Prediction of marine propeller characteristics using RANSE method

A M Ghoniem[1], A S Abo Elazm[2], M I Benaya[3] and S A Mohamed[4]

[1]MSc. Student, Military Technical College, Ships and submarines Engineering Department, 11766 Cairo, EGYPT
[2]Professor, Military Technical College, Ships and submarines Engineering Department, 11766 Cairo, EGYPT
[3]Doctor, Military Technical College, Ships and submarines Engineering Department, 11766 Cairo, EGYPT
[4]Doctor, Military Technical College, Ships and submarines Engineering Department, 11766 Cairo, EGYPT

E-mail: adelmohab@mtc.edu.eg

Abstract. A simulation of a marine propeller was done to obtain the propeller's open water characteristics using RANSE method with the approach of rotating reference frame. The effects of grid type, mesh density and turbulence models on simulation results were analyzed by comparing the results got from a four different cases representing combining turbulence models with grid types. The verification and validation of results were done by using the Potsdam propeller PPTC results. Eventually, choosing the grid type and the right turbulence model corresponding to each case was shown to obtain accurate results with minimum error from the experimental results. The results shows superiority of tetrahedral grid type with K-epsilon turbulent model in simulating the issued marine propeller performance.

1. Introduction

Various ships are being propelled by the screw propeller as its low cost and easiness of maintenance. The accurate prediction of the propeller's performance helps to reduce the cost of manufacture rather than the try and error approach. Because of the relatively accurate results with computational time and lower cost compared to the experimental approach, the rapid development of computational resources led to the solution of many ship hydrodynamics problems using the Computational Fluid Dynamics (CFD) method. Lifting Line Theories, Blade Element-Momentum Theory, Surface Panel Methods, Boundary Element Methods (BEM), and Reynolds Averaged Navier-Stokes Equation (RANSE) are some of the CFD applications used to estimate propeller open water characteristics. The numerical simulation of propeller open water characteristics is an important element of hydrodynamics which can be modelled via RANSE method. The approach has been widely used for the prediction of hydrodynamic characteristics for various purposes, such as designing ships and underwater vehicles [1]. RANSE-based models allow researchers to investigate the effects on the propeller performance due to changes of flow, operating conditions, and shape of the propeller [2].

The RANSE turbulence models are capable to account for turbulence, pressure, as well as viscous effects, which makes it a powerful tool for open water performance analysis [1]. The RANSE algorithm combines the numerical simulation with a control-volume approach in order to solve the equations based on a variety of user-defined inputs [3]. This approach has made the RANSE method a popular choice for simulation studies of open water propeller characteristics, allowing for a cost-effective and accurate representation of the performance of the propeller [4]. RANSE is more popular than other methods because its flow model is similar to flow physics in reality with suitable computational time. However, it comes at a higher cost and takes more time to compute [2-4].Furthermore, with the advancement of technology, computational power, and the development of CFD code, RANSE simulations are now considered the preferred practice.

For a CFD RANSE simulation to function properly and deliver accurate results, the mesh (mesh type, mesh size, and mesh structure) and physics models must be chosen carefully. The performance of propellers has been predicted by numerous authors and researchers using RANSE. In order to compare various methods for calculating propeller open water characteristics, including sliding grid, rotating reference frame, and rotating domain, Tran Ngoc Tu et al. [5] used the RANSE method. Tetrahedral grid calculations and a two equation

"SST k-omega" turbulence model were used for the simulation. In light of the study's findings, a rotating reference frame is an efficient method for simulating open water in terms of computation time, degree of accuracy, and convergence of results. Propeller VP1304 characteristics error at large advance coefficients J varies from 0.33 to 13 percent in comparison to the experiment. The "SST k-omega" turbulence model in combination with a sliding grid, according to research by Nakisa et al., produced the best results when they looked at the impact of various turbulence models on open-water propeller performance results.The mean errors of the thrust coefficient (K_T), torque coefficient (K_Q), and propeller open water efficiency (eta_O) when the results are compared to experimental data are 8, 13, and 11%, respectively [6]. In order to perform an unsteady RANSE analysis on the numerical simulations of the hydrodynamic open-water characteristics of a ship propeller Judyta Felicjancik et al. used the commercial solver Star-CCM+. Stable reference frame techniques were applied during simulation. An unstructured tetrahedral grid was used for the calculations, and a two-equation "SST k-omega" turbulence model was used for the simulation. The propeller open water efficiency (eta_O) errors at large J values of propeller VP1304 range from 3.02 to 11.2 percent when compared to experimental data [7]. Joao M. Baltaza et al. looked into how domain size, boundary conditions, iterative and discretization errors, and propeller force predictions in open water are affected. His findings show that the effect of domain size and boundary conditions on predictions of propeller force is less than 1% [8]. Da-Qing discovered that the error difference between his numerical results and experimental values for K_T and K_Q within a specific range of advance coefficient J is 3 and 5 percent, respectively. Their research involved a RANSE prediction of the open water characteristics of a highly skewed propeller. Furthermore, he stated that grid refinement creates different results for local flow quantities but has little impact on propeller performance characteristics [9].

Future studies predicting propeller open water characteristics using the RANSE method will benefit from the above-mentioned published literatures. However, they did not look into how combining mesh type and turbulence models would affect the outcomes. In order to determine the optimality criteria for the simulation that produces the most accurate propeller open-water characteristics, this paper will examine the effects of the aforementioned factors. The well-known Potsdam Propeller PPTC test case is employed to confirm and validate the veracity of case studies. Utilizing the rotating reference frame method, simulations were run. The computation was done using the FLUENT commercial package from ANSYS.

2. Numerical simulations

2.1. Propeller model

In this study, a propeller in model scale of 12 was considered, which was designed and tested experimentally to provide data for both flow physics exploration and CFD validation: Potsdam propeller PPTC with a pitch coefficient of P/D @0.7R=1.635. In the model scale [10], this is a right-handed adjustable pitch propeller with a diameter of 0.25 m. Figure 1 and Table 1 show the propeller geometry and basic geometric data of the studied propeller.

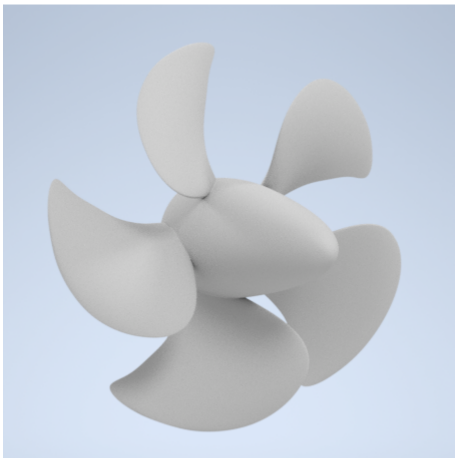

Figure 1 Case study propeller geometry

Table 1 Main dimensions of the propeller selected for the study[10]

Description	Unit	•
Diameter	m	0.25
P/D@0.7R	•	1.365
Blade area ratio	•	0.779
Hub diameter ratio	•	0.3
Number of blades	•	5

2.2. Numerical setup

2.2.1. Test cases The open water simulation is performed under the same conditions as the experiment [10], but with a different advance coefficient J ranging from 0.6 to 1.4 with a step of 0.2 to narrow the study to the highest values of efficiency. The propeller revolution was kept constant at n = 15 rev/sec, while J was changed by varying the advance velocity. The water parameters (density, viscosity) were chosen corresponded to real values (density of water ρ = 998.67 kg/m^3, viscosity of water ν= 1.070.10-6 m^2/s [10]).

2.2.2. Computation method The characteristics of propellers in open water were calculated using the rotating reference frame method. The result is fully equivalent to the case of actual propeller rotation, and this method, which involves adding additional terms to the momentum equation, is completely suitable for open water analysis. However, because physical motion of the computational grid is avoided, the computational time is shorter and the convergence is quicker [5].

2.2.3. Domain size and boundary conditions Boundary conditions and the size of the computation domain are significant variables that affect the numerical outcomes. In order to guarantee uniform incoming flow upstream of the propeller and prevent reflections downstream of the propeller, the computation domain size should be set in such a way. The following dimensions, expressed as multiples of propeller diameter D, define the cylinder-shaped computation domain for open-water propeller simulation, per ITTC recommendations [11]. The outlet and outer boundary are 8D and 2.5D away from the propeller plane and axis, respectively, while the inlet is 4D away from the chord's midpoint in the root section. The inlet boundary condition, also known as the velocity inlet boundary condition, was employed as a constant velocity over the entire inlet plane. A pressure outlet condition with atmospheric pressure value was applied to the outlet. The symmetry plane condition was used on the outer boundary. There are no-slip walls on the shaft, hub, and propeller. Figure 2 depicts the domain and boundary conditions for propeller open water simulation.

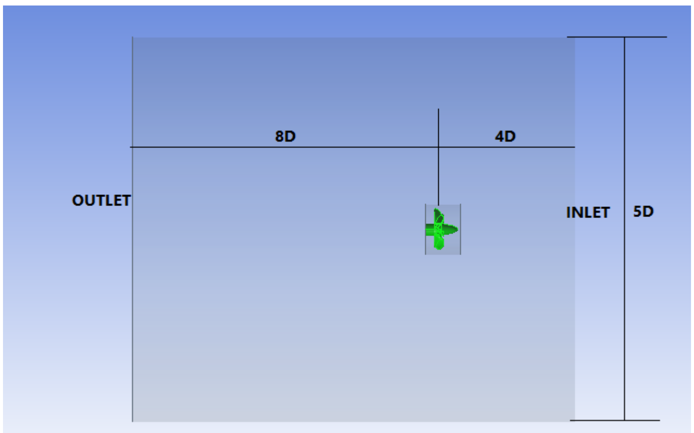

Figure 2 The computational domain specification of the simulation

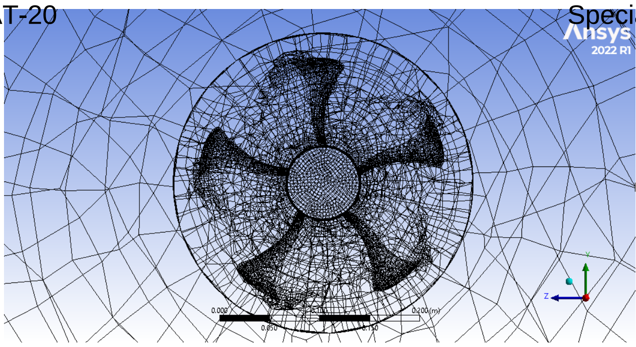

Figure 3 Hexahedral grid generated for simulation

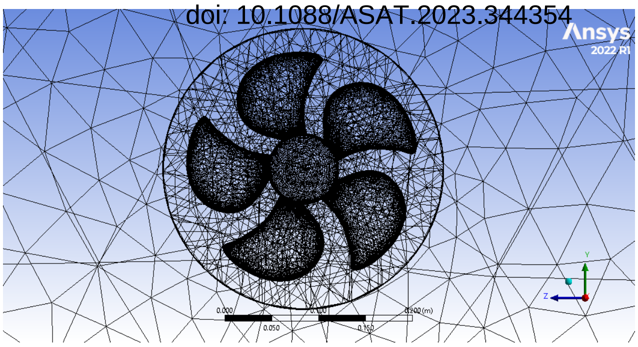

Figure 4 Tetrahedral grid generated for simulation

2.2.4. Mesh generation Hexahedral grids and tetrahedral grids, two different types of core volume meshes, were used to run the simulation. The region in front of and behind the propeller, which is very far from the propeller, was avoided using fine grid by using local refinement; additionally, the leading edge, trailing edge, and tip of propeller blades were further refined because of the significant curvature of the blade surface in these regions [12]. In order to accurately capture the flow behaviour close to the walls, the boundary layer was resolved using a prism layer. The non-dimensional normal distance y+ of the first cell layer next to the wall is kept well below 5 during grid generation in order to resolve the near wall boundary layer. This range corresponds to the viscous sub-layer in model scale simulation. Figures 3-4 depict the mesh generation results.

Table 2 Cases of mesh study

Case no.	Grid and Turbulance model
1	Hexahedral grid type and k-epsilon
2	Hexahedral grid type and SST k-omega
3	Tetrahedral grid type and k-epsilon
4	Tetrahedral grid type and SST k-omega

2.2.5. Physics model The RANSE equation for steady flow in three dimensions is used to calculate the flow field around the propeller. The choice of a suitable turbulence model is still problematic because there is no ”universal” turbulence model [13].As a result, it is very challenging for those who are new to CFD calculations to choose it as a method for accurately forecasting propeller performance. To compare the impact of turbulence models on the results obtained, two equation models K-epsilon (Realizable K-Epsilon Two-layer) and K-omega (SST K-Omega) are used in this study. Table 1 shows the four cases which have been studied.

3. Results and discussion

3.0.1. Mesh independence study Mesh independence study plays vital role in determining the accuracy of the simulation. The difference between the exact solution of the differential equations and the difference equations solution results mainly from the discretization errors. It's a must to ensure that the used grid is suitable enough to reduce the error resulted from these discretization errors. In this study, three grids have been used to conduct the mesh sensitivity study in order to determine the best mesh density at which the difference between results obtained from two subsequent meshes reaches low values. The mesh sensitivity study was carried out using three grids differ in cells number at advance coefficient J = 1, as fine, medium, and coarse grid as shown in table 4. A further grid refinement wouldn't be necessary after the medium grid as the the disparities between solution of fine and medium grid was lower than 1%. On the other hand, the solution of the coarse grid couldn't be chosen as its error was higher than the medium's grid error by 6%. For that reason, medium grid was chosen to complete the study.

Table 3 Open water CFD results compared to experimental results

J	K_T	•	•	$10K_Q$	•	•	η_O	•	•
•	EFD	CFD	Error%	EFD	CFD	Error	EFD	CFD	Error%
Case 1: Hexahderal with k-epsilon									
0.6	0.629	0.610	3%	1.396	1.413	1.2%	0.520	0.510	1.9%
0.8	0.51	0.511	0.2%	1.178	1.226	4.0%	0.544	0.531	2.4%
1.0	0.399	0.401	0.6%	0.975	1.014	4.1%	0.643	0.630	2.1%
1.2	0.295	0.284	3.6%	0.776	0.785	1.1%	0.717	0.692	3.4%
1.4	0.188	0.175	7.1%	0.559	0.535	4.4%	0.740	0.728	1.6%
Case 2: Hexahderal with k-ω-SST									
0.6	0.629	0.608	3.3%	1.396	1.461	4.7%	0.520	0.503	3.3%
0.8	0.51	0.509	0.1%	1.178	1.268	7.6%	0.544	0.512	5.9%
1.0	0.399	0.400	-0.2%	0.975	1.049	7.6%	0.643	0.606	5.7%
1.2	0.295	0.283	4.0%	0.776	0.812	4.6%	0.717	0.666	7.0%
1.4	0.188	0.174	7.3%	0.559	0.553	1.1%	0.740	0.702	5.1%
Case 3: Tetrahderal with k-epsilon									
0.6	0.629	0.621	1.2%	1.396	1.392	0.3%	0.520	0.518	0.5%
0.8	0.51	0.509	0.1%	1.178	1.207	2.5%	0.544	0.537	1.2%
1.0	0.399	0.400	0.2%	0.975	1.005	3.1%	0.643	0.633	1.5%
1.2	0.295	0.297	0.6%	0.776	0.810	4.3%	0.717	0.700	2.3%
1.4	0.188	0.188	0.1%	0.559	0.577	3.2%	0.740	0.725	1.9%
Case 4: Tetrahedral with k-ω-SST									
0.6	0.629	0.614	2.4%	1.396	1.387	0.7%	0.520	0.515	1.0%
0.8	0.51	0.504	1.3%	1.178	1.202	2.0%	0.544	0.533	1.9%
1.0	0.399	0.394	1.2%	0.975	1.009	3.5%	0.643	0.622	3.3%
1.2	0.295	0.287	2.7%	0.776	0.805	3.7%	0.717	0.681	4.9%
1.4	0.188	0.175	6.9%	0.559	0.570	2.0%	0.740	0.684	7.5%

Table 4 Mesh study results

Mesh number	K_T error	K_Q error	η_O error
716988	5.2%	9.5%	6.9%
2361763	0.2%	3.1%	1.5%
4661805	0.2%	3%	1.4%

3.1. Simulation results

Table 3 and Figures 5:8 show the simulations results and the comparison to the measured data obtained from experimental test. The results are from four case studies at all advance coefficient ranges. The difference between the experimental and simulation data is defined by:

$$E\% = \frac{|EFD - CFD|}{EFD}\%$$

The error of thrust coefficient for using hexahedral mesh type with SST turbulence model decreases reaching its minimum value at 0.8 advance coefficient then increases to reach 7.1% at 1.4. While K_Q error increased from 0.6 to 1 advance coefficient then decreases for 1.2 and 1.4 values of J as shown in figure 5. The efficiency error for the same combination reaches 7% at J=1.2 but has lower values for 0.6 to 1 advance coefficient.

The K-epsilon turbulence model gives lower error values for all coefficient when used with the same hexahedral mesh type. The error values gradually increases for K_T from 3.3% to 7.3% at J=1.4 as shown in figure 6. The torque coefficient error acted the same as using the SST turbulence model but with lower values (4.1%) at small advance coefficients. The tetrahedral mesh type gives adequate results when used with SST turbulence model as the K_T's error raises gradually from 1.3% at J=0.8 to 6.9% at 1.4 advance coefficient as shown in figure 7. The error of efficiency reaches 7.5% at 1.4 advance coefficient which is acceptable for the simulation error. The figure 8 shows the minimum errors of thrust coefficient results for all advance ratios compared to the other cases and also the error increases of torque coefficient from 0.5% at J=0.6 to 4.3% at J=1.2 then decreases to 3.2% at 1.4 advance ratio. Efficiency error uniformly increases by raising the advance coefficient value reaching maximum of 2.3% at J=1.2.

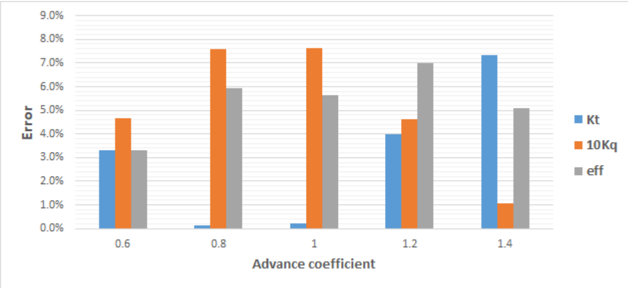

Figure 5 Hexahedral mesh with SST k-omega

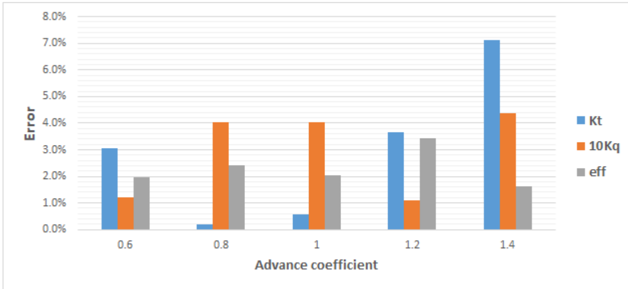

Figure 6 Hexahedral with K-epsilon

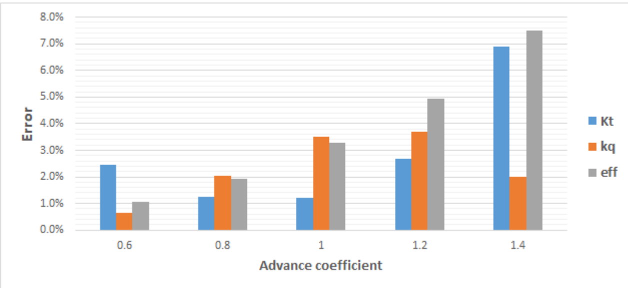

Figure 7 Tetrahedral with SST k-omega

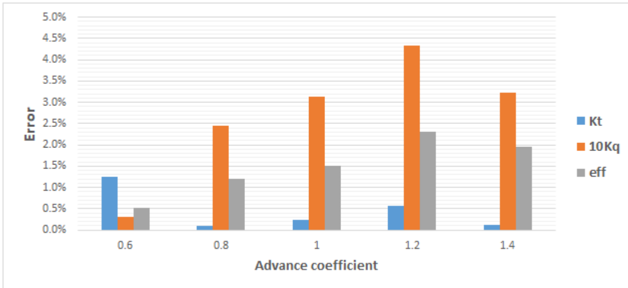

Figure 8 Tetrahedral with k-epsilon

4. Conclusions

In this study, the three-dimensional RANSE method has been used to predict marine propeller open water characteristics. The results were obtained using two mesh types and two turbulence model to perform four case studies, the following conclusions have been made: - Mesh type, mesh density, and turbulence model all influence the accuracy of the propeller hydrodynamics simulation results.

- For this propeller characteristics study, using K-epsilon turbulence model with the same mesh type produces more accurate results with less error compared to using k-ω-SST model.

- Results' discrepancy from the two turbulence models was small compared with that from the two grid types.

- The simulation results is being affected by the type of mesh selected. As using tetrahedral grid in the numerical simulation of open water performance for all ranges of speeds gives better results than those obtained by applying hexahedral grid.

- Results show that mixing tetrahedral mesh type with K-epsilon turbulence model to the numerical simulation for investigating marine propeller open water characteristics gets better quality results compared by those obtained by other cases in this research.

References

[1] Yu and Mishra 2018 Propeller open water characteristics prediction using Reynolds-Averaged Navier–Stokes equations and sliding mesh approach *Ocean Engineering*.
[2] He and McKenna 2014 Open water propeller performance prediction for autonomous underwater vehicles *Ocean Engineering*.
[3] Soong T T, Zijlstra S, 2015A RANS-Based Sliding Mesh Wind Turbine Code Coupled with a User-Defined Function Interface for Loads Prediction *Applied Energy* 150: 298–308.
[4] Alavedra A, Tur A and Sánchez-Soriano 2016 A 3D aerodynamic and acoustic analysis of an unmanned aerial vehicle shrouding.
[5] Chien N 2018 comparison of different approaches for calculation of propeller open water characteristic using RANSE method *Naval Eng.*.

[6] Nakisa M, Abbasi M and A.M. Amini 2010 A Assessment of marine propeller hydrodnamic performance in open water via CFD.

[7] Felicjancik J 2016 Numerical simulations of hydrodynamic open-water characteristics of a ship propeller *Polish Maritime Res.*.

[8] Baltazar J 2015 Numerical studies for verification and validation of open-water propeller RANS computations.

[9] Da-Qing L 2006 Validation of RANS predictions of open water performance of a highly skewed propeller with experiments*Journal Hydrodynam*.

[10] Gmbh S P 2016 ITTC Propeller Benchmark.

[11] ITTC 2014 ITTC – Recommended Procedures and Guidelines ITTC – Recommended Procedures and Guidelines Definition of Variables 1–10.

[12] Bugalski T and Hoffmann P 2011 Numerical Simulation of the Self-Propulsion Model Tests Symp. Mar. Propulsors 1–7.

[13] ITTC 2011 9.1.0-Practical Guidelines for Ship CFD Applications ITTC – Recomm. Proced. Guidel. ITTC 1–8.

[14] Felicjancik J 2016 Numerical simulations of hydrodynamic open-water characteristics of a ship propeller *Polish Maritime Res.* 23 (4).

[15] Dhinesh G, Murali K and Subramanian V 2010 Estimation of hull- propeller interaction of a self-propelling model hull using a RANSE solver *Ships Offshore Struct.* 125–139.