Numerical investigations of submerged vortices in a model pump sump by using Large Eddy Simulation

Y Yamade¹,², C Kato², T Nagahara³ and Jun Matsui⁴

¹ Mizuho Information & Research Institute, Inc., Japan
² Institute of Industrial Science, the University of Tokyo, Japan
³ Industrial Products Company, Hitachi, Ltd., Japan
⁴ Faculty of Engineering, Yokohama National University, Japan

Abstract. Submerged vortices in a model pump sump and their flow structures were investigated numerically. The model pump sump is composed of a 2,500 mm-long water channel with rectangular cross section of 150 mm (channel width) by 100 mm (water height) and a vertical suction pipe with 100 mm diameter installed at its downstream end. At the upstream end of the channel, a uniform velocity of 0.37 m/s is given. In order to capture appearances and disappearances of submerged vortices in the pump sump, large eddy simulations (LES) are performed. The computational grids for the LES are composed of 2 billion hexahedral elements with 0.255 mm resolution. These grids can resolve the streamwise vortices in the approaching turbulent boundary layers that develops on the channel walls. However, it is not sufficiently fine to capture the vortex cores of the submerged vortices. The LES succeeded to capture appearances of the submerged vortices. By performing LES with several different sets of the wall boundary conditions, we have clearly identified, to the best of our knowledge for the first time, the origin of the submerged vortices. Computations that used a simplified computational model, where the computational domain was localized to the region close to the vortex core, were also performed to predict correctly the vortex core and to investigate dynamics of the vortices. The grid resolution in the simplified computational model was 0.03 mm. We successfully computed the size of vortex core in the simplified computational model. For this model, we also investigated the conditions under which a vortex appears by changing inlet tangential velocity.

1. Introduction

Intake vortices which appear in pump sumps will cause noise, vibrations and damages of pump system. Therefore conditions of intake vortices occurrence has to be taken into account in early stage of pump sump designs. Intake vortices in pump sumps can be evaluated by model sump measurements and/or computational fluid dynamics (CFD). By the developments of recent computer, importance of CFD has been increased. Reynolds averaged Navier-Stokes equations (RANS), which is time average base, has been mainly adapted for predictions of intake vortices in pump sumps [1-4]. In computations by RANS, typically steady solution is computed and it is judged whether intake vortices appear or not, by visualizing stream lines, pressure, vorticity and etc. of the computed steady flow. Ansar et al measured velocity fields in a test pump sump by laser Doppler velocimeter (LDV) [1] and showed a potential flow analysis could predict flow patterns in the test pump sump by comparing measure and computed velocity fields [2]. Constantinescu et al investigated accuracy for predicting flow fields in a test pump sump by several RANS models based on two equations [3]. Okamoto et al performed benchmark tests.
by experiments and CFD. They investigated conditions of intake vortices occurrences by changing inflow velocity and water level and evaluated capability of CFD based on RANS to predict the intake vortices occurrences. A time average base method can be useful tool for evaluating a possibility of intake vortices occurrences, since it can compute relatively larger scale flow structures depending on geometries of pump sumps. However it is not possible to accurately predict intake vortices and to compute unsteady phenomena such appearances and disappearances of intake vortices by a time average base method.

The objectives of this study are to understand mechanism of appearances and disappearances of intake vortices and to make use of knowledge in pump sump designs. In our previous study [5], we performed Large Eddy Simulation (LES) of flow in a test pump sump, successfully captured appearances of the intake vortices and discussed the origin of the intake vortices. In this paper, we will focus on submerged vortices which have their root on the bottom wall in a pump sump. In the subsection of 4.1 and 4.2 in this paper, we will briefly explain the origin of the submerged vortices reported in our previous study [5]. Then we will show the details of flow pattern and a process how a submerged vortex appears.

2. Test Pump Sump

The model pump sump in this study is the same one that Okamoto et al examined [4]. Table 1 shows the specifications of the model pump and operations conditions. We selected operation conditions with inlet velocity of 0.37 m/s and water level of 150 mm, which is corresponding to the condition in which both of submerged vortex and air-entrained vortex will take place.

| Table 1 Specifications of the test pump sump |
|---------------------------------------------|
| Diameter of the inlet pipe | 100 mm                        |
| Height of the vermouth        | 100 mm                        |
| Width of the sump             | 200 mm                        |
| Water level                   | 150 mm                        |
| Inlet velocity                | 0.37 m/s                      |

3. Numerical method

We performed LES of the incompressible fluid flow in the test pump sump with FrontFlow/blue (FFB) flow solver [6, 7]. The solver is an in-house general-purpose LES code based on a finite element method (FEM), which enables accurate predictions of turbulence by directly computing dynamics of streamwise vortices in the turbulent boundary layers. Dynamic Smagorinsky Model (DSM) [8, 9] is adopted as the subgrid-scale model of LES. A large-scale LES of various industrial flows, such as turbomachinery internal flows [5, 10, 11], automobile flows [12, 13], and hull boundary layers for ship hydrodynamics [14], can be performed with up to 100 billion grids because this code supports a function to automatically refine the computational grids at run time [15].

![Figure 1 Computational model of pump sump model.](image-url)
4. Submerged vortex in test pump sump

4.1. Computational model for flow in test pump sump
We performed LES of flow in the test pump sump with the condition shown in Table 1. Figure 1 shows the computational model. We set the x-axis in the streamwise direction from the vermouth of the pipe to the upstream boundary; y-axis in the spanwise direction and the z-axis in the vertical direction. The origin in the x-axis is the center of the intake pipe. The upstream boundary is located at X = 2,496 mm, and a constant velocity (0.37 m/s) without freestream turbulence is set there. The outlet of the intake pipe is located at Z = 2,000 mm, and the traction free and constant pressure (the atmospheric pressure) boundary conditions are set. For the remaining boundaries such as the bottom wall, side wall, surface of the vermouth and surface of the intake pipe, non-slip boundary condition is set. We set the small steps with a height of 5mm and a spacing of the 5 mm at 300 mm downstream of the inlet so that we have turbulent boundary layer (TBL) in the approaching boundary layer in the test pump sump. We used two sets of grids for LES. First one has the grid resolution of 0.225 mm in the horizontal direction, and resulting a number of grids of 2.05 billion. This grid resolves the turbulent boundary layer developed on the side and bottom wall. Second has 0.450 mm resolution and 0.25 billion grids. We compared the flow fields computed by these two sets of grids. We didn’t find any apparent differences in the important parameters for the submerged vortex such as vortex core size, pressure drop at the vortex core etc. [5]. We therefore determined to use the coarser grids (the second one) to save computational time and resource.

4.2. Origin of submerged vortex
Test computations of flow in the test pump sump with the different boundary layers developed on the bottom and side wall in order of investigate the origin of the submerged vortices. Three type of boundary conditions such as turbulent boundary layer, laminar boundary layer and no boundary layer developed for the approaching flow. Note that the small steps to make turbulent boundary layers were removed in the two cases of laminar boundary layer and no boundary layer. Figure 2 compares submerged vortices structures in these three cases. The vortices were visualized by the iso-surface of the Laplacian of the static pressure colored by the vertical vorticity. Note that a vortex with the clockwise (viewed from the top) vorticity is colored by blue and a vortex with counter clockwise is colored by red. There are no essential differences between two cases of the turbulent and laminar boundary layers. We found submerged vortices in these two cases. On the other hand, we could not find any visible submerged vortices in the case of no boundary layer. The computation were done for 2 sec which corresponds to 7.4 non-dimensional time based on the main stream velocity and streamwise length of the computational domain. We confirmed that any visible submerged vortices were not appeared all the time in the case of no boundary layer.

These test computations indicate that the origin of submerged vortices is the share of boundary layer of the approaching flow. Submerged vortices appeared in both cases in which approaching boundary layer is turbulent and laminar. It indicates the small streamwise vortices in turbulence boundary layer don’t have important roll about appearances of submerged vortices.

Relationships between submerged vortices and global share on the bottom wall were investigated. In order to focus relatively large scale of vortical motions, unsteady flow field were averaged with each 20 flow data, which corresponds to 80 msec. Figure 3 shows the vortices structures with the limited streamlines and color contour of the vorticity on the bottom wall. We selected typical two snap shots of flow fields. Namely, there is an only clockwise vortex (blue) in Figure 3 (a) and there are both of counter clockwise (red) and clockwise (blue) vortex in the Figure 3 (b). We found extremely strong correlation between submerged vortex and stream lines on the bottom wall. Clockwise (Counter clockwise) submerged vortex corresponds to the clockwise (counter clockwise) vorticity on the bottom wall.
4.3. Structure of submerged vortex

Profiles of velocity and pressure of a submerged vortex in the case with turbulent boundary layer were investigated to understand the flow structure of the submerged vortex. Flow fields were averaged in terms of time and space by moving with the submerged vortex since the submerged vortex was moving. The averaged profiles of velocity and pressure were computed as follows; at first we detected the center of the submerged vortex by the minimum static pressure in an instantaneous field, then we took circumferential average at each height as a function of the distance from the center, and finally we took time average by performing the same operation above for several sets of instantaneous flow fields. Unsteady flow fields were computed for 0.324 sec, which corresponds to 1.2 non-dimensional time based on the main stream velocity and streamwise length of the computational domain. A number of time steps for averaging was 10 thousands. Instantaneous flow fields were output at every 50 time steps. A resulting number of flow fields for averaging were 200. We confirmed that only one submerged vortex were remained under the intake pipe for the computation time above. The operations for averaging the vertical velocity $C_z$ can be written as follows;

$$C_z(r,z) = \frac{\sum_{i_a=1}^{N_a} \sum_{i_T=1}^{N_T} C_z(r,z)_{i_a,i_T}}{N_a N_T},$$

where $i_a$ and $N_a$ are respectively an index and a number of angle. We set $N_a$ to 360, namely flow fields were sampled every 1 degree angle for each height ($z$), radius ($r$) and time ($T$). $i_T$ and $N_T$ are respectively an index and a number of time. As mentioned above, we set $N_T$ to 200. The same operations for averaging were performed for the other parameters such as the tangential velocity $C_u$, radial velocity $C_r$ and static pressure $P$.

Figure 4 shows the averaged radial profiles of $C_z$, $C_u$, $C_r$ and $P$ at the heights of 6 mm, 30 mm and 90 mm. The feature of the submerged vortex under the intake pile was that it was accelerated in the vertical direction. Namely, the vertical velocity at the bottom is 0 and it almost linearly increased with the height $Z$. It is also important feature that there was high speed (large $C_Z$) region near the center ($r = 0$) of the vortex. Correspondingly, there is an inward flow (negative radial velocity $C_r$). About the tangential velocity $C_u$, the forced vortex with constant $C_u/r$ near the center and free vortex with constant $C_{u_f}$ far from the center. For the static pressure, the pressure difference from the outside point of $r = 54$ mm was plotted. The static pressure became minimum at center of the vortex. The pressure drop from the outside point is only 5 kPa, which corresponds to 5% of the atmospheric pressure. We
thought the pressure drop of 5% of the atmospheric pressure was too small. Submerged vortices appeared in measurement seem to include gas phase. The pressure at the vortex core therefore can be decreased to the vapour pressure and the pressure drop can be close to the atmospheric pressure. The possible reason of the underprediction of the pressure drop was the insufficient grid resolution. The vortex core computed by our LES was about 3~4 mm while the horizontal grid resolution was 0.225 mm. As mentioned before, the grid resolution in our LES was enough to resolve the turbulent boundary layer but it could be insufficient to represent sharp profiles of velocity and pressure near the vortex core.

![Figure 4](image1.png)

**Figure 4** Radial profiles of velocity components and pressure of a submerged vortex averaged in time and circumferential direction.

![Figure 5](image2.png)

**Figure 5** Computational domain of the simplified model.

5. Submerged vortex in simplified geometry

5.1. Computational model for flow in simplified geometry

We decided to perform flow computations of a submerged vortex with one order finer grid resolution than one shown in the previous section to capture sharp profiles of velocity and pressure near the vortex core. It was difficult to take one order finer grid resolution for a wide region in the test pump sump since a number of grid resolution would be 200 billion which was 100 times larger than one of the current grids. We therefore used simplified model. In the simplified model, we could have one order finer grid such as 0.03 mm by focusing the region only to small region around the center line. We used a paraboloid of revolution as the geometry of the simplified model to have constant velocity gradient of vertical velocity in the vertical direction to represent flow pattern shown in the subsection 4.3. The radius \( r \) of the computational domain can be written as a function of the height \( Z \) as following:

\[
r(z) = R_1 \sqrt{Z_1/\sqrt{z}},
\]

where \( R_1 \) is the radius of the computational domain at the height of \( Z_1 \). We set \( R_1 = 30 \) mm and \( Z_1 = 30 \) mm. The surface area \( S \) on the height \( Z \) is inversely proportional to the height \( Z \). The vertical velocity \( C_z \) therefore is proportional to the height \( Z \) and then the constant velocity gradient of the vertical
velocity $C_z$ in the vertical direction is realized. The computational domain of this simplified geometry is shown in Figure 5. We set the inlet boundary condition at the height $Z$ of 30 mm. We gave the constant velocity for vertical and radial components and Rankine vortex for the tangential component. Velocity components given at the inlet boundary condition can be written as follows;

$$
C_{z,\text{inlet}} = C_{z0}
$$

$$
C_{u,\text{inlet}} = \begin{cases} 
C_{u0} \frac{r}{R_0} & (r < R_0) \\
C_{u0} \frac{r_0}{r} & (r \geq R_0)
\end{cases}
$$

(3)

$$
C_{r,\text{inlet}} = 0,
$$

where $R_0$ is the radius at the interface of force vortex and free vortex and $C_{u0}$ is the tangential velocity at the radius $r = R_0$. We determined the parameters $(C_{z0}, C_{u0}, R_0) = (0.6 \text{ m/s}, 5.0 \text{ m/s}, 3.0 \text{ mm})$ so that the given boundary condition for velocity were consistent with the flow field computed by the LES of flow in the test pump sump.

For initial flow field, we set $C_z$ and $C_r$ to zero and the vortex formation shown in the equation (3) were given for $C_u$ in a whole region. Initial static pressure field were set so that the pressure gradient and the centrifugal force are balanced as shown in equation (4).

$$
P_{\text{initial}} = \begin{cases} 
\frac{1}{2} C_{u0}^2 \left( \frac{R}{R_0} \right)^2 & (r < R_0) \\
\frac{1}{2} C_{u0}^2 \left\{ 2 - \left( \frac{R_0}{R} \right)^2 \right\} & (r \geq R_0)
\end{cases}
$$

(4)

In this study, two sets of tests were performed with this simplified computational model. The first test was performed in order to investigate how the vortex structure will be changed with one order finer grid resolution compared with one for LES of flow in the test pump sump. We call the first test “base line case” in this paper. The second test was parameter study for $C_{u0}$. We intended to simulate the submerged vortex appearance by changing $C_{u0}$.

5.2. Base line case

We expected that we could analyse momentum balance by checking steady fluid field obtained in this simplified computational model. However, we found that the vortex had unsteady motions as the vortices in the test pump sump. Figure 6 shows the instantaneous distributions of the vertical velocity $C_z$ and the tangential velocity $C_u$ normalized by the inlet vertical velocity $C_{z0}$ on the cross section of the height $Z$ of 60 mm. The vortex core where the tangential velocity $C_u$ became 0 was not located at the center of the computational domain. There is an asymmetry in the tangential direction in both of $C_z$ and $C_u$. These indicate the vortex computed in this simplified computational model has unsteady motions.

Figure 6 Visualizations of instantaneous flow on the cross section of the height $Z = 60 \text{ mm}$ in the base line case.
Figure 7 compares the time averaged radial profiles of the tangential velocity $C_u$ and the static pressure in the two cases of test pump sump shown in the previous section and this baseline case. Profiles in the case of test pump sump are the same as those shown in Figure 4 (a) and (d). A log scale is used for the horizontal axis to make the difference of profiles near the vortex core ($r < 1$ mm) clear. The radius at which the tangential velocity $C_u$ became the maximum (we call this radius $R_{\text{max}}$ in this paper) was 3.0 mm in the test pump sump case, while it decreased to 1.5 mm in the baseline case. The maximum tangential velocity became 4.9 m/s which is 3.1 times larger than one in the test pump sump case. That led to increase of pressure drop to 41 kPa which was 8 times larger than one in the test pump sump case.

![Figure 7](image.png)

(a) $C_u$
(b) Pressure

**Figure 7** Comparisons of radial profiles of tangential velocity $C_u$ and static pressure between the pump sump case and the baseline case.

There were two findings from the baseline case. First, there was no steady solution when the boundary conditions which correspond to submerged vortex in the test pump sump were given. That indicates the unsteady motions of the submerged vortex have important roles for its momentum balance. Secondary, the pressure drop increased to 41 kPa which was about 40% of the atmospheric pressure and 8 times larger than one in the test pump sump case with the computational grid of 0.033 mm resolution.

### 5.3. Parameter study for simulating vortex appearance

As mentioned above, we found that the origin of submerged vortices was a global shear of boundary layer in pump sumps. Appearance of the submerged vortices can be understood as a result of concentration of global shear to the root of a submerged vortex on the bottom wall. In the baseline case, we investigated the flow fields with the condition after the submerged vortex appeared. In this section we performed the parameter study by changing the inlet tangential velocity $C_{u0}$. We started the test computations with small $C_{u0}$ (of 0.25 m/s which was 1/20 smaller than one of the baseline case), which corresponds to starting of concentration of global shear to the root of a submerged vortex. We intended to simulate a submerged vortex appearance by gradually increasing $C_{u0}$.

Figure 8 shows the relationship between the inlet tangential velocity $C_{u0}$ and pressure drop calculated as the static pressure difference between the vortex core and far point on the cross section of height $Z$ of 60 mm. Note that the left figure shows dimensional value, while the right shows non-dimensional value normalized by the dynamic pressure calculated by the inlet tangential velocity $C_{u0}$. The pressure drop became larger with the larger inlet tangential velocity $C_{u0}$. A sudden increase of the pressure drop was found around the point with $C_{u0}$ of 2 m/s. We found the pressure drop increased up to 50 kPa which corresponds 50% of the atmospheric pressure if we give $C_{u0} = 7.5$ m/s which is 1.5 times larger than one of the baseline case. The trend of increase of pressure drop was kept up to the point of $C_{u0} = 7.5$ if we evaluated the dimensional pressure drop as shown in Figure 8 (a), while non-dimensional pressure drop decreased with $C_{u0}$ larger than 2.5. That indicates that some of flow structures start changing at the point with $C_{u0} = 2.5$ m/s as explained later.
Figure 8 Comparisons of pressure drops on the cross section of Z=60 mm as the function of the tangential velocity $C_u$ given at the inlet boundary condition.

Figure 9 Distributions of instantaneous tangential velocity $C_u$ normalized by $C_{u0}$ on the cross section of the height Z=60 mm in the cases with $C_{u0}$ = 1.0, 1.75, 2.5 and 5.0 m/s.

Figure 10 A time series of static pressure at the center point on the cross section of the height Z = 60 mm in the base line case

Flow fields of 4 cases (including the base line case) with $C_{u0}$ = 1.00, 1.75, 2.50 and 5.00 (base line case) were selected for the detail analysis. Figure 9 compares the instantaneous tangential velocity $C_u$ normalized by the inlet tangential velocity $C_{u0}$ on the cross section of the height Z = 60 mm. The distribution of $C_u/C_{u0}$ with $C_{u0}$=1.0 is concentric, which indicate the vortex core remained at the center. On the other hand, there is an asymmetry in the tangential direction in the other three cases, which indicates the vortex had unsteady motions. Figure 10 compares time series of pressure fluctuations at the center point on the height Z of 60 mm. The horizontal axis is non-dimensional time normalized by the inlet velocity $C_{u0}$ (= 0.6 m/s) and the radius of computational domain $R_0$ (= 30 mm). RMS of the pressure fluctuation in the four cases ($C_{u0}$ = 1.00, 1.75, 2.50 and 5.00) were respectively, 0.0, 0.8, 3.7 and 5.7 kPa.
Figure 11 compares the radial profiles of the vertical velocity $C_z$ and tangential velocity $C_u$ on the height $Z$ of 60 mm. As shown in the Figure 9, there is a sudden increase of the pressure drop around $C_{u0} = 2.0$ m/s. Notable changes of velocity profiles were found near the point with $C_{u0} = 1.75$ m/s. At first, the vertical velocity $C_z$ at the center with $C_{u0} = 1.75$ m/s became 1.8 times larger than one with $C_{u0} = 1.00$ m/s. Note that the high vertical velocity localized to the vortex core is common feature of flow pattern in both of the test pump sump case and the base line case. Secondary, big changes occurred in terms of the tangential velocity $C_u$ between the cases with $C_{u0} = 1.75$ and 2.50 m/s. The $R_{max}$ decrease to 0.5 mm from 1.5 mm and the maximum $C_u$ increased to 3.3 m/s (2.1 times larger) from 0.8 m/s. Note that the ratio of maximum $C_u$ (= 2.1) is higher than the ratio of the inlet tangential velocity $C_{u0}$ (=1.4). The difference of velocity ratio leads to sudden increase of the pressure drop shown in Figure 9 (b). The sudden increase of $C_u$ can be understood as an extension of a vortex in the vertical direction due to the flow with the high vertical velocity near the center. It is also notable that $R_{max}$ again increased in the cases with $C_{u0} = 5.0$ m/s. This is consistent with that non-dimensional pressure drop was decreased from the case with $C_{u0} = 2.5$ m/s. The increase of $R_{max}$ and the corresponding decrease of non-dimensional pressure drop indicate that unsteady motion of a vortex have effect to suppress an extension of a vortex.

By the parameter study shown in this section, the process of a submerged vortex appearance can be understood as follows;

- Global shear starts to be concentrated to the root of a submerged vortex. In this moment, the vortex is very weak and it is not moving.
- If the tangential velocity exceeds the critical value the vortex starts moving and that leads to high vertical velocity localised near the vortex center.
- The critical value of the tangential velocity was about 2.0 m/s in this case. In general, we may be able to find the critical Reynolds number at which state of vortex changes base on the $R_{max}$ and $C_u$ at $r = R_{max}$. The Reynolds number of above definition was about 6,000 in this case.
- Finally, the high vertical velocity near the vortex core leads to an extension of the vortex. That leads to smaller vortex core, larger tangential velocity and larger pressure drop.

6. Conclusions

LES of inner flow of the test pump sump are performed in order to understand mechanism of submerged vortex appearances and we obtained the following conclusions.

- The origin of a submerged vortex is the shear of the approaching boundary layer. A submerged vortex can be assumed the result of involved and concentrated into the core of it. If there is no development on the bottom wall, submerged vortex loses its origin so is will be very weak. It is not important whether the approaching boundary layer is turbulence or laminar, therefore the
small vertical vortices in turbulent boundary layer does not affect appearance of submerged vortices.

- The grid resolution of 0.225 mm ~ 0.45 mm is enough for computing the dynamics of intake vortices, however insufficient for predicting its inner structure such as core size and pressure drop. It is confirmed that the grid resolution should be less than 0.03 mm, however it is possible that the core size will be smaller with the finer grid resolution. The pressure drop computed by the finer grids becomes 50% of atmospheric pressure at maximum.

- A process of a submerged vortex appearance can be understood as follow;
  - A global shear of the boundary layer starts to be concentrated to the root of a submerged vortex.
  - At the critical Reynolds number of about 6000, based on the vortex core and the maximum tangential velocity, the vortex will start moving. That leads to the high vertical velocity near the vortex core. This flow pattern is typical feature of submerged vortices.
  - Finally, high vertical velocity near the vortex core leads to an extension of the vortex in the vertical direction. That leads to smaller vortex core and increase of the tangential velocity and the larger pressure drop at the vortex core.

Acknowledgments
A portion of this research was supported by the grant for “Strategic Programs for Innovative Research” Field No. 4: Industrial Innovations from the Ministry of Education, Culture, Sports, Science, and Technology (MEXT)’s “Development and Use of Advanced, High-Performance, General-Purpose Supercomputers Project.” We express our thanks to all parties involved.

References
[1] Ansar M and Nakato T 2001 Journal of Hydraulic Engineering 127 No. 10 pp. 825-834
[2] Ansar M, Nakato T and Constantinescu G S 2002 Journal of Hydraulic Research 40, No. 4 pp 461-470
[3] Constantinescu G S and Patel V C 2000 Journal of Hydraulic Engineering 126 pp 387-391.
[4] Okamura T, Kamemoto K and Matsui J 2007 Proc. 9th AICFM (Jeju: AFMC) No. AICFM9-053
[5] Yamade Y, Kato C, Nagahara T and Matsui J 2015 Proc, AJK-2015 (Kore: ASME-JSME-KSEM), AJK2015-140753
[6] Kato C, Kaiho M and Manabe A 2003 Journal of Applied Mechanics 70, pp 32–43
[7] Kato C, Yamade Y, Wang H, Guo Y, Miyazawa M, Takaishi T and Takano, Y 2005 Computers & Fluids 36 No. 1 pp 53–68
[8] Germano M, Piomelli U, Moin P and Cabot W H 1991 Physics of Fluids A 3 No. 7 pp 1760-1765
[9] Lilly D K 1992 Physics of Fluids A, 4, No. 3, pp 633-635
[10] Guo Y, Kato C, Yamade Y, Ohta Y and Iwase T 2015 Proc. 13th AICFM (Tokyo: AFMC), No. AICFM13-137
[11] Hagiya I, Kato C, Yamade Y, Nagahara T and Fukaya M 2015 Proc, AJK-2015 (Kore: ASME-JSME-KSEM), AJK2015-33438
[12] Makihara T, Kitamura T, Yamashita T, Maeda K, Kato C, Takayama T, Yamamoto K, Yamade Y and Suzuki Y 2016 Proc. SAE Technical Paper (Detroit: SAE) 2016-01-1585
[13] Yamade Y, Kato C, Iida K, Iida A, Yoshimura S, Onda K, Hashizume Y and Guo Y 2016 Proc. SAE Technical Paper (Detroit: SAE) 2016-01-1617
[14] Nishikawa T, Yamade Y, Sakuma M and Kato C 2013 Proc. 16th Numerical Towing Tank Symp. (Munich )
[15] Yoshimura S, Tokunaga K, Sugimoto S, Okuda H, Suemitsu K, Kato C, Yamade Y and Yoshimura H 2013 Transactions of JSCES, Paper No. 20130012 (in Japanese)