TẠP CHÍ KHOA HỌC CÔNG NGHỆ HÀNG HẢI ISSN: 1859-316X
JOURNAL OF MARINE SCIENCE AND TECHNOLOGY

NUMERICAL PREDICTION OF BOW THRUSTER PERFORMANCE USING RANSE METHOD

NGHIÊN CỨU TÍNH TOÁN CÁC THÔNG SỐ THỦY ĐỘNG CỦA CHONG CHÓNG MŨI BẰNG PHƯƠNG PHÁP RANSE

BUI THANH DANH
Shipbuilding Faculty, Vietnam Maritime University
Email: danhbt.dt@vimaru.edu.vn

Abstract
The paper presents the numerical simulation results of hydrodynamic characteristics of bow thruster (thrust coefficient K_T, torque coefficient K_Q) in different pitch angles using RANSE method. Sliding mesh method is used to simulate a rotating propeller. The grid independence study on the simulation results is investigated. To ensure that the calculation findings are reliable, the numerical results are compared to the experiment results. Besides, the flow fields are performed to study the effect of flow around the bow thruster and the hull. It is explained for the additional thrust to the hull in the results.

Keywords: *Bow thruster, hydrodynamic characteristics, numerical simulation, RANSE.*

Tóm tắt
Bài báo trình bày kết quả tính toán các đặc tính thủy động của chong chóng mũi (hệ số lực đẩy K_T, hệ số mô men K_Q) ở các góc bước khác nhau sử dụng phương pháp RANSE. Phương pháp lưới trượt được sử dụng để mô phỏng sự chuyển động của chong chóng. Ảnh hưởng của việc chia lưới đến kết quả mô phỏng được tính đến trong nghiên cứu. Ngoài ra, bài báo đưa ra các hình ảnh về đường dòng bao quanh chong chóng mũi và thân tàu ở các góc bước khác nhau. Điều này giải thích cho việc xuất hiện thêm thành phần lực đẩy tác động lên thân tàu trong kết quả tính toán.

Từ khóa: *Chong chóng mũi, các thông số thủy động, mô phỏng số, RANSE.*

1. Introduction

As the demand for transportation grows, so does the number of larger inland vessels navigating the waterways. The propeller powers must also grow to keep the vessels maneuverable when their draught and deadweight increase. Bow and stern thrusters are placed aboard ships to eliminate the need for tug assistance during berthing procedures. Compared with the traditional propeller rudder system, the bow thruster has better maneuverability, only at low speeds.

The original numerical methods used to evaluate the hydrodynamic performance of propellers are lifting surface theory and the surface panel method. Nowadays, with the rapid development of computing resources, compared with experimental methods, the calculation results are relatively accurate, the cost is lower, and the calculation time is short. Therefore, Computational Fluid Dynamics (CFD) methods can be used to solve many ship hydrodynamic problems. Today, the numerical technique for solving the Reynolds-averaged Navier-Stokes (RANS) equation is extensively used and provides more precise results and flow details [1].

The majority of early bow thruster research was based on model tests. Feng Yukun [1] and his colleges evaluate the influences of the hull, the opening fairing, and the multiple thrusters at different inlet flow angles via experimental and numerical methods, u-RANS method with STAR-CCM+. The SST k-ω turbulence model with unstructured trim mesh and sliding mesh method was applied in this paper. The errors of K_T and K_Q compared to the experiment data were 9.51% and 3.13% respectively. Nakisa and his colleges [2] studied the effect of different turbulence model on hydrodynamics characteristics of propeller. The results showed that using SST k-ω and sliding mesh gave the numerical results closet to model tests. The maximum difference between K_T, K_Q of propeller and model tests was 9.6% and 7.4%. Ping Lu and Sue Wang [3] researched CFD simulation of propeller and tunnel thruster performance. They proved that SST k-ω model performed better solutions. The errors of K_T, K_Q were around 6%. Van den brink [4] conducted fine numerical simulations of the bow thruster using OpenFOAM and studied the scour depth at quay walls. Tu [5] used CFD with STAR-CCM+ to study the effect of grid types, grid sizes and turbulence models on the propeller performance. The results indicated that the mesh types, mesh sizes and the turbulence models were the factors affecting the simulation results. In which, SST k-ω with hexahedral mesh produced the good results in comparison with model tests.

The above findings play an important role in applying CFD in calculations of propeller hydrodynamics characteristics. On the basis of inheriting previous studies, this paper performs the bow thruster performance at different pitch angles using RANSE method with ANSYS CFX. The SST k-ω turbulence is used and the effect of mesh generation is also investigated. The flow field is evaluated based on the numerical data in order to understand why and how the hull contribute to the thrust.

2. Numerical setup

2.1. Thruster Parameters

The bow thruster which is taken from Ship Design and Research Centre (CTO S.A.) uses a four-blade controllable pitch propeller (CPP). It belongs to a research project of CTO S.A. and partners. The result from model test for the thrust and torque of the propeller at 20°: $T_{Prop} = 92.4$ N, $Q_{Prop} = 3.27$ Nm. The thrust on the hull and total thrust: $T_{Hull} = 52.1$ N, $T_T = 144.5$ N. Typical hull geometry for the simulation is shown in Figure 1. The dimensions of the hull model are listed in Table 1, and the main particulars of the thruster is illustrated in Table 2.

Table 1. Dimension of the hull model

Description	Unit
Length [m]	1.59
Width [m]	0.487
Depth [m]	0.6875
Height of the propeller shaft [m]	0.2338
Diameter of the tunnel [m]	0.24

Table 2. Main Particulars of the thruster

Description	Unit	Full scale	Model scale
Scale coefficient λ	-	1	5.133
Diameter D	m	1.216	0.2369
Number of blades Z	-	4	
Direction of rotation	-	Left-handed	

2.2. Numerical method and physical model

Various types of CFD solvers have their own advantages and drawbacks in terms of accuracy and computational costs. The RANSE method which relies on complete averaging of Navier-Stokes equations provide a strong balance of accuracy and computational requirements, so they were chosen for this paper. The SST k-omega is carried out for all the simulation cases. This is turbulence model giving more reliable results than other ones [5]. Based on previous research [1, 2], sliding mesh produces the sufficient accurate results. Hence, in order to simulate a rotating propeller, a sliding grid method is applied. This means that the rotation speed of the entire propeller domain is equal to the defined rotation speed of the propeller.

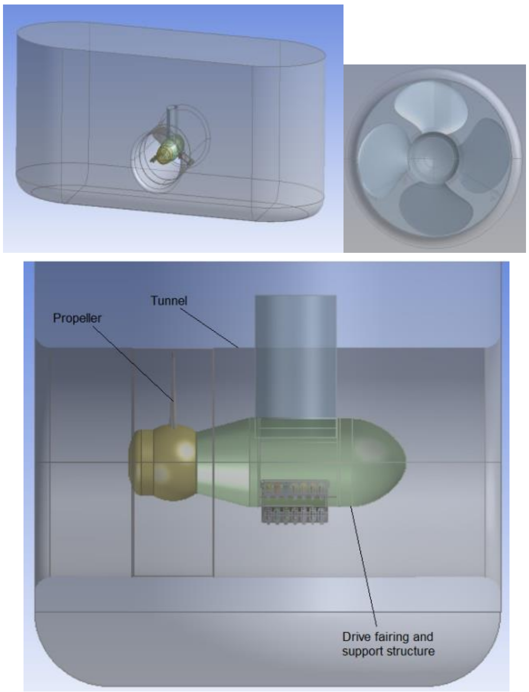


Figure 1. 3D hull geometry (from CTO S.A.)

2.3. Computational domain and test cases

Model test is carried out in the deep-water towing tank with the dimensions: length - 300.0m, width - 14.0m, depth - 6.0m (According to the description about the deep-water towing tank from CTO S.A.) [10]. The computing domain's height and breadth are set to 6.0m and 12.0m respectively, to match the experimental conditions and 12.0 m in length to ensure the wake field is properly created, reference from above research [1]. The simulation is conducted at different pitch angles: -20°, 10°, 15°, 20° and 24°. There are two computational domains: The whole domain (including the hull and support structure) - the stationary domain and the propeller domain - the rotation domain. The details of those domains are shown in Figure 2.

The rotational speed is constant, $n = 12$ rps. The density of water $\rho = 997$ kg/m^3 and the kinematic viscosity of water at 25^o is $\nu = 0.8926 \cdot 10^{-6}$ m^2/s.

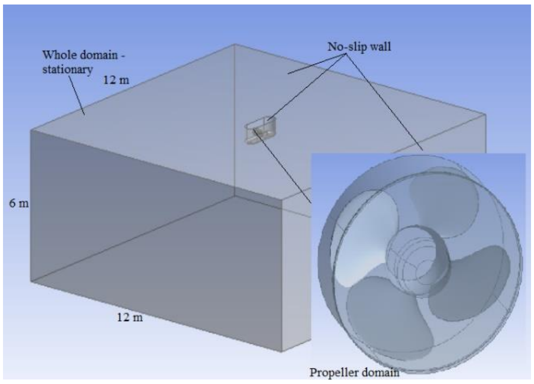


Figure 2. The computational domain

2.4. Mesh generation

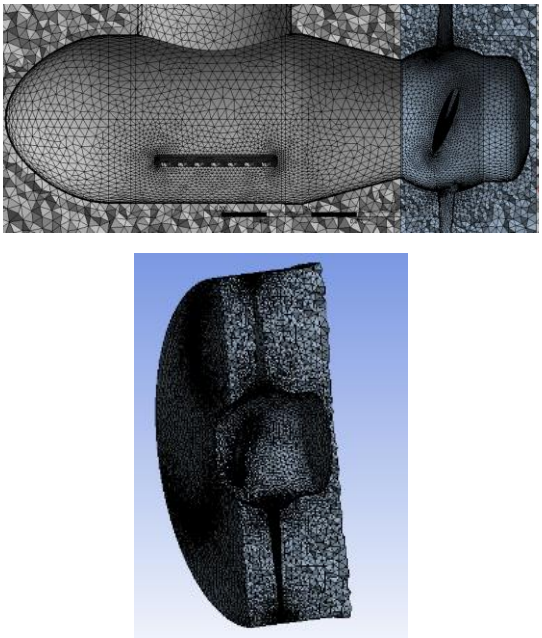


Figure 3. Mesh structure of the domain

The mesh type, mesh size and mesh generation are of the important factors affecting to the numerical results. The Patch Conforming Tetra mesh method provides support for 3D inflation and Built-in growth and smoothness control. The mesh will try to create a smooth size variation based on the specified growth factor [6]. Therefore, the tetrahedral cell type is used for the solutions. Because of the propeller's complex geometrical characteristics and the narrow distance between the blade tip and the tunnel, the inflations are applied. The inflation layer improved the accuracy of the viscous flow solution, thus, the inflation layers is applied at propeller blades, hub, around the tunnel and support structure part. In these area, the mesh is smoother to ensure the numerical results. Total elements for two domains is about 8.80 million. Figure 3 shows the general results of mesh in the domain.

2.5. Boundary conditions

No-slip boundary condition is applied for the stationary domain - whole domain and rotating domain - propeller domain, in addition, the rotating frame type is used in rotating domain (see as in Figure 2). The simulation is conducted in maneuvering condition, so the inlet velocity is set to 0. The influence of the free surface is assumed to be small and the symmetry condition is used to approximate the free surface effect. For all simulations here, 1^o per time - step was chosen according to the recommendations of the ITTC [7].

3. Results and discussion

3.1. Grid-convergence study

The first step in CFD calculation is to study the grid independence to avoid the errors caused by meshing. According to ITTC recommendation [8], the grid-convergence study is conducted in three different meshes with non - integer grid refinement ratio $r_G = \sqrt{2}$. This is used for pitch angle 20^o and the number of grid elements is illustrated in Table 3.

Table 3. The grid elements used in grid-convergence study

Grid types	Mesh size, m	The number of elements, million
Coarse	0.2	5.89
Medium	0.15	6.58
Fine	0.1	8.78

The change in obtained results when using various grids is defined as:

$$\varepsilon_{12} = (S_1 - S_2)/S_1; \varepsilon_{23} = (S_2 - S_3)/S_2 \qquad (1)$$

where: S_1, S_2, S_3 - The propeller numerical results of three kinds of grid - fine, medium and coarse grid.

The convergence of simulation results is evaluated through R_k.

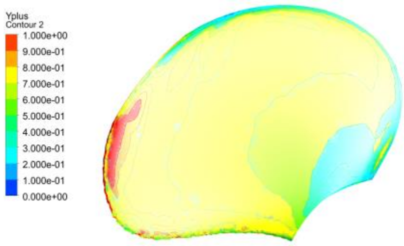
$$R_k = \frac{\varepsilon_{12}}{\varepsilon_{23}} \qquad (2)$$

$0<R_k<1$ - Monotonous convergence, $R_k<0$ - divergent convergence, $R_k>1$ - No convergence.

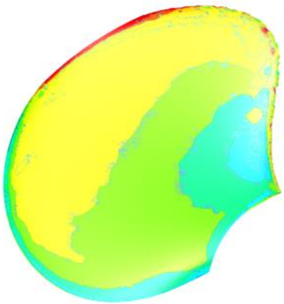
TẠP CHÍ ISSN: 1859-316X
KHOA HỌC CÔNG NGHỆ HÀNG HẢI
JOURNAL OF MARINE SCIENCE AND TECHNOLOGY

Table 4. Results of grid-convergence study at ϕ=20°

	Coa-rse	Medi-um	Fine	ε_{23}	ε_{12}	R_k
K_T	0.185	0.189	0.190	0.022	0.002	0.1
K_Q	0.031	0.031	0.030	0.025	0.012	0.5



Pressure side



Suction side

Figure 4. Distribution of y+ on propeller blade surfaces

Table 4 indicates the results of grid-convergence study at pitch angle 20°. From Table 4, the numerical results of propeller characteristics are monotonous convergence. However, the differences between the results of fine grid and medium grid are very small and not much different to experiment results (0.204 for K_T and 0.030 for K_Q). As shown Figure 4, the $y+$ distribution maintains below 1 when using fine mesh. This means that this kind mesh solution gives highly accurate results. Therefore, the fine mesh is chosen for all simulation calculations in this paper.

3.2. Numerical simulation results

The numerical simulation results of K_T, K_Q at various pitch angles ϕ compared to the experiment data are shown in Table 5 and Figure 5.

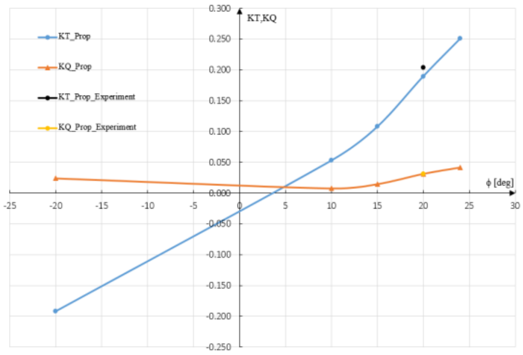


Figure 5. Performance characteristics of bow thruster

It can be seen that the numerical simulation results are almost close to the model test. At pitch angle 20°, the error for K_T is about 7.06% and 2.09% for K_Q. In short, the CFD results are in good agreement with the experiment data for the rest of simulation cases. Figure 5 shows that at around 5°, the flow goes in reversed direction, from support structure part to the propeller, as indicated in -20° in Figure 6.

The added thrust on the hull

This problem is explained according to Bernoulli's theory. It is said that any increase in kinetic energy in a fluid must be followed by a decrease in static pressure [9]. When the propeller starts operating, the fluid moves from outside into the tunnel, a corresponding volume must move a greater distance forward in the tunnel (narrower place) thus have a greater speed. Hence, with the changing of the fluid speed, it appears an area of low pressure on the hull and has additional force acting on the hull. Within the tunnel, pressure is lowest, then rises gradually as the flow field widens and

Table 5. Results of K_T, K_Q at various pitch angles ϕ

ϕ [deg]	K_T [-] Experiment data	K_T [-] CFD results	K_T [-] Error %	K_Q [-] Experiment data	K_Q [-] CFD results	K_Q [-] Error %
-20	-	-0.192	-	-	0.024	-
10	-	0.053	-	-	0.008	-
15	-	0.108	-	-	0.015	-
20	0.204	0.190	7.06	0.0305	0.031	2.09
24	-	0.250	-	-	0.041	-

TẠP CHÍ KHOA HỌC CÔNG NGHỆ HÀNG HẢI ISSN: 1859-316X
JOURNAL OF MARINE SCIENCE AND TECHNOLOGY

smaller beyond the outing of the tunnel. The flow filed at various pitch angle are shown in Figure 6. The flow field's direction at positive angles tends to start from the hub while at negative angle it goes in the opposite direction, as mentioned above. And, there are some turbulence flow appearing near the hull after the flow goes out. Consider ϕ=20º, the thrust on the hull (T_{Hull}=20.523N) contributes approximately 20% of force for the total thrust (T_T=114.706N).

4. Conclusion

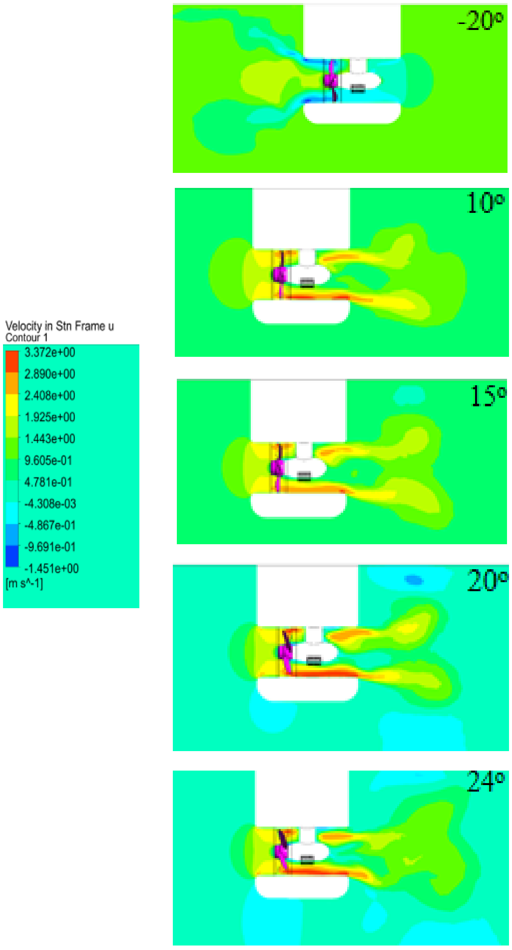


Figure 6. Velocity distribution in pitch angles: -20º, 10º, 15º, 20º and 24º

The paper applied successfully CFD method when studying the hydrodynamics characteristics of bow thruster. The obtained CFD results are in good agreement with the results of experiment, the errors of 7.06% for thrust coefficient and 2.09% for torque coefficient at pitch angle 20º. In addition, the influence of mesh size on the simulation results is investigated to get the suitable mesh for the calculations. Besides, the flow field around the hull and propeller is shown to explain how the hull also produces the thrust. This will be considered to predict the power of the bow thruster and its efficiency.

REFERENCES

[1] Feng Yukun, Chen Zuogang, Dai Yi, Zhang Zheng, Wang Ping: *An experimental and numerical investigation on hydrodynamic characteristics of the bow thruster*, Ocean. Eng. 209, 2020.

[2] Nakisa, M., M.J. Abbasi, and A.M. Amini: *Assessment of marine propeller hydrodynamic performance in open water via CFD*, Proceedings of the 7th International Conference on Maritime Technology (MARTEC 2010), 2010.

[3] Ping lu, Sue Wang: *CFD Simulation of Propeller and Tunnel Thruster Performance*, Proceedings of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering (OMAE2014), 2014.

[4] Van den brink, A.J.W.: *Modelling Scour Depth at Quay Walls Due to Thrusters*, Delft University, 2014.

[5] Tu, T.N., *Numerical simulation of propeller open water characteristics using RANSE method,* Alexandria Engineering Journal, Vol.58(2), pp.531-537, 2019

[6] Inc. ANSYS, *ANSYS Meshing User's Guide*, 2016.

[7] ITTC: *Recommended procedures and guidelines - practical guidelines for ship CFD applications*, section 7.5-03-02-03., 2014.

[8] ITTC: *Recommended procedures and guidelines - Uncertainty Analysis in CFD, Verification and Validation, Methodology and Procedures,* section 7.5-03-01-01, 2008.

[9]https://en.wikipedia.org/wiki/Bernoulli%27s_principle

[10] CTO S.A., *Research facilities tailored design, manufacture, delivery, launching and training*, 2014.

Received:	23 March 2022
Revised:	02 April 2022
Accepted:	06 April 2022