Tạp chí Khoa học và Công nghệ 121 (2017) 095-100

# A CFD Study on Hydrodynamic Performances of a Propeller-Rudder System Used for the Cargo Ships

Nghiên cứu tính toán mô phỏng số CFD đặc tính thủy động lực học
hệ thống chân vịt – bánh lái sử dụng cho tàu hàng

***Le Thi Thai***
*Hanoi University of Science and Technology - No. 1, Dai Co Viet, Hai Ba Trung, Hanoi, Viet Nam*

*Received: November 03, 2016; accepted: September 9, 2017*

## Abstract

*In this paper, the hydro dynamic performances of the propeller – rudder interaction of the 5500DWT cargo ship was study by mean of CFD tool and comparing with theoretical calculation. The two cases of the propeller with and without a rudder in the propulsive system was investigated for prediction and comparison in hydrodynamic performances. The CFD results have shown the pressure field, flow velocity distributions around system in order to make prediction for hydrodynamic performance of the propeller and system. By controlling the driven attached angle of the rudder, the hydrodynamic coefficients are estimated for considering applying in operating the propulsive system of the 5500DWT cargo ship. The appearance of some low-pressure areas under the saturated vapor pressure on the suction side of the propeller at an advance coefficient J less than 0.35 that makes the suggestion that the cavitation model should be considered for further study.*

Keywords: Hydrodynamics, propeller, propulsive system, CFD, cargo ship.

## Tóm tắt

*Trong bài báo này, tác giả trình bày một số kết quả nghiên cứu về đặc tính thủy động lực học hệ thống chân vịt – bánh lái tàu thủy sử dụng cho loại tàu chở hàng khô 5500 DWT* thông qua sử dụng phương pháp tính toán lý thuyết kết hợp với mô phỏng số CFD*. Hai trường hợp khảo sát gồm: chân vịt độc lập và hệ chân vịt – bánh lái để dự báo và so sánh về mặt hiệu quả thủy động lực học. Kết quả mô phỏng số đưa ra trường áp suất và vận tốc dòng chảy bao hệ thống, phân tích dự báo đặc tính thủy động lực học chân vịt và hệ chân vịt – bánh lái. Khi thay đổi góc bánh lái, các hệ số thủy động lực học được xem xét nhằm áp dụng hiệu quả vào vận hành hệ thống đẩy của tàu hàng 5500 DWT. Việc xuất hiện các vùng áp suất thấp dưới ngưỡng áp suất hơi bão hòa ở mặt hút của chân vịt ở giá trị tốc độ tương đối J nhỏ hơn 0,35 cho thấy cần tiếp tục có những nghiên cứu với mô hình xâm thực tiếp theo.*

Từ khóa: Đặc tính thủy động lực học, chân vịt tàu thủy, thiết bị đẩy, CFD, tàu hàng khô.

## 1. Introduction

A study on improving the hydrodynamic performances of a system rudder-propeller is important for the maritime transportation. From now, it has too many authors presented a research to solve this problem. Some studies had presented new method to calculate hydrodynamic force of the rudder and propeller as shown in the papers [1, 2]. Some authors developed a new rudder profile as a fish type with a high lift force [3]. Some research reported on the optimum profile of rudder, effects of distance between rudder and propeller in a system rudder-propeller [4, 7, 10]. Other authors studied on effects of a Twisted rudders and propellers in a ship, [5, 6], target of the paper is to increase the knowledge on flow straightening influence of propeller and hull on effective angle of drift at the stern of a ship. The author used a modified Wageningen B4.40 propeller and rudder for CFD simulation and experience. The paper reported that comparison between CFD results and experimental results for both rudder and propeller can be predicted within 10% of measured data. In general the magnitude of drift angle depends on advance ratio. When rudders are placed behind a propeller, lift force increases with increasing propeller loading. Others papers reported on interaction between ducted propeller and rudder base on CFD and cavitation, [8].

In the field of study on hydro dynamic performances of the propeller ship has some important

---

* Corresponding author: Tel.: (+84) 904.048.555
Email: thai.lethi@hust.edu.vn

proplems which are now studied most on the world as follows:

- Improving hydro dynamic performances efficiency for a system propeller – rudder.
- Cavitation of a propeller, improving efficient thrust and torque for the system propeller – rudder.
- Strengthen, technique structured manufacture and material of propeller.

In previous paper [10, 11], the authors presented a study on effect of distant gap between rudder and propeller of the 5500 ton cargo ship in fully scale on hydrodynamic performances of the system rudder-propeller. In the paper, the effects of CFD results of the hydrodynamic performances of the propeller and rudder by the 2 simulating cases as well as rudder, propeller independence and whose in a system rudder-propeller was solved. The effects of the distance gap between rudder and propeller was solved too. The best distance gap between rudder and propeller was proposed for the system rudder-propeller of the 5500 ton in the research.

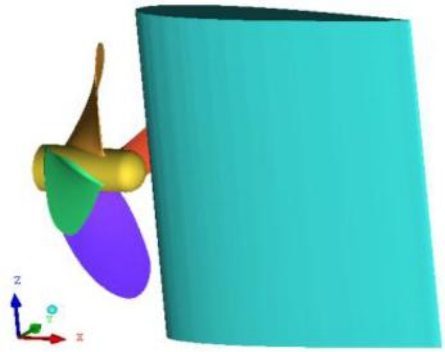


**Fig. 1**. 3D model of the rudder-propeller system of the 5500 DWT cargo ship

**Table 1.** Principal dimension of the system

| Name | Value | Unit |
|---|---|---|
| Propeller diameter, D | 3 | m |
| Propeller rate of rotation, n | 200 | rpm |
| Number of blades, z | 4 | - |
| Free stream velocity, $V_a$ | 15 | hl/h |
| Propeller advance coefficient, J | 0.54 | - |
| Rudder height, h | 3.8 | m |
| Rudder length, l | 3.0 | m |
| Rudder profile | Naca0018 | - |

In this paper, the author reports some important results of the study in the field of hydro dynamic performances of the propeller, with full scale of the propeller which uses for the 5500DWT cargo ship in Vietnan by using combined theoretical and CFD method. Figure 1 shows model propeller - rudder of the 5500DWT cargo ship used for computation.

## 2. Theoretical method and commercial CFD

In this study, the RANS solver in ANSYS – Fluent V.14.5 is used. The software license has been registered by the authors' School of Transportation and Engineering, Hanoi University of Science and Technology. The set-up procedures, such as: the designed models, the calculating fluid domain, meshing area and the boundary condition, have done following to the CFD's user guide which published by the International Towing Tank Conference (ITTC2008, 2011), [12-14]. The RANS solver is based on finite volume techinque method which use tetrahedron and prism grids. A suite of basic discretization schemes and solution algorithms are avaible. In this study, a second order upwind difference scheme and central difference scheme are selected to approximate convective terms and diffusive terms respectively. Turbulent viscous model k-ε is used. Under the assumption of incompressible Newtonian fluid, the flow around the propeller has to satisfy conservation equations of mass and moment which could be written in tensor notation as follows:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial (U_i)}{\partial t} + \frac{\partial (U_i U_j)}{\partial j} = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\nu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + \tau_i\right] \tag{2}$$

Where $x_i = (x, y, z)$ are independent coordinates, $U_i=(U, V, W)$ are Reynolds Averaged velocity components, $\rho$ is water density, $\nu$ is kinematic viscosity coefficient, $f_i$ are body force components and $\tau_{ij}$ is Reynolds's tensor resulting from the time averaged procedure to Navier – Stockes equation.

The turbulent viscous model is applied to approximate the eddy viscousity equation. The transport equation are defined as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{3}$$

For dissipation ε:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{4}$$

Where the modeling turbulent viscousity is defined as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

And the production of *k*, $G_k$, effect of buoyancy $G_b$ are defined as follows:

$$G_k = -\rho\, \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} \qquad (6)$$

$$G_b = \beta g_i \frac{\mu_t}{\mathrm{Pr}_t} \frac{\partial T}{\partial x_i} \qquad (7)$$

And, S is the modulus of the mean rate of strain tensor, defined as:

$$S \equiv \sqrt{2 S_{ij} S_{ij}} \qquad (8)$$

The coefficient of thermal expansion $\beta$ is defined as:

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p \qquad (9)$$

Others model constants are valuated as follows:

$$C_{1\varepsilon} = 1.44; C_{2\varepsilon} = 1{,}92; C_{\mu} = 0.09; \sigma_k = 1.0; \sigma_{\varepsilon} = 1.3$$

In Figure 2, the calculating fluid domain is limited in 18.5m of length, 12m of breadth and 12m of height for a 3m of propeller diameter, 3.8m of rudder height and 3m of rudder length. Meshing of the calculating fluid domain in T-gird generates in 2.2 millions meshes. For simulation, the turbulent viscous model k-ε is used. The velocity inlet is set for the inlet, the pressure outlet is set for the outlet. The rotation wall is used for this problem. Figures 2, 3 show the calculating fluid domain, meshing and boundary condition.

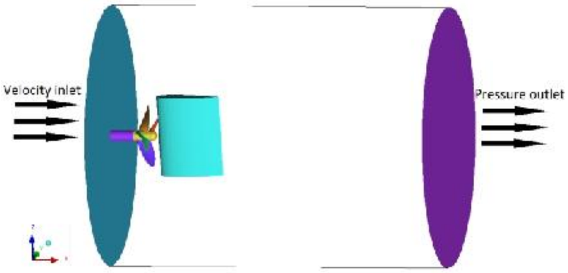


**Fig. 2.** Computing domain of fluid and boundary conditions set-up

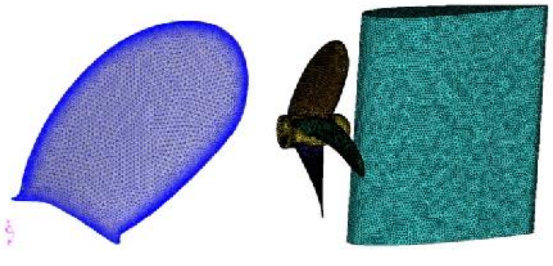


**Fig. 3.** Meshing over surface of the propeller and rudder with unstructure mesh, T-grids

## 3. Computed results of hydro dynamic performances of the propeller and rudder

In this section, the effects of computed results in the simulating problems, which are independent and in the system with a rudder, on hydro dynamic performances of the propeller are investigated.

At first, the thrust and torque coefficients of the propeller which is independent of the rudder are investigated by the CFD. Then the CFD results are compared with those of the calculated results given by the theory experimental equation. Follows as theory experimental equations the thrust ($K_T$) and torque ($K_Q$) coefficients are defined as follows [9]:

$$K_T = \frac{T}{\rho \quad {}^{2} D^{4}} \qquad (10)$$

$$K_Q = \frac{Q}{\rho \quad {}^{2} D^{5}} \qquad (11)$$

where: T is thrust of the propeller, N

Q is the moment of the propeller, Nm

ρ is density of the sea water, kg/m$^3$

The CFD result and the theoretical result calculated by equations (10), (11) present a good agreement as shown in the propeller coefficient curves of $N_p$, $K_T$ and $K_Q$ with J > 0.35 (Figure 4). This acceptable result is based for further computational analysis.

In flow visualization, Figures 5 and 6 show the pressure distribution and the velocity field arounding the propeller at the different rotating speed. At a high speed value n = 300 rpm (J = 0.36), the low pressure area at the blade edge of the suction side is under the saturated vapor pressure of water. So the cavitation problem should be considered more for this propeller, but in another paper. Further, this can explain why there is a difference between the theoretical calculation with the CFD's result when J is under around 0.35 in Fig. 4.

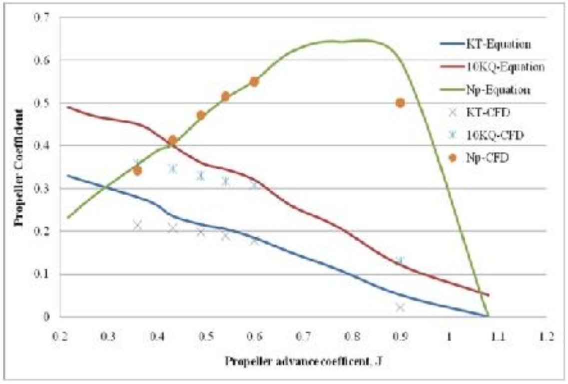
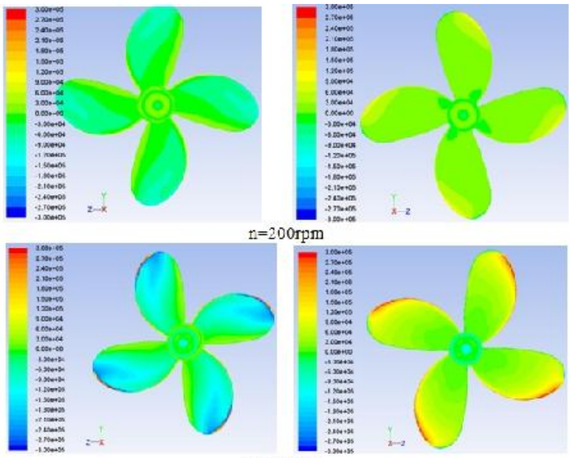
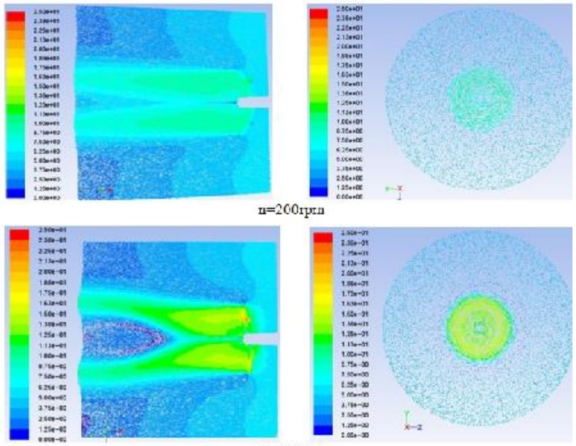


**Fig. 4.** Comparison of CFD's and calculating theory results for the propeller


n=200rpm

n=300rpm

**Fig. 5.** Pressure distribution over blades of propeller independent of rudder


n=200rpm

n=300rpm

**Fig. 6.** Velocity distribution around propeller in calculating fluid domain

Additionally, the hydrodynamic performances of the propeller is computed in the case with a rudder for estimating the effects of the driven attached angle and the propeller – rudder interaction. Figure 7 shows the the relation of the rudder forces, lift and drag, and the attached angle in the two calculations: rudder without propeller and rudder with propeller. The rotating flow induced from the propeller makes the unaxisymmetric curves. There is not significant effect of the attached angle when driving the rudder in the range of value from -5° to 5°. The calculation of CL and CD based on the pressure distribution around the rudder as shown in Figure 8.

Figures 9 and 10 show the pressure distributions around propeller and rudder of the system with rudder attacked angle 0 degree.

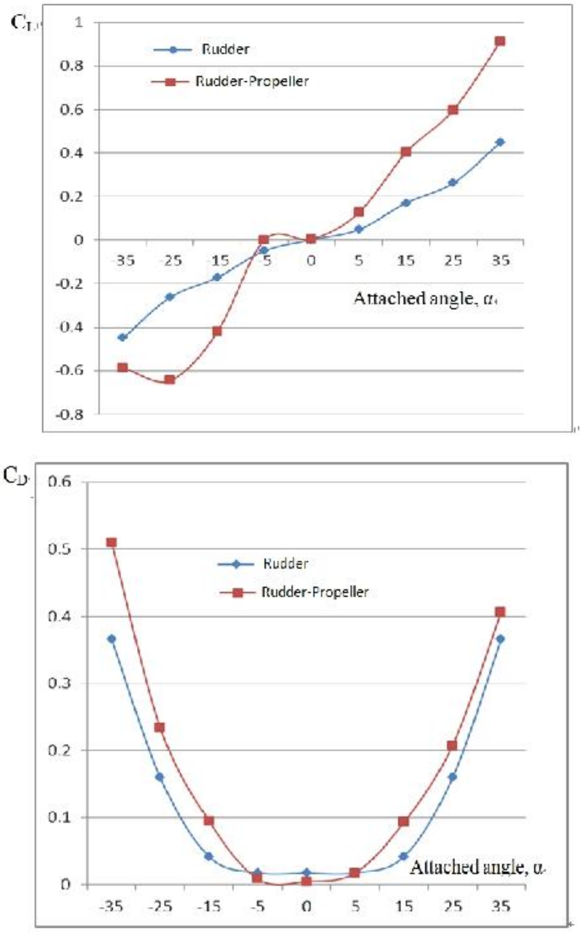
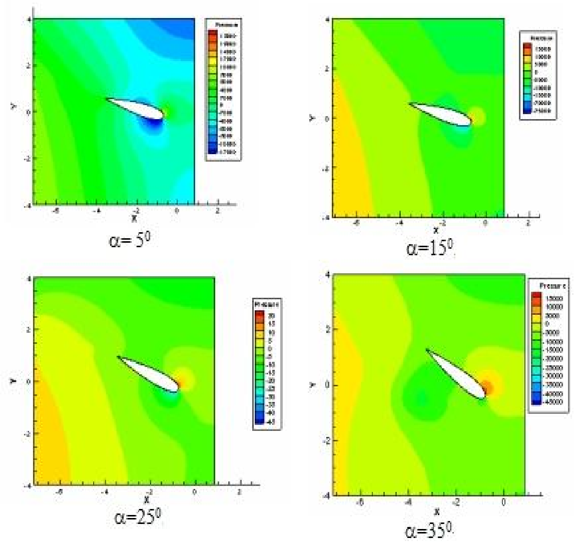


**Fig. 7.** The draft and lift in rudder - propeller system (n=200rpm, J = 0.54)


α= 5°

α=15°

α=25°

α=35°

**Fig. 8.** Pressure distribution around rudder and over surface of the rudder

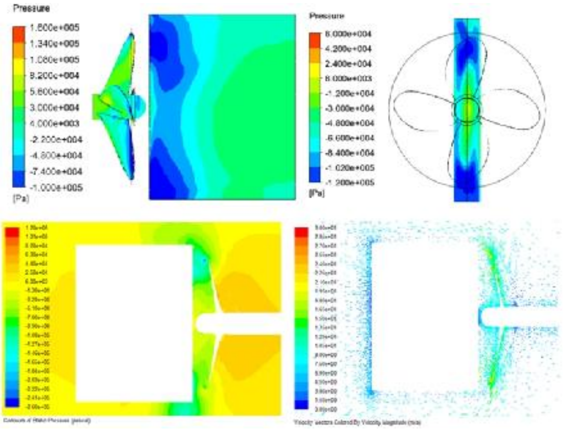


**Fig. 9.** Pressure and velocity distribution around rudder in the system, $D_0$

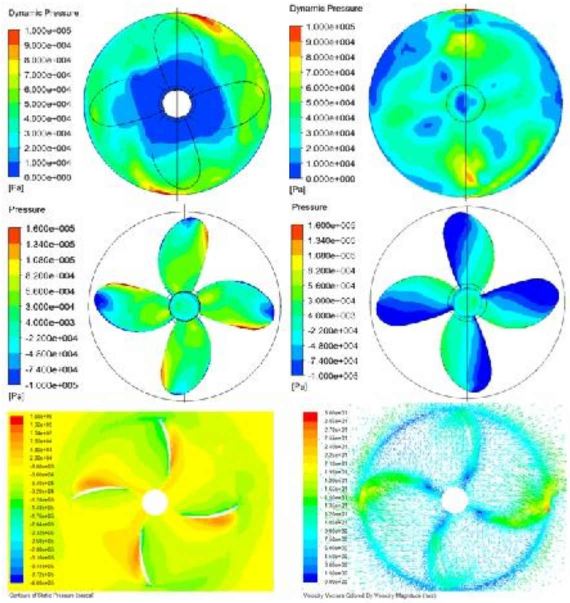


**Fig. 10.** Pressure and velocity distribution around propeller in the system, $D_0$

Figure 11 shows the comparison of the thurst, torque and propeller efficiency, $K_T$, $K_Q$, $\eta$ between the two cases of the propeller with and without rudder. The results shows clearly that the effects of the driven attached angle on thrust, torque and propeller efficiency. The hydrodynamic coefficients of the propeller are improved when the attached angle changing from 0° to 40°. The power coefficient is highest at the attached angle 10° but the thrust coefficient is highest at the attached angle 30°. Comparing to a single propeller, the propeller - rudder system can make power coefficiency higher around 20%.

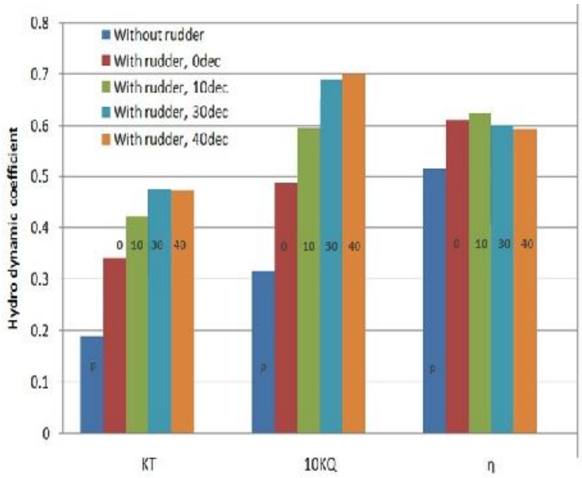


**Fig. 11.** Effects of rudder on hydro dynamic coefficient of the propeller

## 4. Conclusion

In this paper, the hydro dynamic performances of the propeller – rudder interaction of the 5500DWT cargo ship was study by mean of CFD tool and comparing with theoretical calculation. The two cases of the propeller with and without a rudder in the propulsive system was investigated for prediction and comparison in hydrodynamic performances.

The CFD result can used to make prediction for hydrodynamic performance of the propeller with $J > 0.35$. At the value of $J \leq 0.35$, the appearance of some low pressure areas under the saturated vapor pressure on the suction side of the propeller makes the suggestion that the cavitation should be considered and we have to make further study.

At the driven attached angle 0 -5°, the hydrodynamic coefficients are not change significantly. At the driven attached angle around 30°, the thrust is the best. By controlling the rudder, the power efficiency can be improved up to 20% (highest at around 10° - 30°).

The present result is useful and applicable in operating the propeller – rudder system of the 5500DWT cargo ship.

## References

[1] Tomasz. B, Jan, Szantyr. B, Numerical study of propeller scale effects, 15th Numerical Towing Tank Symposium, Cortona, Italy, 7-9 Oct (2012).

[2] Kruger. Ch, Kornev. N, Paschen. M, Semlow. Ch , Influence of roughened propeller tip surface on tip vortex structures, 15th Numerical Towing Tank Symposium, Cortona, Italy, 7-9 Oct (2012), pp. 140-145.

[3] Nguyen. V.T, Ikeda. Y, Hydrodynamic characteristic of rudder sections with high lift force, Proceeding of the 18th Japan Society of Naval Architects and Ocean Engineers, Sendai, Japan, (2013), pp. 171-174.

[4] Ahn. K, Choi. G, Son. D, Rhee. K.P, Hydrodynamic characteristics of X-Twisted rudder for large container carrier, Journal of Naval Architecture and Ocean Engineering, JSNAOE, (2012), pp. 322-334.

[5] Durante. D, Dubbioso. G, Broglia. R, The turning circle maneuver of a twin screw vessel with difference stern appendages configuration, 15th Numerical Towing Tank Symposium, 7-9 Oct, Cortona, Italy, (2012) , pp. 64-69.

[6] Badoe. Ch, Phillips. A, Turnock. S.R, Influence of drift angle on rudder-propeller interaction, 15th Numerical Towing Tank Symposium, 7-9 Oct, Cortona, Italy, (2012).

[7] Druckenbrod. M, Greitsch. L, Bednarek. S, Geometric modeling for optimization of propeller hub caps, 15th Numerical Towing Tank Symposium, Cortona, Italy7-9 Oct, 2012, pp. 58-63.

[8] Xie. Y, Wang. G, Wang. W, Research on hydrodynamic performance of the interaction between ducted propeller and rudder base on CFD, International Journal of Ocean System Engineering, (2013), pp. 169-174.

[9] Thai. L.T, A study on cavitation of the shapes moving with high speed, Doctor thesis at Hanoi University of Science and Technology, 2013.

[10] Ngo. V.H, Le. T.T, Le. Q, Ikeda. Y, A Study on interaction effects on hydrodynamic performance of a system rudder-propeller by distant gap", Proceeding of the 12th International Marine Design Conference, Tokyo, Japan, (2015)pp.179-193.

[11] Le. T. T, Ngo. V.H, Ikeda. Y, A study on improving hydrodynamic performances of a system rudder and propeller by attaching a fix plate on the rudder, The 8th Asia-Pacific Workshop on Marine Hydrodynamics - APHydro 2016, Hanoi, Vietnam, pp. 277-284.

[12] ITTC, The proc. of the 25th International Towing Tank Conference, Fukuoka, Japan, (2008), Website:

[13] http://ittc.sname.org/proc25/assets/documents/VolumeI/Proceedings-Vol-01.pdf.

[14] ITTC, The proc. of the 26th International Towing Tank Conference, Rio de Janeiro, Brazil, (2011), Website: http://ittc.sname.org/proc26/assets/documents/VolumeI/Proceedings-Vol-01.pdf.