CFD simulation for the Wageningen B-Series propeller characteristics in open-water condition using k-epsilon turbulence model

Pham Minh Triet, Phan Quoc Thien, Ngo Khanh Hieu

***Abstract*— In the maritime industry, propellers are propulsive devices and play an important role in the performance of a ship. The hydrodynamic attributes of a propeller are described in terms of some dimensionless coefficients, such as thrust coefficient (K_T), torque coefficient (K_Q), and efficiency (η). However, it is arduous and usually expensive to determine the characteristics of a full-size propeller in open water condition tests. Thus, we need to look for another approach to analyze propeller characteristics. Nowadays, computational simulation has given us a powerful and efficient method to evaluate the performance of a propeller without consuming too many resources. In the scope of this paper, we shall evaluate the compatibility of using the k-epsilon turbulence model in Computational Fluid Dynamics (CFD) to analyze propeller performance, especially for the Wageningen B-Series propellers. For the validation of results, the numerical solutions will be compared with experimental data taken from the Netherlands Ship Model Basin open-water test in Wageningen. The goal of the research is to provide a well-founded framework for applying CFD in analysis and selection of Wageningen B-Series propeller, as well as other well-known propeller series.**

***Index Terms*— k-epsilon turbulence model, CFD, Wageningen B-Series propeller.**

Received: September 17th, 2017; Accepted: April 02th, 2018; Published: April 30th, 2018

This research is funded by Vietnam National University Ho Chi Minh City (VNU-HCM) under grant number C2017-20-01

Pham Minh Triet is a senior student at Department of Aerospace Engineering, HCMUT, VNU-HCM (e-mail: minhtriet240@gmail.com).

Phan Quoc Thien is a graduate student at Department of Aerospace Engineering, HCMUT, VNU-HCM (e-mail: phanquocthien@gmail.com).

Ngo Khanh Hieu is a senior lecturer at Department of Aerospace Engineering, HCMUT, VNU-HCM (e-mail: ngokhanhhieu@hcmut.edu.vn).

1 INTRODUCTION

Marine propeller characteristics play an important role to the performance of a ship. To operate effectively, those propellers are designed to provide the maximum thrust as well as minimum torque at the optimum rotational speed. One of the most common methods for evaluating propeller performance is the open-water test. However, due to the high cost of basin construction and propeller modeling, we tend to find a better approach. Along with the development of computer hardware, numerical simulation is emerging as an ideal solution because of its effectiveness and reliable result.

In Computational Fluid Dynamics (CFD), the flow is predicted by enforcing the conservation of mass and momentum. These conservation equations are commonly known as the Navier-Stokes equations. In general, marine propeller has complex geometry and as a consequence, the flow around it is very complicated and often turbulent. For simplicity, we can average the Navier-Stokes equations to get the mean flow, which is all we need during the design process (Fig. 1).

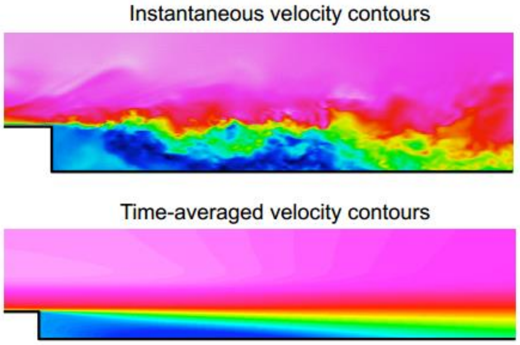

Figure 1. Different approaches to calculate a turbulent flow [1]

This method is called Reynolds-averaged Navier-Stokes (RANS). Nevertheless, there are some turbulence terms that must be calculated to accurately characterize the flow field. One method used to predict the effects of these terms is Turbulence Modeling.

Throughout the study, we shall use OpenFOAM—an open source framework solving fluid dynamics problems based on finite volume method—to analyze the Wageningen B-series propeller hydrodynamic performance. In OpenFOAM, there are many types of turbulence models based on RANS applicable for rotational motion problems such as pump, turbine, and propeller. Chang [2], Sanchez-Caja [3], and Senthil [4] used k-epsilon model for their studies, whereas Guilmineau [5] and Toumas [6] used k-omega SST model (a variation of k-omega model). These studies are all relevant and obtained appropriate results.

In this study, we analyze the Wageningen B-series propellers hydrodynamic characteristic using CFD simulation with k-epsilon turbulence model. The purpose is to verify the basic knowledge of how to predict and assess the effects of k-epsilon model on numerical results. A direct comparison between the obtained numerical results and theoretical analysis of Wageningen B-Series propeller [7] will be employed to validate the simulation.

2 PROPELLER GEOMETRY

2.1 Nomenclature

D	Propeller diameter	*m*
J	Advance ratio	--
P	Pitch	*m*
n	Rotational speed	*rpm*
k	Turbulent kinetic energy	m^2/s^2
ε	Turbulent dissipation	m^2/s^3
ω	Specific rate of dissipation	s^{-1}
T	Thrust	*N*
Q	Torque	*Nm*
K_T	Thrust coefficient	--
K_Q	Torque coefficient	--
η	Efficiency	--

2.2 Geometry

The geometry considered in this study is a Wageningen B-series propeller design. The 3D model of this propeller was created from the composite Ferguson curves and Conics with Ferguson segments [8] by the approach proposed by Ngo Khanh Hieu [9]. So according to Bernitsas [7], this Wageningen B-series propeller is a three blades propeller with the outlet diameter (D) of 240 mm, the blade area ratio (A_E/A_O) of 0.45 and the pitch to diameter ratio (P/D) of 0.70 at r/R = 0.75. It is named "B3_45_070" in short (see Fig. 2).

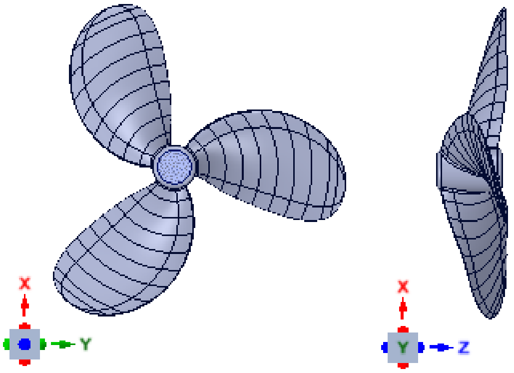

Figure 2. Wageningen B-series "B3_45_070" propeller

In the maritime industry, it is often desirable to consider the performance characteristics of a propulsion system through three non-dimensional coefficients which are the thrust coefficient (K_T), the torque coefficient (K_Q) and the efficiency (η). As a general rule, to present the hydrodynamic performance of marine propeller, the triad of those coefficients (K_T, K_Q, η) is plotted against advance ratio (J) [10].

To obtain the performance characteristics of the considered propeller in open water condition, simulations were done with a fixed rotational speed (n) of 330 RPM. The water velocity at the inlet varies from 0.132 m/s to 0.99 m/s corresponding to the advance ratio (J) from 0.1 to 0.75. The simulation results of each case will be validated by the experimental data [7] to ensure the reliability of the proposed CFD simulation.

3 MESH GENERATION

3.1 Computational domain

Multiple Reference Frame method is used to model the rotational motion of the propeller. This method requires two separated computational regions where two different reference frames are applied. The first region is the rotating part, surrounds the propeller, and virtually turns around the rotation axis. The second is the static part which covers the rest of the simulation domain limited by the far-field condition [11]. The mesh of B3_45_070 propellers was generated with ANSA pre-processor and then directly transferred to OpenFOAM, as shown in Fig. 3.

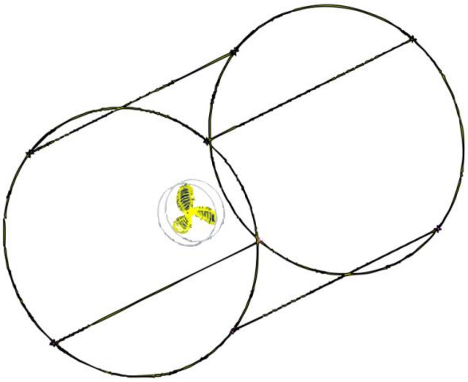

Figure 3. Computational domain generated in ANSA

Thereby, the rotating region contains the entire propeller specified with the dimensions of 1.15*D* in diameter and 0.38*D* in length. If the rotating region is too small, simulation results may be inaccurate due to the effect of large swirl near the propeller. However, if this region is too large, it will increase the calculation time. The static domain only needs to be large enough for the accelerated flow after the propeller can expand freely. Therefore, we chose the dimensions of the static domain are 2.5*D* in diameter and 10*D* in length. Computational domain is illustrated in Fig. 4.

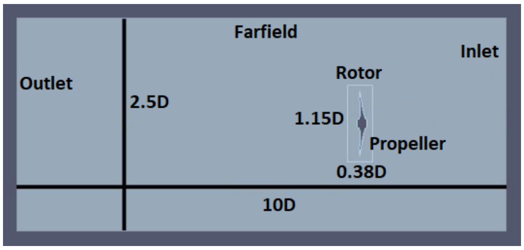

Figure 4. Mesh dimensions

3.2 *Meshing method*

The surface mesh was generated with triangle elements, as shown in Fig. 5. The minimum cell sizes located in the surface of the blade and the hub of the propeller are 0.24 mm (0.001*D*) and 1.2 mm (0.005*D*) respectively.

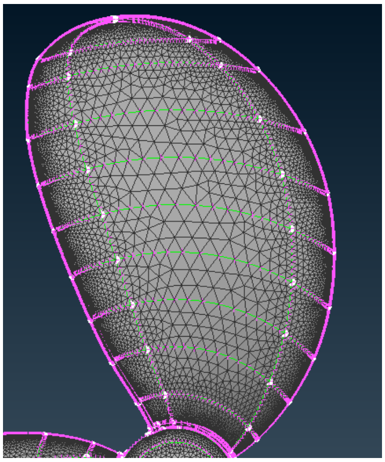

Figure 5. Surface mesh on the propeller blade

This study only focuses on assessing turbulence model rather than analyzing mesh, therefore, basic unstructured hybrid mesh with tetrahedron and prism elements was used (see Fig. 6). The near-wall region was split into prism elements forming boundary layers and tetrahedron elements were applied for space out of those layers (see Fig. 7). The growth factor of the mesh was chosen as ANSA default, which is 1.2. This method produces high boundary layer resolution, and can maintain the accuracy of simulation results.

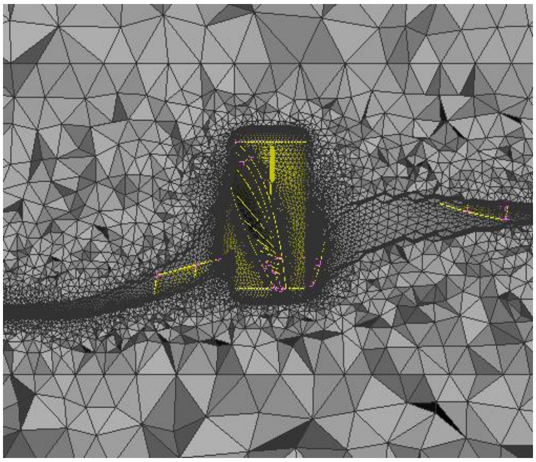

Figure 6. Unstructured mesh with tetrahedron elements

Moreover, the advantage of this type of mesh is its ease of generation, especially for complex geometries like propellers.

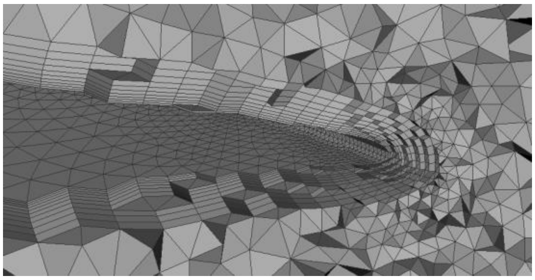

Figure 7. Prims elements at boundary layers

3.3 *Mesh quality*

In OpenFOAM, there are three vital aspects using as standard parameters for evaluating mesh quality [12], including non-orthogonality, aspect ratio, and skewness:

- *Non-Orthogonality*: the angle between the face normal and the vector between the cell midpoint and the face. In order to obtain well-converged solutions, high non-orthogonal cells should be avoided.
- *Aspect ratio*: the ratio between the longest and the shortest length in a cell. High aspect ratio implies that the cells are stretched in one direction. One of the reasons lead to poor results is that the cells with high aspects ratio are not aligned with the local flow structure.
- *Skewness*: the nearest of the intersection between the face nodes and the vector from the center node and the neighbor node. Although it reduces the solution quality, this issue is unavoidable when dealing with complex geometry, such as marine propellers.

The optimal range of each parameter is introduced briefly in Table I below.

Table I. Optimal values for mesh quality

Keyword	Optimal Value
Aspect Ratio	As low as possible
Non-orthogonality	< 70 (65 will be ok)
Skewness	< 4

The criteria of the mesh are checked using the *check Mesh* module in OpenFOAM and satisfy the computational requirements.

3.4 *Mesh sensitivity analysis*

We shall perform the mesh sensitivity analysis at J = 0.6 with k-epsilon model, and compare the results with experimental data (K_T = 0.0682, K_Q = 0.0102). Mesh sensitivity results are shown in Table II below.

Table II. Mesh sensitivity analysis

Y^+	Elements	Times (s)	ΔK_T (%)	ΔK_Q (%)
60	2.43E+6	1042	1.99	10.31
40	2.58E+6	1403	1.53	9.33
30	2.77E+6	1627	1.49	7.41
20	3.40E+6	1694	0.83	6.38
10	3.85E+6	2263	1.93	3.43
5	4.47E+6	2463	1.42	3.76

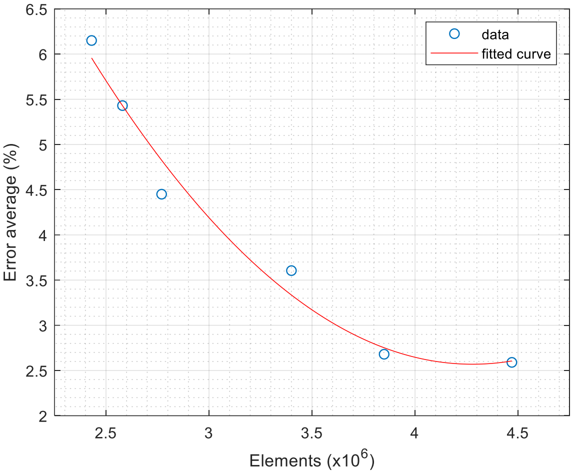

Figure 8. Mesh convergence graph

Generally, the results will be better with more elements, but there is an optimum point, where the results are good enough and the computational time is not too high. This optimum point is crucial for evaluating the influence of mesh model on simulation results, and can be found by a mesh convergence study. It should be noted that there is a correlation between the Y^+ value and the number of mesh elements. The higher the Y^+, the greater the number of mesh elements. For this reason, we shall carry out a mesh sensitivity analysis based on the Y^+ value.

In most cases, we should choose the Y^+ value for k-epsilon model from 30 to 300 [1]. However, this value can be varied and affected by many aspects, such as the object of interest, simulation condition, and study purpose. Particularly, our mesh sensitivity study shows an optimum point beyond the recommended range. As the mesh density increase, the error average converges at around 2.55% (Fig. 8). Ultimately, increasing the mesh density further produces only minor increases in accuracy. Therefor we shall choose the model with Y^+ = 10 to have a proper balance between the simulation time and accuracy.

4 SOLUTION AND SOLVER SETTING

4.1 Solver

We use Multi-Reference Frame method to simulate the propeller rotation. In this method, the simulation domain is divided into two regions corresponding to two different reference area (rotor and stator). This is a commonly used method in rotary motion simulation such as pump, turbine, and propeller [12].

In simulating the hydrodynamic performance of a propeller, we set out a number of hypotheses to simplify the case. These are steady-state flow, non-cavitation and incompressible. After giving the above assumptions, we shall use MRF simple Foam algorithm in OpenFOAM to start the simulation. This algorithm is based on multi-reference frame method, which is applicable to steady-state and incompressible problems.

4.2 Initializations

B-series open-water tests were conducted at Netherlands Ship Model Basin (NSMB) with the following properties [9].

- Propeller diameter: D = 240 mm
- Rotational speed: n = 330 rpm
- Water at 20^0C

The initial conditions for our simulation case will be set exactly the same with the test conditions of NSMB.

Boundary conditions for the case study are described carefully in Table III and IV.

Table III. Boundary conditions for *u, P* and *nuy* turbulence

PATCH	*U*	*p*	*nut*
Inlet	Fixed Value	Zero Gradient	calculated
Outlet	Inlet Outlet	Fixed Value	calculated
Farfield	slip	slip	calculated
Propeller	Fixed Value	Zero Gradient	Nut Wall Function

Table IV. Boundary conditions for *k* and *epsilon*

PATCH	*K*	*Epsilon*
Inlet	Fixed Value	Fixed Value
Outlet	Inlet Outlet	Inlet Outlet
Farfield	slip	slip
Propeller	kqR Wall Function	Epsilon Wall Function

In general, a directory of *simple Foam* simulation case includes three folders: *0, constant,* and *system*. In these folders, folder *0* contains the initial condition files of the case; folder *constant* contains geometry file (polyMesh) and model properties files; folder *system* contains the program's control files.

4.3 Turbulence modeling

Turbulence models play an important role in CFD simulations. Since each turbulence model has its own advantage and weaknesses, the application of a turbulence model must base on the specific requirements of the simulation case.

K-epsilon is a two-equation model which gives a general description of turbulence by means of two transport equations. This model is widely used for industrial applications because of its robustness and reasonably accurate for a wide range of applications. This model uses a wall function to compute the area near the propeller wall (boundary layer), thus requiring a mesh model with Y^+ within the outer region ($Y^+ > 5$) [1]. From the mesh sensitivity study, we created a mesh with $Y^+ = 10$ for k-epsilon model.

5 RESULTS AND EVELUATION

5.1 Mesh model

The mesh properties obtained from *checkMesh* module are summarized in Table V.

Table V. Mesh quality

Property	Value
Tetrahedron element	1,970,908
Prism element	1,883,820
Max skewness	2.26462 (ok)
Max non-orthogonality	63.9777 (ok)
Max aspect ratio	32.3013 (ok)

5.2 Results analysis

To evaluate the simulation results, in addition to the convergence criterion of residuals, we also consider other factors such as velocity distribution, pressure distribution and compare with experimental data.

The convergence residuals for our case study were set at 5.0E-5 (see Fig. 9). The figures below show the flow field distribution at the point of highest efficiency (J = 0.6).

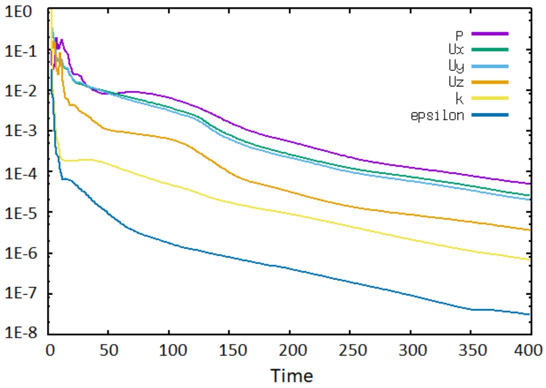

Figure 9. Convergence graph

❖ Velocity field at J = 0.6

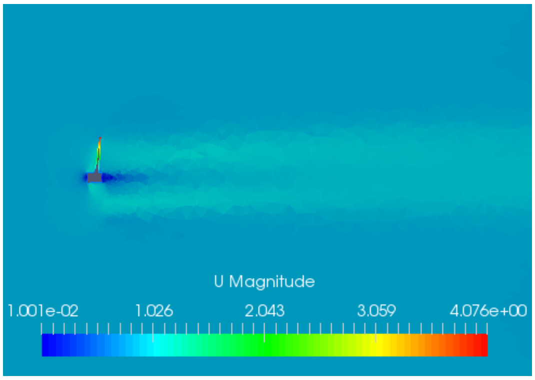

Figure 10. Velocity distribution at J = 0.6

❖ Pressure field at J = 0.6

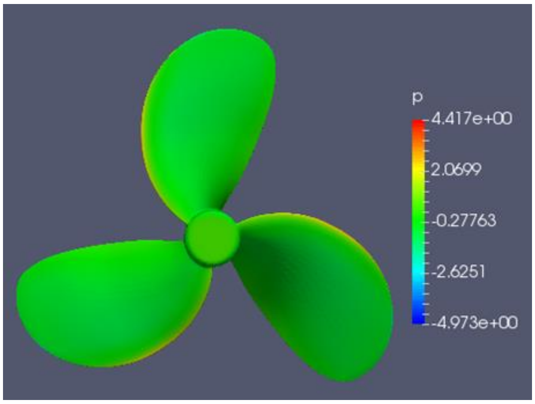

Figure 11. Pressure distribution at suction side

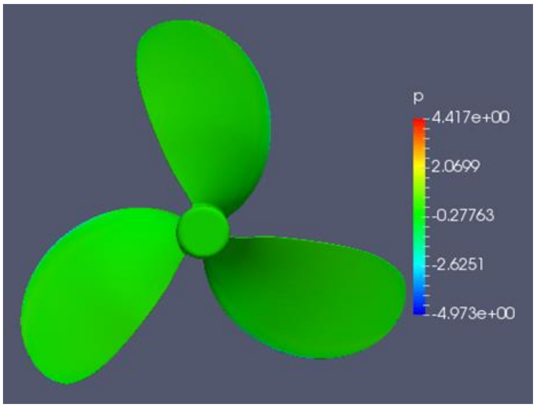

Figure 12. Pressure distribution at pressure side

It can be seen that the velocity field (Fig. 10), and pressure distribution fields (Fig. 11 and 12) precisely describe the actual response of the free stream over a rotating surface. The flow is accelerated and expands freely behind the propeller. The pressure at the suction side of the propeller will be lower than the pressure side.

To give more accurate assessments about the turbulence model, the results of K_T, K_Q, and η at different value of J from 0.1 to 0.75 are compared with experimental data. Tables VI, VII, and VIII show the simulation results of B3_45_070's performance in open water condition. These results will then be compared with the experimental data, obtained from the tests at Netherlands Ship Model Basin [9].

Table VI. Thrust results

J	T (N)	K_T	K_T (exp)	ΔK_T (%)
0.1	26.5024	0.2574	0.2390	7.689
0.2	23.3104	0.2264	0.2101	7.747
0.3	19.7908	0.1922	0.1782	7.854
0.4	15.8962	0.1544	0.1437	7.428
0.5	11.6857	0.1135	0.1069	6.159
0.6	7.15826	0.0695	0.0682	1.931
0.65	4.79996	0.0466	0.0482	3.289
0.7	2.37683	0.0231	0.0279	17.267
0.75	0.15556	0.0015	0.0038	60.653

Table VII. Torque results

J	Q (N.m)	K_Q	K_Q (exp)	ΔK_Q (%)
0.1	0.6222	0.02518	0.0254	0.885
0.2	0.5691	0.02303	0.0228	0.994
0.3	0.5079	0.02055	0.0201	2.239
0.4	0.4337	0.01755	0.0170	3.242
0.5	0.3458	0.01399	0.0138	1.381
0.6	0.2434	0.00985	0.0102	3.429
0.65	0.1877	0.00759	0.0084	9.559
0.7	0.1289	0.00522	0.0065	19.75
0.75	0.0657	0.00266	0.0045	40.91

Table VIII. Efficiency results

J	η	η (exp)	$\Delta\eta$ (%)
0.1	0.1627	0.1499	8.546
0.2	0.3129	0.2928	6.876
0.3	0.4466	0.4241	5.294
0.4	0.5599	0.5368	4.313
0.5	0.6455	0.6178	4.482
0.6	0.6739	0.6359	5.980
0.65	0.6348	0.5954	6.611
0.7	0.4929	0.4822	2.239
0.75	0.0678	0.1964	65.466

From the simulation results, it is obvious that k-epsilon model gives quite good result and well match with experimental data, except at the high advance ratio (J = 0.7 and 0.75). In particular, at low advance ratio, the differences between simulation results and experimental data are lower than 10%, and even below 3% for torque coefficient. At medium advance ratio, the differences are remained the same for thrust and efficiency. There is a minor increment in torque error, but the overall differences are still in an acceptable range (lower than 10%). Since Senthil [4] accepted the percentage difference of 12.5% between the CFD values and experiment based data, our simulation results can be considered acceptable.

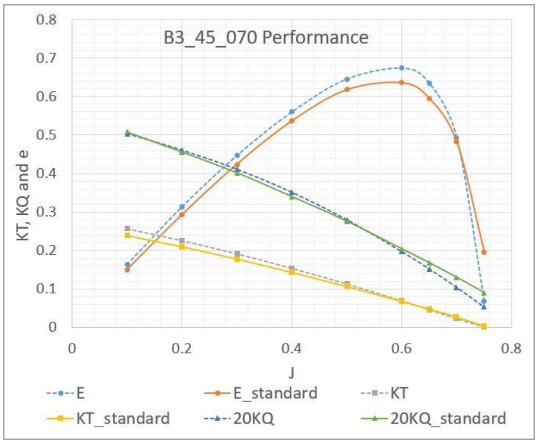

Figure 13. Performance graph of B3_45_070

As shown in figure 13, there is a significant difference in the range of high advance ratio (J = 0.7 to 0.75). This is the range where propeller efficiency drops very fast. The reason for this phenomenon is due to the generation of large swirls and separation flows. In this range, the flow behind propeller became slowdown and eventually slower than the free stream flow. The propeller still rotates but no longer creates thrust, resulting in an increment of drag. At the same time, the flow around the propeller will be separated and creates large vortices.

One weakness of k-epsilon model is that it will give poor prediction with large swirl and strong separation flows. Therefore, the simulation results using k-epsilon model will be inaccurate at J = 0.7 and 0.75.

6 CONCLUSION

From the study above, we have had some knowledges about applying k-epsilon model in turbomachinery simulation. During the conceptual design of a ship, we only concern about propeller performance at the maximum efficiency. The weakness of k-epsilon model can be ignored. In fact, if we accept the difference between simulation results and experimental data in a suitable range (lower than 10%), then k-epsilon will be the best turbulence model due to its ease of application.

K-epsilon model uses wall function to calculate the near-wall region flows. This method will theoretically require a coarser mesh at the boundary layer, thus well suited for simple problems and facilitates fast simulation time. However, this method cannot handle flows with large separation due to the coarse mesh at the boundary layer.

In order to achieve lower tolerances in simulation result, other turbulence models which have better prediction at the boundary layer such as k-omega and k-omega SST should be applied. However, these models require mesh resolution with $Y^+ < 1$ to take full advantages. This could be helpful in-depth analysis but still inefficient in industrial application. Therefore, further studies on meshing method and rotational modeling should be conducted if we want to use these turbulence models.

REFERENCES

[1] ANSYS INC, *Introduction to ANSYS FLUENT – Lecture 6*, ANSYS Inc, 2010.

[2] B. Chang, *Application of CFD to P4119 propeller*, 22nd ITTC Propeller RANS/Panel Method Workshop, France, 1998.

[3] A. Sanchez-Caja *P4119 RANS calculations at VTT*, 22nd ITTC Propeller RANS/Panel Method Workshop, France, 1998.

[4] S. Prakash and D. Nath, "A computational method for determination of open water performance of a marine

propeller," *International Journal of Computer Applications*, vol. 58, no. 12, 2012.

[5] E. Guilmineau, G.B. Deng, A. Leroyer, P. Queutey, M. Visonneau and J. Wackers, *Wake simulation of a marine propeller*, 11th World Congress on Computational Mechanics, 2014.

[6] T. Turunen, T. Siikonen, J. Lundberg, and R. Bensow, "Open-water computations of a marine propeller using OpenFOAM," in *ECFD VI-6th European Congress on Computational Fluid Dynamics, Barcelona, Spain, 20-25 July 2014*, 2014, pp. 1123-1134.

[7] M. M. Bernitsas, D. Ray, P. Kinley, *K_T, K_Q and Efficiency Curves for the Wageningen B-Series Propellers*, University of Michigan, 1981.

[8] A. Saxena and B. Sahay, *Computer aided engineering design*. Springer Science & Business Media, 2007.

[9] Ngô Khánh Hiếu, Lê Tất Hiển, "Đặc trưng hình học và đặc tính thủy động lực chân vịt phương tiện thủy nội địa cỡ nhỏ", *Tạp chí Phát triển Khoa học và Công nghệ, Đại học Quốc gia TP. HCM*, tập 18, số K7-2015, tr. 110-116.

[10] A. B. Murray, B. Korvin-Kroukovsky, and E. V. Lewis, *Self-Propulsion Test with Small Models*. SNAME, 1951.

[11] Phan Quốc Thiện, Bùi Khắc Huy, Lê Tất Hiển, Ngô Khánh Hiếu, "Mô phỏng số chân vịt tàu thủy theo phương pháp đa vùng tham chiếu sử dụng OpenFOAM," *Tạp chí Khoa học Công nghệ Giao thông Vận tải, Trường Đại học Giao thông Vận tải TP. HCM*, tập 20, tr. 56-60, 2016.

J. H. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*, 3rd edition, Springer, 2002.

Pham Minh Triet was born in Ho Chi Minh City, on March 29, 1995. He is currently a senior-year Aerospace Engineering student at Ho Chi Minh City University of Technology, VNU-HCM.

Phan Quoc Thien (1989, Vietnam) received Bachelor degree in Aerospace Engineering (2012) at HCMUT (VNU-HCM). He is currently a M.Sc. student at VNU-HCMUT.

Ngo Kanh Hieu (1978, Ho Chi Minh, Vietnam) received Bachelor degree in Aerospace Engineering (2001) at HCMUT (VNU-HCM), M.S. degree in Mechanics (2002) and PhD degree in Computer Science (2008) from LIAS-ENSMA, France. He is currently working as an Associate Professor of Aerospace Engineering at HCMUT, VNU-HCM. Work experience: Flight Dynamics, Propeller-driven Propulsion System, Control System Analysis and Design.

Mô phỏng số đặc tính thủy động học của chân vịt Wageningen B-series với mô hình rối k-epsilon

Phạm Minh Triết, Phan Quốc Thiện, Ngô Khánh Hiếu*

Trường Đại học Bách khoa, ĐHQG-HCM
*Tác giả liên hệ: ngokhanhhieu@hcmut.edu.vn

Ngày nhận bản thảo: 17-9-2017; Ngày chấp nhận đăng: 02-4-2018; Ngày đăng: 30-4-2018

***Tóm tắt* - Trong ngành công nghiệp tàu thủy, chân vịt là một bộ phận cấu thành hệ thống đẩy giữ vai trò quan trọng đối với đặc tính hoạt động của tàu. Đặc tính thủy động của chân vịt tàu thủy được thể hiện thông qua các đại lượng vô thứ nguyên đặc trưng như hệ số lực đẩy (K_T), hệ số moment xoắn (K_Q), và hiệu suất (η). Tuy vậy, việc thử nghiệm đặc tính thủy động của một chân vịt tàu thủy ở kích thước thật của nó trong điều kiện dòng tự do là việc rất khó khăn và tốn kém. Ngày nay, công cụ mô phỏng số đã cho thấy được khả năng và tính hiệu quả của phương pháp mô phỏng số đối với việc đánh giá đặc tính hoạt động của chân vịt tàu thủy mà không tốn quá nhiều nguồn lực. Bài báo sẽ tập trung vào việc đánh giá sự phù hợp của mô hình rối k-epsilon áp dụng cho mô phỏng số đặc tính thủy động học của chân vịt tàu thủy, đặc biệt cho mẫu chân vịt B-series của Wageningen. Kết quả mô phỏng số sẽ được so sánh với dữ liệu thực nghiệm trong bể thử đã được công bố bởi Netherlands Ship Model Basin (NSMB). Mục tiêu của nghiên cứu là cung cấp một mô hình mô phỏng số đặc tính thủy động học của chân vịt Wageningen B-series với mô hình rối k-epsilon hướng đến đến áp dụng công cụ mô phỏng số vào quá trình thiết kế lựa chọn phù hợp chân vịt tàu thủy với chuẩn Wageningen B-series, cũng như các mẫu chân vịt thông dụng khác.**

***Từ khóa* - mô hình rối k-epsilon, CFD, chân vịt Wageningen B-series**