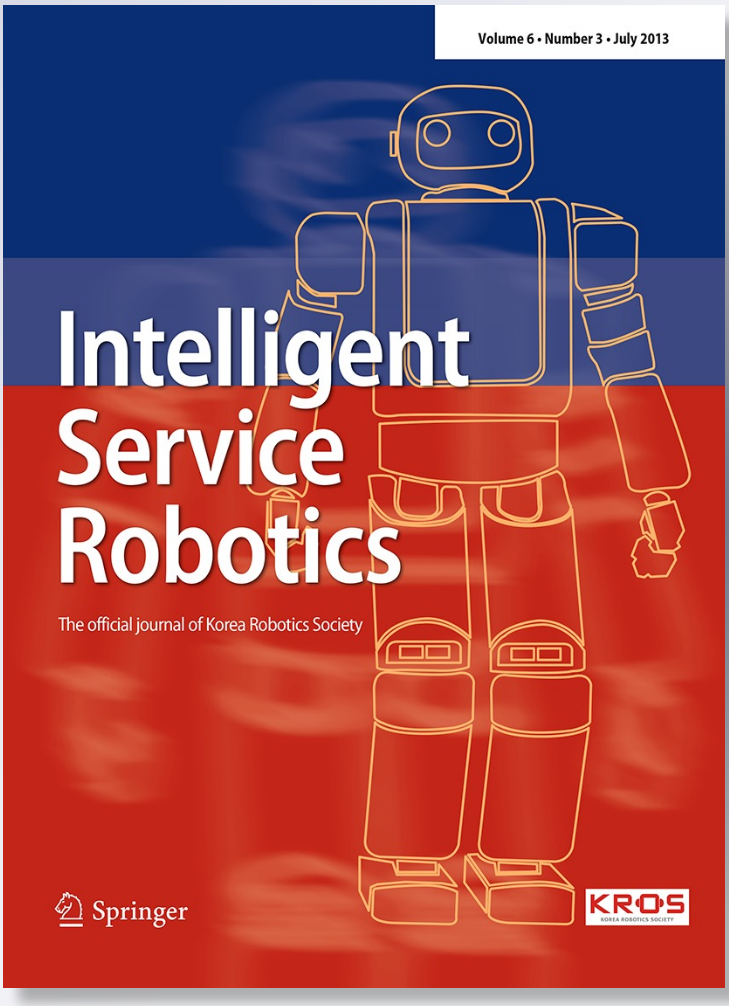
Analysis of underwater thruster model with ambient flow velocity using CFD

Han-Kyo Chung & Jinhyun Kim

Intelligent Service Robotics

ISSN 1861-2776
Volume 6
Number 3

Intel Serv Robotics (2013) 6:163-168
DOI 10.1007/s11370-013-0133-7



Springer

Intel Serv Robotics (2013) 6:163–168
DOI 10.1007/s11370-013-0133-7

ORIGINAL RESEARCH

Analysis of underwater thruster model with ambient flow velocity using CFD

Han-Kyo Chung · Jinhyun Kim

Received: 22 April 2013 / Accepted: 16 May 2013 / Published online: 16 July 2013

Abstract Thrust force is a very important factor for underwater vehicles. The thrust force that is determined by the pressure gradient between a propeller and a thruster can be represented by the ambient flow velocity introduced as the control volume and the axial flow velocity of a propeller. Because a change in ambient flow velocity triggers a change in the pressure gradient between a propeller and a thruster, a model taking account of the ambient flow velocity is required for an unmanned underwater vehicle (UUV) system. However, the axial flow velocity introduced into a propeller is very difficult to measure without accurate test devices. Therefore, in this study, the axial flow velocity is calculated with the computational fluid dynamics (CFD) method to use it as a basis for estimating the approximate value of the thrust force. As a result, a relatively accurate analysis of the effect of the ambient flow velocity on the thrust force can be obtained with considerable time and cost effectiveness as compared to the existing experimental methods. To evaluate the validity of the data from the CFD analysis results depending on the change in ambient flow velocity and the pressure gradient of a thruster, the resulting CFD values were compared with the thrust forces obtained in the previously performed thrust force experiment of a thruster depending on the ambient flow velocity in a circulating water channel.

Keywords Underwater vehicle · Thruster · Ambient flow velocity · CFD

1 Introduction

An unmanned underwater vehicle mainly performs exploration missions that are difficult for a man to carry out in an underwater environment. It can be employed for various missions such as the maintenance and repair of underwater structures and the detection and removal of naval mines. For the basic system control of an underwater vehicle, modeling of a thruster is required, and the thrust force is a very important factor for modeling a thruster. The thrust force of a thruster is determined by the pressure gradient between a propeller and a thruster such that the thrust force also changes when the ambient flow velocity changes. The thrust force is the main factor affecting the system. In previous studies, experimental methods were used for calculating the thrust force, for example, from experiments in a static fluid condition using simple models to complicated models where the ambient flow velocity was taken into account [1–7].

In general, in a propulsion experiment for underwater thrusters, a thruster is fixed at a frame in an experiment water tank, and devices such as strain gauges, load cells, and force torque sensors are installed to measure the two-dimensional force and the moment of a rotation at the same time, thereby obtaining the thrust force [4]. Such a method, without any other information but the thrust force, requires relatively simple thruster modeling; however, the method is prone to errors depending on the experimental environment. In another method, while the thrust force is measured by a device such as a load cell, the angular velocity information of the propeller is simultaneously obtained using a high-precision angular velocity sensor inside a thruster, along with

---

The authors acknowledge the financial support of the project 'Development of inspection equipment technology for harbor facilities' funded by Korea Ministry of Oceans and Fisheries.

H. K. Chung · J. Kim (✉)
Department of Mechanical Engineering,
Seoul National University of Science and Technology,
Seoul, Korea
e-mail: jinhyun@seoultech.ac.kr

H. K. Chung
e-mail: jhk1367@gmail.com

the flow velocity information in front of a thruster using a flow sensor, to improve the accuracy of thrust modeling. However, this method requires expensive high-precision devices and complex modeling [5].

As a method to overcome the drawbacks of the two experimental methods described above, a method for measuring the thrust force using a strain gauge while fixing a thruster within a circulating water channel was proposed [1]. This method can efficiently control the ambient flow velocity and requires no expensive high-precision sensors, while reducing many experimental errors relatively well as compared to other experimental methods. Although this method has the advantages of less complex modeling and increased accuracy, its major drawback is the considerable amount of time required for experiments.

In all previous studies, experimental methods were used for obtaining the thrust force, which is the most important factor in thruster modeling, and the ambient flow velocity and the propeller angular velocity were calculated as well to increase the accuracy. One of the problems associated with the application of the propeller angular velocity to the modeling was that the increase in accuracy increased the complexity of modeling. Nonetheless, most thruster modeling studies used the advance number J_0, which is a ratio of the water velocity introduced to a propeller to the product of the propellers diameter and the number of revolutions, as well as the propellers angular velocity to model a thrust because it is very difficult to measure the axial flow velocity experimentally.

In [1], a first-order linear equation consisting of the ambient fluid velocity and the propellers rotational velocity was proposed to simulate the axial velocity of a propeller. The result thus obtained was compared with the experimental results of the thrust force, with the results yielded within ±2 N of the error range. Therefore, it is very important to obtain the axial velocity when calculating the thrust force [1]. If the axial flow velocity of a propeller can be measured, simpler thrust modeling can be obtained and the thrust force can be calculated using simultaneously the equation of Bernoulli and the momentum equation in the control volume.

Therefore, in this study, to solve the two problems of complex thrust modeling and the required thrust force experiments associated with the consideration of the effect of the ambient flow velocity, the axial fluid velocity of a propeller was calculated using CFD, instead of using the complicated experimental methods used in previous studies. The axial fluid velocity of a propeller, which is difficult to measure experimentally, was defined as an approximate value of the axial fluid velocity estimated using CFD, thereby analyzing the effect of the ambient flow velocity on the thrust force of a thruster. Further, to validate the proposed CFD analysis result, the tendency of the thrust force according to the ambient flow velocity was compared with that obtained using a thrust force experiment conducted on a thruster in a circulating water channel under similar conditions. As a result, the tendency of the thrust force showed similarity within a certain range, and it was found that the thrust force in relation to the ambient flow velocity could be estimated within a certain range using only the thrust measurement experiments of a thruster under a static fluid condition. Through the CFD analysis of a thruster, the effect of ambient fluid velocity on the thrust force could be analyzed without complex experiments, and prior to the modeling stage of the thruster, the effect of the thrust force in relation to the ambient fluid velocity could be analyzed to enable its application when determining the required system specifications.

2 Thrust force of a thruster with respect to ambient fluid velocity

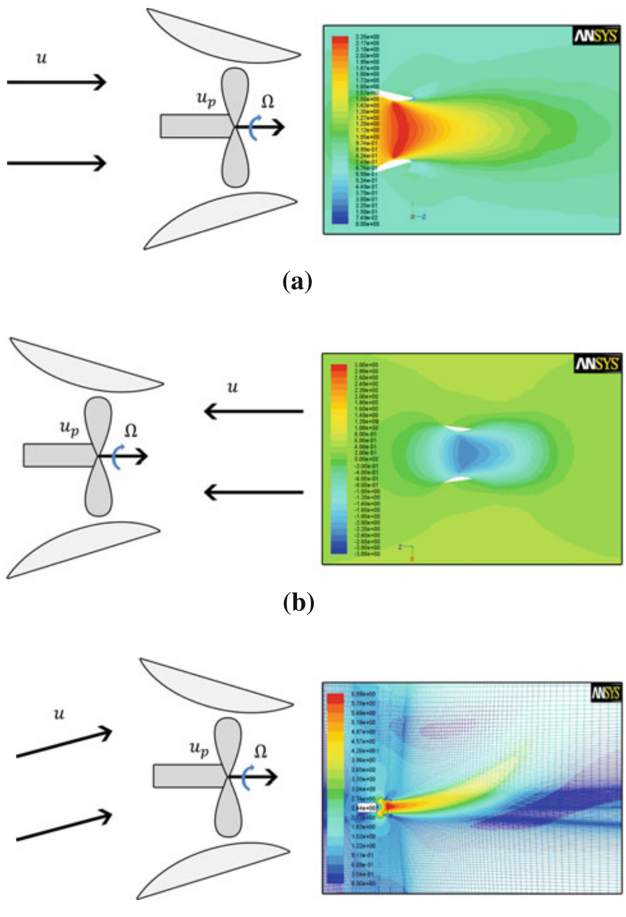
2.1 Definitions of conditions of ambient fluid

The thrust force differs depending on the velocity and direction of the fluid introduced into the thruster at the same rotational velocity state of a propeller. The fluid state around the thruster can be divided into an equi-directional state, anti-directional state, and vague-directional state [1].

The equi-directional state occurs when the ambient fluid direction and the axial fluid velocity direction due to the propeller rotation are the same as in the case of moving in a static condition fluid or moving against the fluid direction. In this state, when the ambient fluid velocity increases, the thrust force decreases because of the reduction in the pressure difference.

The anti-directional state is one in which the fluid flows in the opposite direction to the axial fluid velocity direction because of the propeller rotation. The streamline curve, which was produced by making the velocity vector direction tangential at each point of the moving fluid, as shown in Fig. 1b, was formed in the shape of a sink and a source. When the ambient fluid velocity increased, the thrust force also increased because of the increase in the pressure difference.

As for the vague-directional state, it is difficult to define a fluid state as the axial fluid direction is not clear because of the fact that the axial fluid cannot push the fluid introduced from an external source. The streamline is also difficult to predict, and it is impossible to apply the proposed thrust relation. In previous studies, anti-directional and vague-directional states were not taken into consideration. However, these states occur inevitably when unmanned underwater vehicles stop while moving, or move against the fluid direction.Hence, in this study, the CFD analysis was carried out for all three states, even though the condition volume could not be correctly defined in the vague-directional state.



(a)

(b)

(c)

Fig. 1 Three states of ambient velocity. **a** Equi-directional state, **b** anti-directional state and **c** vague directional state

2.2 Thrust force of a thruster

The thrust force of a propeller can be represented by u, the ambient fluid velocity; a disk having an inner area A_{p} where the propeller is located; and u_{p}, the axial fluid velocity of the surface of the disk. The fluid pressure that passes through the propeller exhibited a pressure jump phenomenon in the control volume, as shown in Fig. 2. Accordingly, if there is no variance during energy generation, the thrust force T occurs in the opposite direction to the fluid velocity direction [3].

The momentum relationship of the horizontal direction between one and two using the control volume can be represented by Eq. (1) of the thrust force, whereas Eq. (2) of the thrust force can be obtained if the control volume is defined as a region of the immediate front and rear of the disk.

$$T = \dot{m}(u_{\mathrm{out}} - u_{\mathrm{in}}) \tag{1}$$

$$T = A_{\mathrm{p}}(p_{\mathrm{b}} - p_{\mathrm{a}}) \tag{2}$$

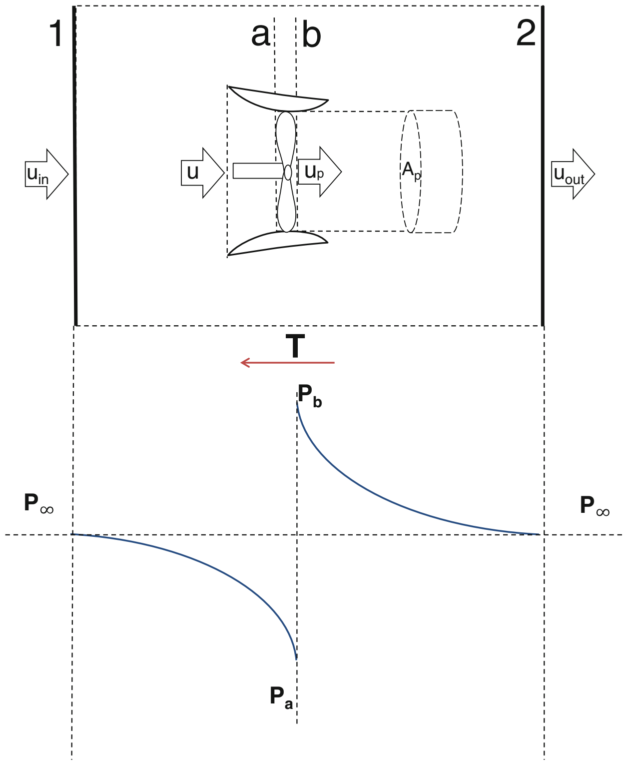


Fig. 2 Propeller race contraction: velocity and pressure changes

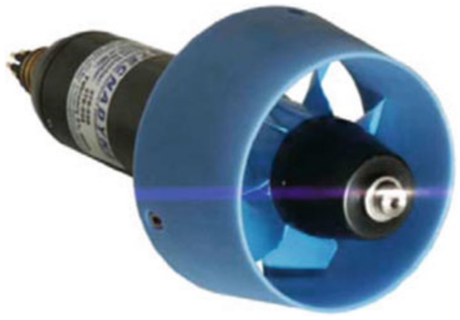


Fig. 3 Tecnadyne model 250 thruster

Assuming that water is an ideal fluid, the following equation can be summarized with respect to the water density ρ according to the incompressible Bernoulli principle.

$$p_{\infty} + \frac{1}{2}\rho {u_{\mathrm{in}}}^2 = p_{\mathrm{a}} + \frac{1}{2}\rho {u_{\mathrm{p}}}^2 \tag{3}$$

$$p_{\infty} + \frac{1}{2}\rho {u_{\mathrm{out}}}^2 = p_{\mathrm{b}} + \frac{1}{2}\rho {u_{\mathrm{p}}}^2 \tag{4}$$

The mass flow per unit time, which passes through a propeller cross section, is $\dot{m} = \rho A_{\mathrm{p}} u_{\mathrm{p}}$. Then, Eqs. (5) and (6) can be obtained by deriving equations using Eqs. (3) and (4) with respect to $p_{\mathrm{b}} - p_{\mathrm{a}}$.

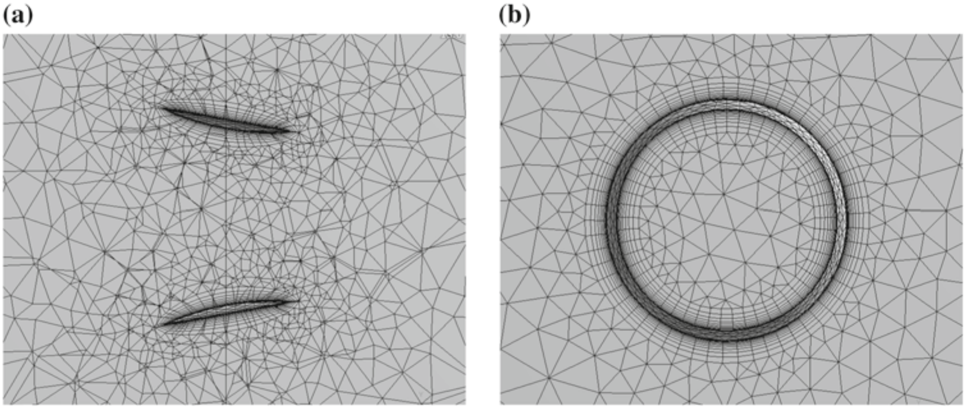


Fig. 4 Grid system of thruster model

$$p_\mathrm{b} - p_\mathrm{a} = \frac{1}{2}\rho({u_\mathrm{out}}^2 - {u_\mathrm{in}}^2) \tag{5}$$

$$u_\mathrm{p} = \frac{1}{2}(u_\mathrm{out} + u_\mathrm{in}) \Rightarrow u_\mathrm{out} = 2u_\mathrm{p} - u_\mathrm{in} \tag{6}$$

The details are as shown in [1].

3 CFD analysis

3.1 Analysis model and mesh generation

In this study, a 3D analysis model was built using the 400HFS-L Hi-Flow thruster model of crustcrawler, Inc. for the numerical analysis of a thruster and Tecnadyne model 250 thruster for compare to the experiment.

An analysis model for a thruster of the same size as that of the actual model in which a motor part was removed was produced. The mesh used for the analysis of the thruster was created using ANSYS Workbench 12.1. The mesh used was a mixed mesh consisting of a hexahedral mesh and a tetrahedral mesh. (Figs 3, 4)

3.2 Numerical analysis condition

The 3D model analysis was carried out using FLUENT of the commercial analysis program ANSYS 12.1. The working fluid was water, and in the case where the flow velocity inside the duct in the thruster was 1.8 m, the Reynolds number was expected to be approximately 89,000, which indicates turbulence. Therefore, as a turbulence model, a standard $k-\epsilon$ model was selected, which had good efficiency in terms of time and cost for a reasonable analysis at a boundary layer between the water and the thruster. The $k-\epsilon$ model analyzes turbulence by applying the turbulent kinetic energyrelated scalar value k and the dissipated energyrelated scalar value ϵ of the turbulent kinetic energy to the fluid transport equation. To analyze the flow over time, average values of the axial fluid velocity in the propellers cross section at 8, 10, and 12 s under an abnormal condition were calculated.

To simulate a propeller-like effect, a pressure jump condition of 500 to 4,000 Pa was applied to the propeller area, using the fan feature of FLUENT, as shown in Fig. 5. Under all conditions, the velocity condition of the fluid was used as the inlet condition and the pressure condition for the outlet condition. The detailed analysis conditions are as Table 1:

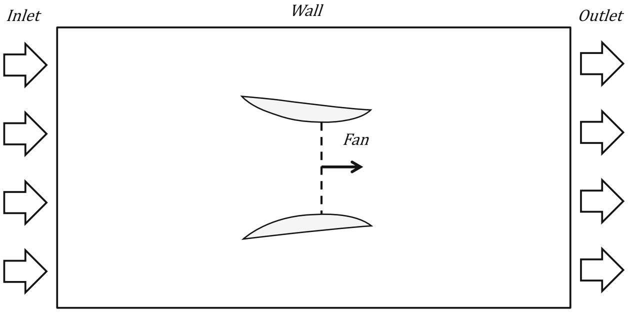


Fig. 5 CFD boundary condition

Table 1 Fluid mesh information

Description	Value/condition
Dimension	3D
Number of node	303,248
Material	Water–liquid
Diameter	0.076 m
Maximum velocity	1.8 m/s
Reynolds number	0 – 89,000
Viscous model	Standard $k-\epsilon$
Fan (pressure jump)	500 – 4,500 Pa

3.3 Analysis of the numerical analysis result

Figure 6 shows the thrust force of a thruster estimated using CFD when the ambient fluid velocity was −1.8, −1.6, −1.4, ±1.2, ±1.0, ±0.8, ±0.6, ±0.4, and 0 m/s. In the figure, the voltage level means the control input which is related to propeller rotational velocity [1]. In the CFD experiments, we use a pressure jump instead of the propeller rotational velocity.

To validate the analysis result of CFD, the experiment result of a thrust force Eq. (1) in a circulating water channel and the thrust force with respect to the ambient flow velocity were compared. Because the voltage applied to the thruster was simulated using a pressure jump in the fan, the absolute size of the thrust force showed a difference, whereas the thrust force with respect to the changes in the ambient fluid velocity showed a similar tendency. There was a range that showed large errors of thrust force values as compared to the experimental values. This was attributed to the increasing Reynolds number inside the duct of the thruster; the ambient flow velocity was increased, causing the Reynolds number to exceed the limitation of the commercial turbulence model. However, the use of more meshes for an accurate analysis and introduction of inexpensive turbulence models could lead to losing the advantages of time and cost effectiveness obtained using CFD, which can replace the complex thrust force experiments of a thruster. Therefore, it is important to define an effective area with respect to the ambient flow velocity, which can be analyzed using a commercial turbulence model.

$$T = 24.8{u_{\text{in}}}^2 - 5.1u_{\text{in}} + 0.1 \qquad (7)$$

Although the thrust force decreases rapidly once it has reached the maximum value in the negative region as the ambient fluid velocity moves further in the negative direction, the thrust force tends toward an increase once it has reached the minimum value in the positive region as the velocity moves further in the positive direction. This is attributed to the fact that the ambient flow velocity is more influential than the pressure gradient generated by the thruster when the absolute value of the ambient fluid velocity increases. Therefore, as the absolute value of the ambient flow velocity increases, turbulence increases, too, whereas the thrust force diverges to the negative and the positive directions. Using the least-squares method for the maximum thrust force of the ambient flow velocity in the negative range and the minimum thrust force of ambient flow velocity in the positive range, we can obtain Eq. (7) in a quadratic form. According to Eq. (7), the experimental and CFD values of the thrust force had an average error of 6.7 % and a maximum error of 14.4 %; thus, it was verified that the area governed by Eq. (7) can be set as the effective area (Fig. 7).

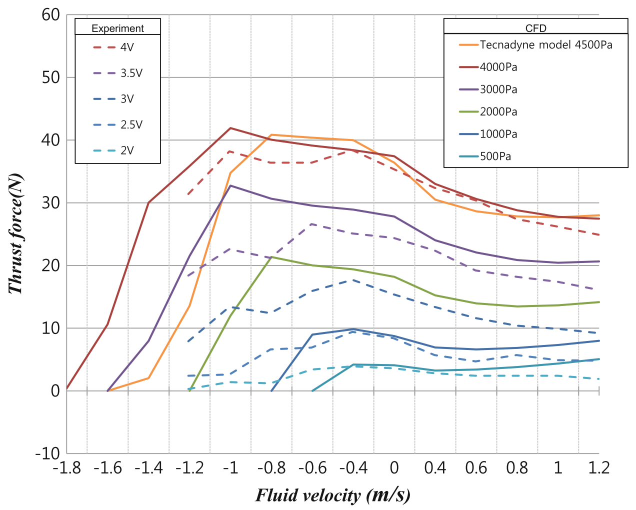


Fig. 6 Comparison results of thrust force: CFD and experiment

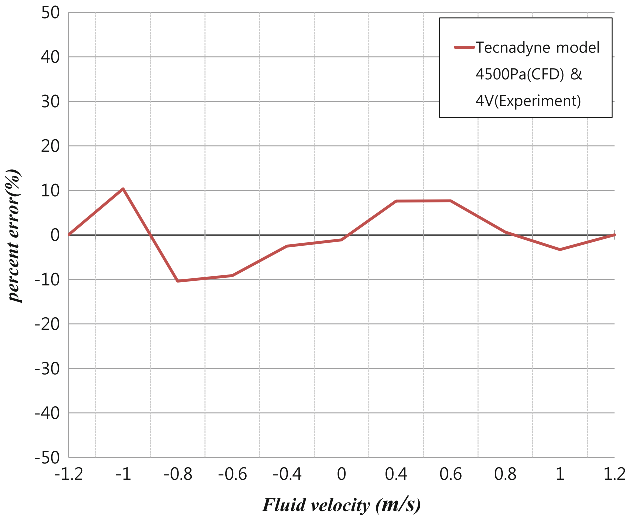


Fig. 7 Percentage error of thrust force CFD and experiment in effective area

4 Conclusion

In this study, we analyzed the effect of ambient fluid velocity on the thrust force using CFD instead of the thrust force experiment of a thruster, which has been generally performed to obtain the thrust force. The thrust force is considered to be an important factor for underwater vehicle systems. To validate the CFD analysis, the result was compared with that of the thrust force experiment result of the ambient flow velocity in a circulating water channel, and the comparison revealed a similar tendency within a specific range. The range could be relatively large depending on the turbulence model and meshes used. Although the standard $k - \epsilon$ model, which is a generally used turbulence model, was used, the effect of the ambient flow velocity on the thrust force of the thruster could be verified using CFD. Further, if the optimum pressure jump was derived through repetitive analyses based on the thrust force of the thruster under a static fluid condition, a single thrust force experiment under a static condition could obtain the thrust force according to the ambient flow velocity using CFD and optimum pressure jump of 4,500 Pa derived through repetitive analyses is applied to CFD analyses of tecnadyne model 250 as shown in Fig. 6. Under this situation, an error between the CFD and the experiment was only 5.29 % on average in the ambient flow in effective area.

In the future, we intend to study the effect of the incident angles of ambient fluid on the thrust force through a CFD analysis without any complicated experiments by investigating the vague-directional state that was not included in the CFD analysis of this study in light of the difficulty in defining its fluid state.

References

1. Kim Jinhyun, Chung Wan Kyun (2006) Accurate and practical thruster modeling for underwater vehicles. Ocean Eng 33:566–586
2. Lee Chong-Moo, Choi Hyun-Teak, Kim Kihun, Yeo Dong jin, Lee Pan-Mook (2011) The estimation of the thrust performance of a UUV using commercial thruster. J KSOE 24(6):27–31
3. Newman JN (1977) Marine hydrodynamics. MIT Press, Cambridge
4. Whitcomb LL, Yoerger DR (1999) Development, comparison, and preliminary experimental validation of nonlinear dynamic thruster models. IEEE J Oceanic Eng 24(4):481–494
5. Yoerger DR, Cooke JG, Slotine J-JE (1990) The influence of thruster dynamics on underwater vehicle behavior and their incorporation into control system design. IEEE J Oceanic Eng 15(3):167–178
6. Blanke M, Lindegaard K-P, Fossen TI (2000) Dynamic model for thrust generation of marine propellers. In: Proceedings of the IFAC conference maneuvering and control of marine craft (MCMC2000), pp 23–25
7. Bachmayer R, Whitcomb LL (2003) Adaptive parameter identification of an accurate nonlinear dynamical model for marine thrusters. J Dyn Sys Meas Control 125(3):491–494