehicle fuel efficiency. For this reason, transmission design seeks to enhance the housing in order to enhance oil flows, and baffles are placed in the area around each gear as ways to achieve a balance between lubricating ability and loss reduction. However, the techniques for confirming the effects of such measures at the design stage are not adequate, and as things actually stand, the process depends on such things as the experience of the people in charge of development and confirmation performed after first having fabricated prototypes for endurance testing, tests using flow visibility, and so on. With visibility testing, as well, it is often the case that interior flows cannot be adequately confirmed because they are hidden by parts or by oil adhering to the inside wall of the housing.

Simulation is used today for a variety of reasons, including the shortening of development time, reduction in

prototype expenses, and grasping phenomena that are not readily analyzed through testing on actual machinery. This is also the case with oil flows inside the transmission, for which analysis of flow phenomena using Computational Fluid Dynamics (CFD) is gradually becoming the norm. CFD includes a variety of different methods, and the methods are selected according to the intended use. Many issues still remain, however, with regard to analysis for the purpose of predicting lubrication performance and loss, which are mutually exclusive in performance, throughout the interior of the transmission, and methods for the purpose have not yet been established.

In the interior of the transmission, oil is agitated by multiple gears and other such rotating bodies. Unsteady and more complex flows are also formed by oil and air mixing together, and the calculation cost becomes immense. There are also minute droplets and sudden pressure fluctuations where gears are engaged, and it is no simple matter to apprehend the behavior of fluid surfaces.

This article reports on the creation of a technique for predicting the state of lubrication in terms of oil flows of these kinds inside a transmission within a practical length of calculating time.

2. Research Aims

Prediction of oil flows in the interior of the transmission calls for the ability to reproduce both the movement of bodies such as rotating gears and the behavior of fluids

* Automobile R&D Center

induced by those movements. Representative methods of CFD include the finite volume method and the finite element method. In these methods, the region that is calculated is converted to meshes that are treated as minute fluid elements and the behavior of the fluid is calculated. When this is done, the usual approach is to generate the meshes to follow object shapes. Since the calculation applies to both gas and liquid, this method can perform calculation that takes into account the air entrainment that has such a large influence on oil flows at high rotational speeds. In a transmission, however, this approach needs to handle fluid regions such as engagement of gears that change form from moment to moment while the calculation is underway, and this therefore calls for a special approach that will be described below. The dynamic mesh model⁽¹⁾ automatically regenerates meshes in conjunction with the rotation of a body, while the sliding mesh model⁽²⁾ simulates gear movements by taking the region around a gear centered on the axis of rotation, dividing it into cylindrical domains, and rotating those cylindrical areas. In the dynamic mesh model, however, it is a virtually insuperable challenge to continue maintaining mesh quality during the process of calculation, and the calculation can readily become unstable. In the sliding mesh model, the rotating meshes need to be kept from interfering one with the other, so this approach cannot take the engagement of gears into consideration. Also, the dynamic mesh model and sliding mesh model both need enormous amounts of time for generating meshes and for calculations when they handle large numbers of parts.

At the same time, however, attention is also being given to the particle method of analysis⁽³⁾, which converts oil and other fluids into particles for calculation. Compared to methods that use meshes, this has the advantage of higher calculating speeds because it calculates only particulated fluids. However, it cannot readily take into account the effects of air entrainment and negative pressure on oil flows, so that, under present conditions, application of this method is limited to oil flows at low rotational speeds that are less affected by those factors.

Given these circumstances, the aim of this research was defined as the creation of a technique that can predict transmission lubrication performance and that satisfies the following conditions:

- (1) It can reproduce gear rotation, including engaged portions, and calculate the oil flows formed by those movements, thus predicting the state of lubrication in each part
- (2) It has a calculating time that is practically usable in development
- (3) It can take into consideration the effects on oil flows from air entrainment and negative pressure

3. Analysis Objects and Methods

3.1. Analysis Objects

In order to judge whether oil flows induced by rotating

bodies are accurately grasped, the object was defined as an automotive manual transmission that does not include hydraulic circuits (Fig. 1).

3.2. Calculation Method

This research was based on the finite difference method, which uses meshes in the calculation domains, in order to enable consideration of the effects on oil flows from air entrainment and negative pressure. In order to address the issue of how to handle gear engagement, it applied the approach not of meshes generated according to shapes, but rather of incorporating information on gear shape and status of movement in cells (individual elements delimited by meshes) (Fig. 2)⁽⁴⁾. The rotation of a gear is reproduced not by changing the shapes of cells but by providing that information again. This approach is able to reproduce the engagement of gears without diminishing mesh quality, unlike the dynamic mesh model. This approach is able to reproduce the engagement of gears without diminishing mesh quality, as in the dynamic mesh model. In addition,

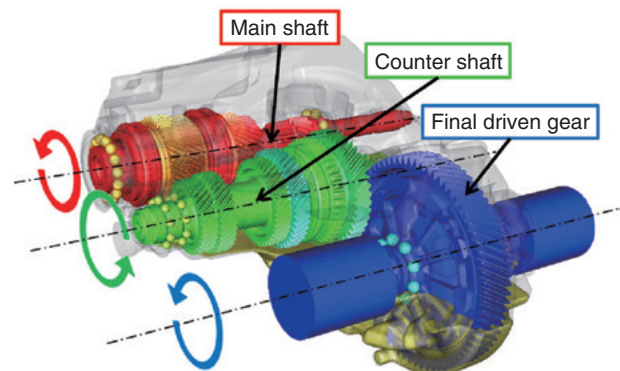


Fig. 1 Mechanical components of automotive manual transmission

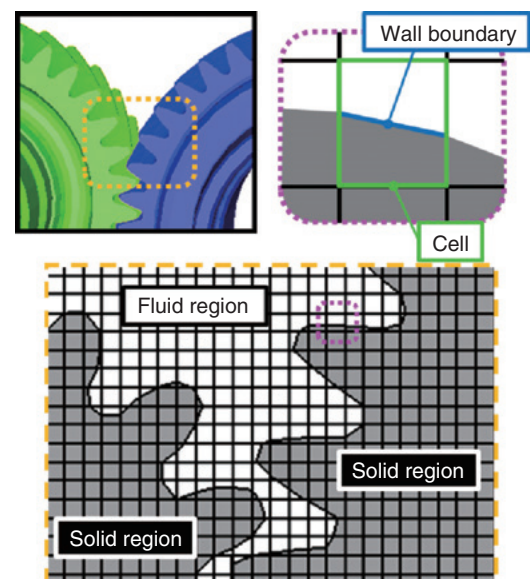


Fig. 2 Wall boundary treatment

this approach can shorten the calculating time as well as the time taken to prepare for analysis because it does not need to generate meshes according to shapes.

The behavior of fluid surfaces was calculated using the Volume of Fluid (VOF)⁽⁵⁾ method.

Large Eddy Simulation (LES)⁽⁶⁾ was used for the turbulence model because this approach is suited to accurately grasping highly nonstationary phenomena such as oil churning.

These methods are being applied as a way of achieving a balance between increased speed and accuracy of calculation.

4. Accuracy Validation

4.1. Validation Method

Compared to conventional methods in which the meshes are created to match the shapes, the reproducibility of object shapes is lower with this method. Consequently, the influence on the accuracy of calculations needs to be checked. The following steps were therefore taken to verify the accuracy of this method:

- Step 1. Accuracy of flow field formed by single-phase flows and moving wall boundary
- Step 2. Accuracy of flow path formed by multi-phase flows and moving wall boundary

4.2. Accuracy Verification of Flow Field Formed by Single-Phase Flows and Moving Wall Boundary

In order to confirm that a velocity field formed by rotating bodies within a fluid was analyzed accurately, actual measurements were made of flow velocity distribution using Particle Image Velocimetry (PIV)⁽⁷⁾,⁽⁸⁾ as an experiment in single-phase flows (Fig. 3). The measurement object was an oil flow by agitation created by a Rushton turbine (Fig. 4) in a cylindrical tank. The use of

stereo PIV, which employs two cameras, further enabled measurement of the vertical component of flow velocity with reference to a laser sheet surface. Measurements were taken under three conditions, at a rotational speed of 375 rpm, 525 rpm, and 750 rpm.

Figure 5 shows a comparison between the measurement

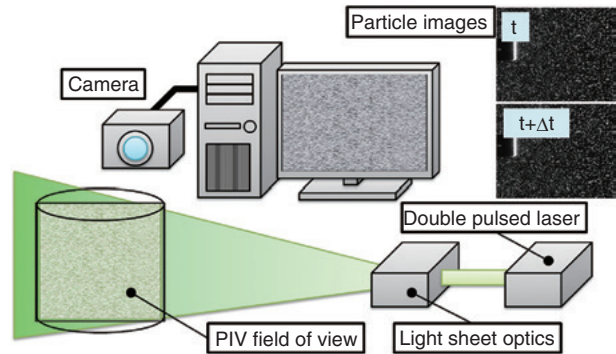


Fig. 3 PIV experiment system

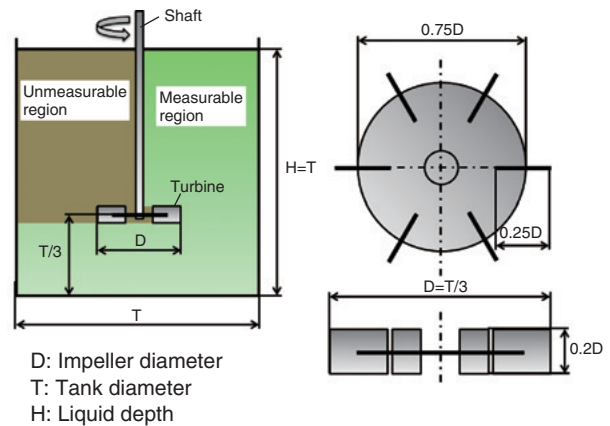


Fig. 4 Rushton turbine blade

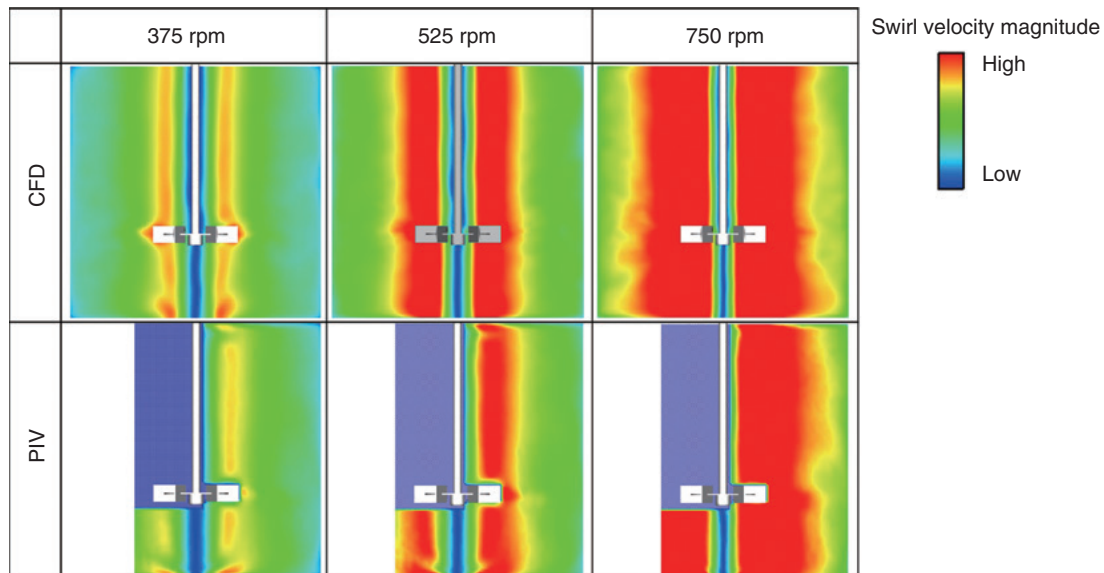


Fig. 5 Comparison of swirl velocity magnitude distribution between CFD and PIV experiment

results and CFD for the distribution of swirl velocity magnitude. The colors represent time averages of flow velocity, with red indicating higher velocity areas that shade into blue where the velocity is lower. Figure 6 shows a comparison of quantified velocities at different positions in a radial direction. It is apparent that changes in flow velocity due to differences in rotational speed as well as the decline in flow velocity at increasing distances from a rotating body in a radial direction are captured correctly in quantitative terms, as well.

4.3. Accuracy Validation of Flow Path Formed by Multi-Phase Flows and Moving Wall Boundary

Next, a gearbox composed of only one pair of gears as shown in Fig. 7 was used to conduct an agitation experiment, and this was taken as the object of comparison in order to confirm the reproducibility of gas-liquid flows using this method. The housing was made of transparent acrylic resin and the device was arranged to enable visual confirmation of the oil trajectories caused by rotation of the gears.

Figure 8 shows a comparison between the experiment results and CFD for trajectories. Rotational speed conditions of 200 rpm, 400 rpm, and 800 rpm were used. The reproducibility of oil trajectories resulting from different

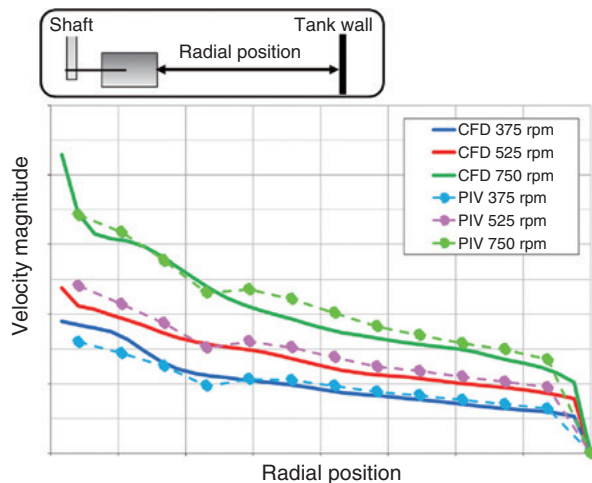


Fig. 6 Comparison of velocity magnitude distribution of radial position between CFD and PIV experiment

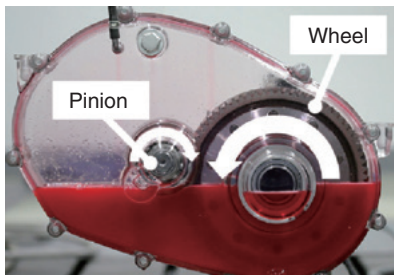


Fig. 7 PIV experiment system

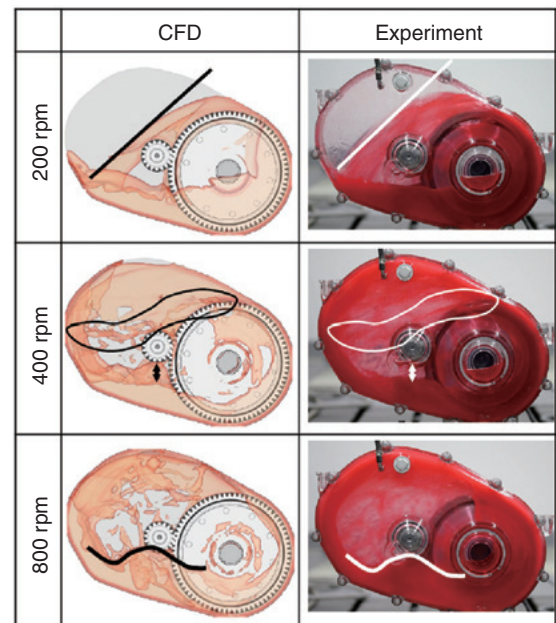


Fig. 8 Comparison of oil trajectory between CFD and experiment

rotational speeds was verified.

First, at 200 rpm, the trajectories of oil lifted by the wheel are in close agreement. There is also close agreement at 400 rpm, showing a portion of the oil that has been thrown off by the wheel and separated from the main flow of oil, and also showing it in contact with the pinion. There is agreement again at 800 rpm, showing an increase in the flow velocity of the component moving in the direction of the gear rotation, oil being pressed against the outer periphery, and the appearance of the fluid surface of the oil sump. The fluid surface that was not in contact with the pinion at 400 rpm has come into contact with it at 800 rpm, and the behavior of the fluid surface is in close agreement at all rotational speeds.

5. Application to Automotive Transmission

Oil flows inside an automotive manual transmission were analyzed using this method, and the flow characteristics were compared with those in imagery from visibility experiments using an acrylic case. Evaluation was performed under three conditions of gear rotational velocity, set at vehicle speeds of 15 km/h, 30 km/h, and 60 km/h.

Figures 9 and 10 show comparisons of the results of analysis using CFD and oil flows confirmed by visibility experiment at each vehicle speed. The element that has an important influence on flows inside the transmission is the gear. The final driven gear, in particular, has a large external diameter, is located at the lowest position, and has the greatest influence on oil flows. Attention was therefore focused on this area. The characteristics by vehicle speed were, first of all, at 15 km/h, that oil in the vicinity of the gear was lifted only as high as the red

dotted line in Fig. 11. When the speed rises to 30 km/h, it is apparent that the oil that is lifted as high as the top of the gear passes through the channels placed in the case interior walls and is supplied to the bearings at both ends that support the final driven gear. At 60 km/h, a larger amount of oil is transported by agitation, and there

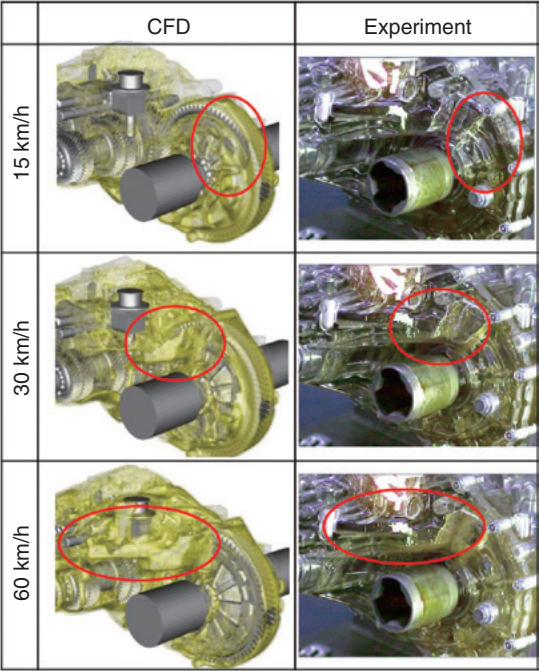


Fig. 9 Comparison of oil trajectory between CFD and experiment: rear view

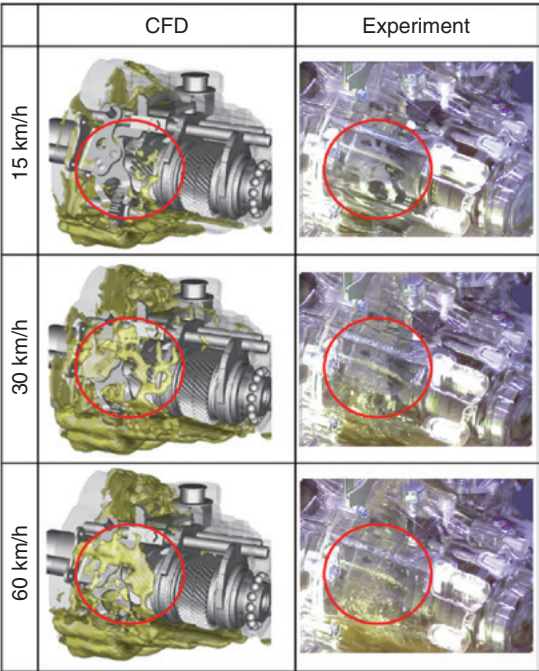


Fig. 10 Comparison of oil trajectory between CFD and experiment: front view

is an increase in the drops that are visible where they are thrown to the top of the housing and in oil adhering to the wall surface.

The characteristics of the flows described above are also similarly found in the analysis results. This method does capture the changes in fluidity that accompany changes in vehicle speed.

When the conditions of flow are confirmed from a different viewpoint, the experiment results and the analysis results both confirm that the amount of oil adhering to the wall surface increases as the vehicle speed increases.

It is apparent from the above results that this method accurately reproduces oil flow experiment results and that it will be effective in investigating oil flows at the design stage.

Taking the analysis at 30 km/h as an example for calculation time, the time taken by analysis to reach the quasistationary state at 3.0 sec was 3.57 days (performing calculations in parallel using 12 CPUs). This achieved a time approximately 1/7 that taken for an equivalent analysis performed using a conventional technique (Fig. 12). This is a practical calculation time for predicting lubrication performance at the design stage.

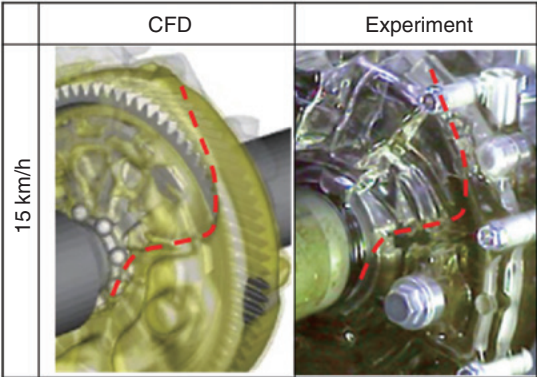


Fig. 11 Comparison of oil trajectory between CFD and experiment: detailed view at 15 km/h

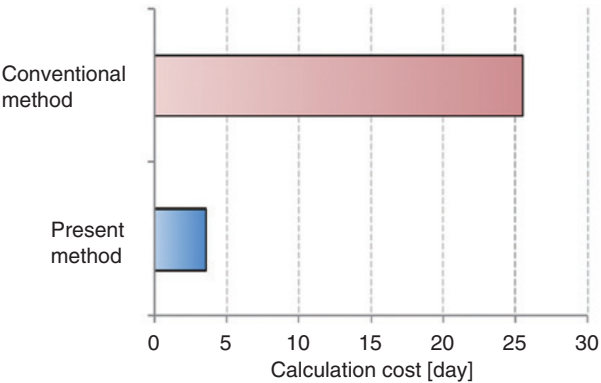


Fig. 12 Comparison of calculation cost between conventional method and present method

6. Conclusion

A technique was created for rapidly and accurately predicting oil flow within a transmission.

This technique is premised on the use of meshes in calculation domains to enable consideration of air entrainment and negative pressure, and it uses the finite difference method as a discretization method. For shapes of bodies and status of movement, it takes the approach of incorporating information in cells. This enabled the gear rotation and gear engagement portions that shape oil flows to be handled more readily and also reduced the calculation cost. In order to capture the behavior of fluid surfaces accurately, this approach uses the VOF method, while for its turbulence model it adopts LES. This enhanced its accuracy.

This technique was applied to oil flows inside an automotive manual transmission, and it was able to achieve prediction results for lubrication performance that have a close correlation with actual transmissions. The technique was also able to shorten calculation time to the point where results could be obtained in a practical length of calculation time at the design investigation stage.

References

- (1) Li, L., Versteeg, H., Hargrave, G., Potter, T., Halse, C.: Numerical Investigation on Fluid Flow of Gear Lubrication, SAE Int. J. Fuels Lubr., Vol. 1, Issue 1, p. 1056-1062, (2009)
- (2) Moshhammer, T., Mayr, F., Kargl, K., Honeger, C.: Simulation of Oil Flow in Gear Box Housing, SAE Technical Paper, 2006-01-1574, (2006)
- (3) Muto, K., Sakai, I., Ozaki, N.: Prediction of the Fluid Resistance in Stirred Fluid Using Particle Method, JSAE Annual Congress (Spring), 352-20095482, (2009)
- (4) Hirt, C. W., Sicilian, J. M.: A Porosity Technique for the Definition of Obstacles in Rectangular Cell Meshes, International Conference on Numerical Ship Hydrodynamics, 4th, p. 450-468, (1985)
- (5) Hirt, C. W., Nichols, B. D.: Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, Journal of Computational Physics, Vol. 39, p.201-225, (1981)
- (6) Derksen, J., Van den Akker, H. E. A.: Large Eddy Simulations on the Flow Driven by a Rushton Turbine, AIChE Journal, Vol. 45, No. 2, p.209-221, (1999)
- (7) Visualization Society of Japan: PIV Handbook, 328p, Morikita Publishing Co., Ltd. , (2002)
- (8) Nishino, K.: Fukuzatsu Ranryu no Zenba Keisoku - Ranryu Moderu Keisan tono Hikaku ni Mukete -, Saikin no Kagaku Kougaku 57 [Ryushi / Ryutai Kaiseiki, Suuchi Shimyureeshon no Tenbou to Jissen Jirei] Kagaku Kogakkai Hen, p.24-36, (2007) (in Japanese)

■ Author ■



Daiki SAEGUSA



Shinji KAWAI