Research Article
Journal of Marine and Engineering Technology (JOINMET) 1(2), 85-97, 2021
Recieved: 14-Jun-2021 Accepted: 28-Dec-2021

SAKARYA UNIVERSITY OF APPLIED SCIENCES

Numerical Investigation of Flow Around a Marine Propeller

Mesut TANER[1*], Aytunç EREK[1]

[1] Dokuz Eylul University, mesut.taner@deu.edu.tr, aytunc.erek@deu.edu.tr

ABSTRACT

This article is aimed to provide a comprehensive understanding about Computational Fluid Dynamics (CFD) analysis of a ship propeller and the validation of the results obtained in the analyses with those of open water tests. The open water tests of the propellers and their outputs are explained briefly. Alternative computational solutions and its importance are also emphasized. A propeller of the modified type of Wageningen B.4 was investigated to validate CFD results.

It is also aimed to focus on cavitation phenomena of this propeller. Nature of cavitation is introduced and the aforesaid propeller is also investigated to visualize the starting and development phases of cavitation. The simulation results of both cavitating and non-cavitating flows are compared with the experimental data.

Keywords: Propeller, propeller characteristics, cavitation, CFD

1 Introduction

Propulsion of a marine vessel can be provided by various devices from simple mono-block fixed pitch propellers to magneto-hydrodynamic pumps. However, propellers are the most preferred propulsion devices among them. An excellent propeller design and performance prediction of this propeller is considered as one of the important design goals in naval architecture. Generally a marine propeller is designed in order to generate the best propulsive efficiency for a ship. However there are some limitations on this design process. Cavitation, for example, is a vital issue since it causes vibration, noise, blade corrosion problem, and especially reduction of the propulsion efficiency. Consequently, cavitation directly impairs service life of a propeller. Besides, the noise and vibration being caused by cavitation are not only a comfort issue, but also a vital problem for war ships and submarines.

In design process of the propellers model tests are needed such as model scale open water tests and full scale behind-hull propeller tests. Full scale behind-hull tests are performed in the open seas and therefore real ships are used in the tests. Thus, the tests are more realistic than the model scale open water ones. Nevertheless, the costs of these tests are very high. The model scale open water tests for propellers are

* Corresponding Author

usually take place in the ocean basins. Hence, they are not time consuming and expensive as much as full scale tests. Nowadays it is possible, the hydrodynamics of the flow around model-scale propellers to be analyse and their results to transform to the full scale (Zhao, 2015).

Cavitation can be defined as a general fluid mechanics phenomenon that can occur whenever a liquid is used in a machine which induces pressure and velocity fluctuations in the fluid (Molland, 2011). The static pressure in an accelerating fluid can drop below the vapour pressure. This "cold boiling" phenomena may happen around a pump blade or a marine propeller. Since the marine propellers accelerate the water flow very rapidly, it becomes an unavoidable problem especially for the propeller blade tips where resulting accelerations are very high. Cavitation not only damages on propeller blades, but also causes vibration, noise and loss of efficiency.

The cavitation is normally defined as the formation of vapour/gas or their mixture and subsequent activities such as growth, collapse and rebound in liquids (Arazgaldi et al., 2009). Cavitation erodes machine elements, deteriorates machine performance, causes noise, vibration and even oscillation of the entire system, and also enhances corrosion/ silt erosion through synergism mechanisms (Zhang et al., 2011). Kinnas et al (2003) used the Boundary Element Method (BEM) for the analysis of sheet cavitation in the non-uniform wake field of marine vehicles. This method also has been developed for the investigation of partial and super cavitation on the back and face surfaces of marine propellers (Arazgaldi et al., 2009).

Open water performance characteristics, velocity field and prediction of cavitation phenomena are investigated by numerous experiments. Computational analysis by using the propeller flow code developed by INSEAN have successfully been performed (Salvatore et al., 2011).

Computational methodology and examples of validation studies are not limited to conventional propellers. Experimental analysis and CFD investigation of Contracted and Loaded Tip (CLT) propellers is also another area that has been focused on. Model tests are performed to obtain the performance characteristics of the propellers as well as their cavity extension for different loads. The numerical and experimental analyses as well as comparison of the results highlight the peculiarities of this kind of propellers give the possibility to increase efficiency and reduce cavitation risk by exploiting the design approaches already well proven for conventional propellers also in the case of these unconventional geometries (Bertetta et al., 2012).

One of the selected propellers in this study is the four bladed INSEAN (Italian Ship Model Basin Instituto Nazionale per Studi ed Esperienze di Architettura Navale) E779A model, which has detailed data for CFD validation. The performance of the E779A model was considered as reference case in many studies (Pereira et al., 2004). The other propeller investigated in present study is three bladed David Taylor Model Basin (DTMB) 4119 model which was examined by applying a finite volume CFD method in conjunction with the standard k-ε turbulence model to calculate the flow pattern and performance parameters (Chang et al., 2010).

2 Theory and Calculation

One of the propellers selected for this study is the E779A INSEAN propeller model, which is the type B.4.70 Wageningen having a constant pitch distribution. The propeller was designed at the end of the 50's for a twin-screw ferry, but unfortunately no full-scale data are available. In the 60's the model of this propeller was chosen as reference propeller model of the Italian Navy Cavitation Tunnel (Istituto Nazionale per Studi ed Esperienze di Architettura Navale Vasca Navale INSEAN, 2006). Since 1997 this propeller model is being used in experimental activity performed by INSEAN. A comprehensive series of experimental data addressing the propeller in a uniform as well as in a non-homogeneous flow is available for this propeller from an experimental programme performed at INSEAN over the last years (Salvatore et al., 2009). Basic geometric characteristics are summarized in Table 1.

Table 1. Dimensions of the propeller of INSEAN E779A model (Salvatore et al,, 2009)

Diameter (mm)	227.27
Blades	4
Pitch Ratio	1.1
Skew Angle at Blade Tips	$4^0 48$ (positive)
Rake (nominal)	$4^0 35$ (forward)
Expanded Area Ratio	0.689
Hub Diameter (mm)	45.53
Hub Length (mm)	68.30

As computational domain for the propeller it was convenient to select a quarter cylinder passage, a quarter cylinder passage selected. In order to consider the interaction with the other three blades, periodic boundary conditions were applied. There are two sub-domains in the main domain. The first one is the stationary domain that surrounds the rotating domain as shown in the Figure 4.2. The other one is the rotating domain which includes propeller wall and the propeller shaft. The static domain has a 7D length and 4D where D denotes the radius of the propeller. The rotating domain has a 4D length and 0.55D radius. The inlet is 4D far from the propeller blades. Outlet has a distance of 3D from same point to the other side. The outer plane of the stationary is called farfield-wall boundary. Figure 1 and Figure 2 show the boundary conditions on the computational domain and grid over the domain.

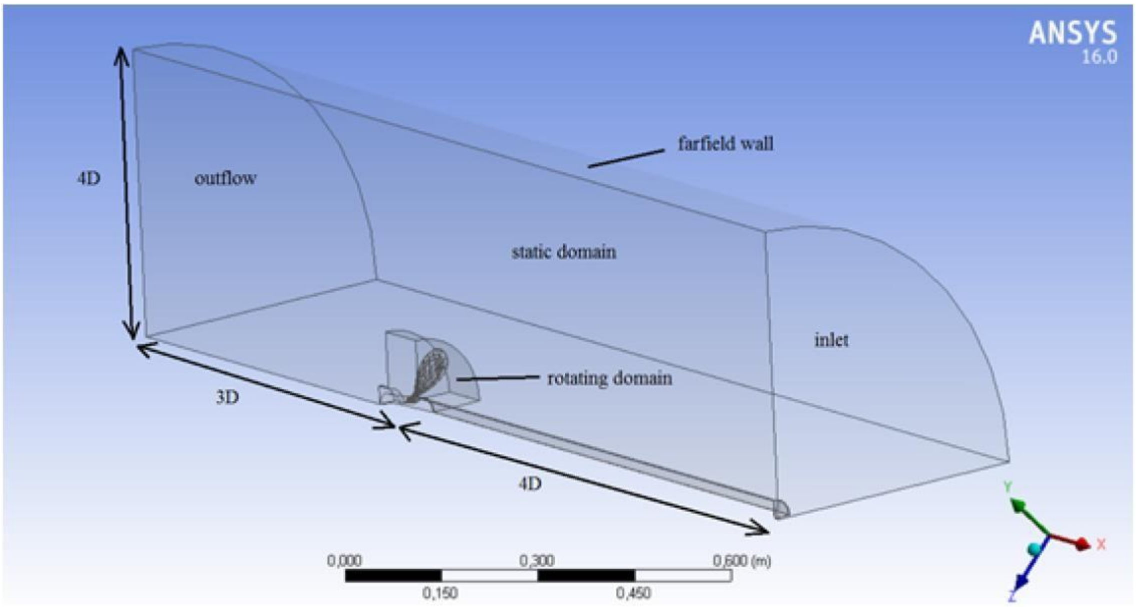

Figure 1. Computational domain investigated and boundary conditions

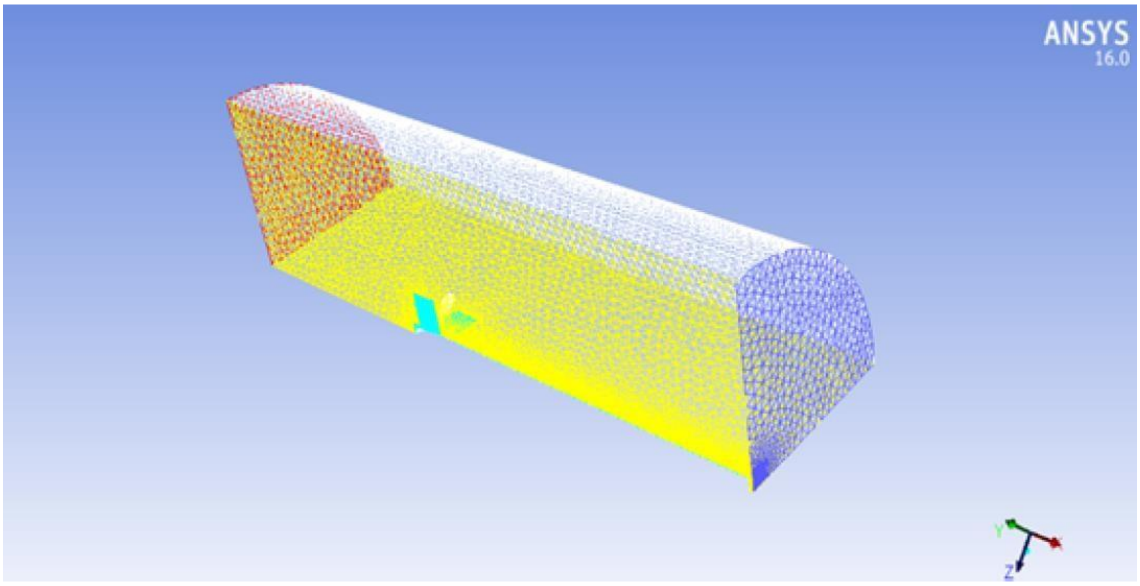

Figure 2. Mesh structure over the domain

The other propeller investigated in this study is DTMB 4119 model. It was designed at United States David Taylor Model Basin (DTMB) and is a three blades submerged noncavitating propeller with a variable pitch. This DTMB 4119 propeller has no rake or skew which means that the propeller has a quite simple geometry. There are sufficient experimental data available for it. The basic geometric characteristics are summarized in Table 2.

Table 2. Dimensions of the propeller of DTMB 4119 model

Blades	3
Diameter (mm)	304.8
Skew	0º
Rake	0º
Blade Section	NACA66 a=8 mean line for all radius

The computational domain for this propeller is a cylinder passage. This domain also consists of two sub-domains. The first one is the stationary domain which surrounds the rotating domain. The other one is the rotating domain which has the blade and hub of the propeller. The static domain has a length of 7D and radius of 4D as shown in the Figure 4.5. The rotating domain has a length of 0.5D and where D denotes the radius of the propeller. The inlet was considered at a distance of 4D from the propeller surface. The outlet is at a distance of 3D from same point to other side. This peripheral plane is called far-field boundary. Figure 3 and Figure 4 show the boundary conditions on the computational domain and grid structure over the domain respectively.

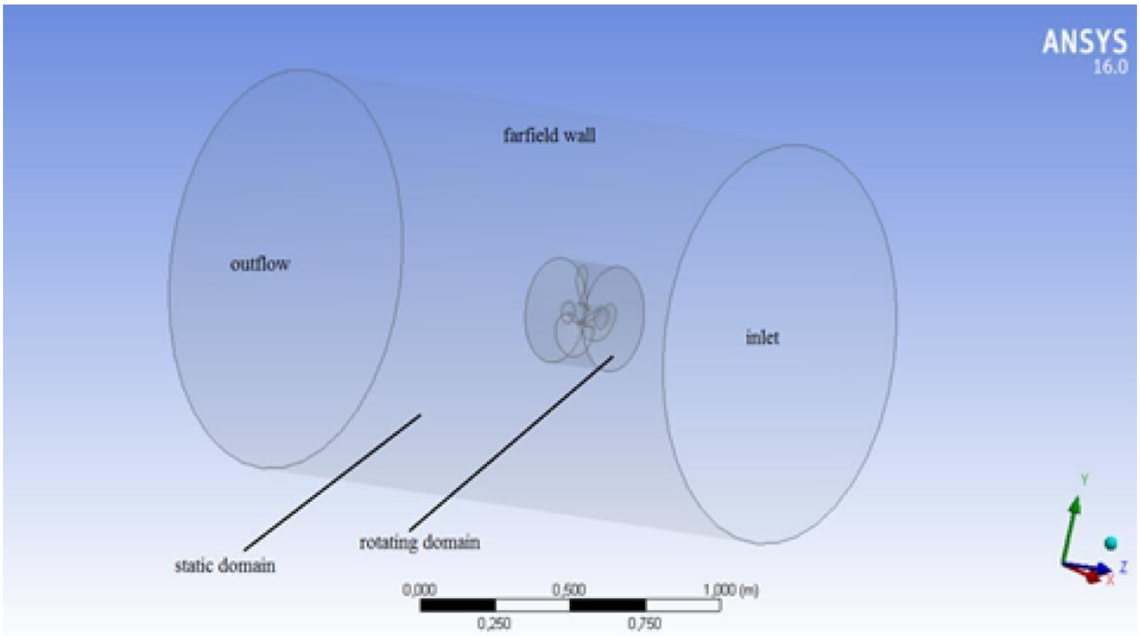

Figure 3. Computational domain investigated and boundary conditions

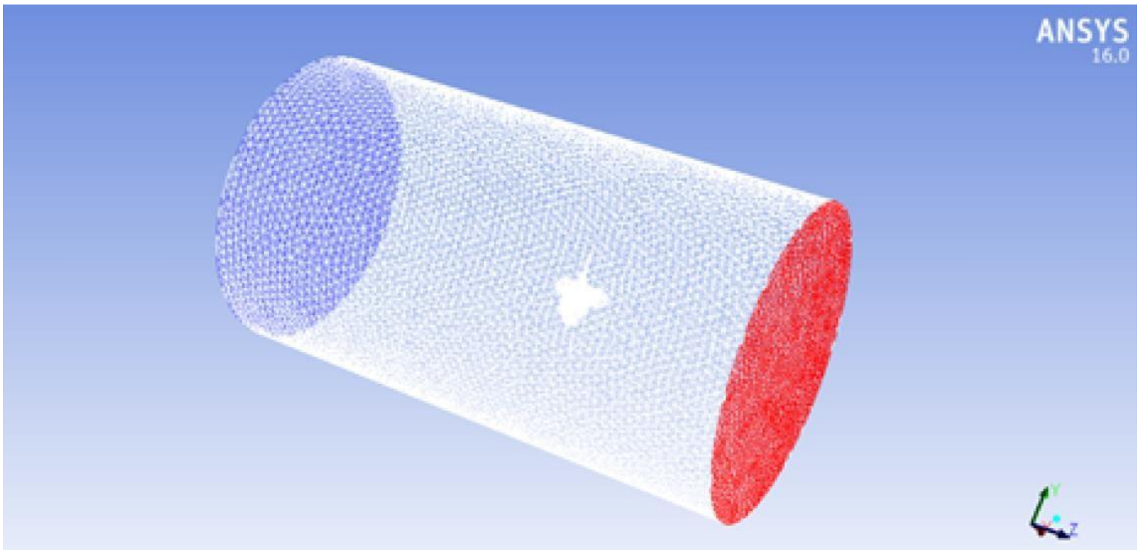

Figure 4. Mesh structure over the domain

The material was chosen as fluid whose properties were selected as water. Moving reference frame method is used for solution. Rotational velocities of the rotating domain were 50 rps, 45 rps, 40 rps, 35 rps, 30 rps and 25rps respectively. The propeller blade defined as wall and has no relative velocity with respect to neighbour meshes. A uniform velocity of 6.22 m/s was defined at the inlet. The far field wall also has no velocity. Table 3 and Table 4 show the details of non cavitating and cavitating flow for the propeller of model E779A respectively.

Table 3. Details of non-cavitating flow of the propeller E779A

Domain Size (m)	Length = 1.6 Diameter = 0.9
Mesh Count	12514342 elements
Viscous Model	$k-\varepsilon$ (2 equations) & k- ω
Near Wall Treatment	Standard Wall functions
Solver	Pressure Based – Steady
Pressure Velocity Coupling - Pressure	SIMPLE - Standard
Discretisation scheme	Quadratic Upwind (QUICK)

Table 4. Details of cavitating flow of the propeller E779A

Propeller Diameter (m)	0.227
Domain Size (m)	Length = 1.6 Diameter = 0.9
Mesh Count	12514342 elements
Mesh Orthogonal Quality	Min: 0.175
Viscous Model	$k-\varepsilon$ (2 equations) & k- ω
$k-\varepsilon$ Model	Realizable
Near Wall Treatment	Standard Wall functions
Solver	Pressure Based – Unsteady
Operating Pressure	100000 Pa
Vapor Pressure	5000 Pa
Pressure Velocity Coupling	SIMPLE
Pressure	Standard
Discretisation scheme for convective fluxes and turbulence parameters	First Order Upwind
Model	Multiphase-Mixture

The material and its properties were selected as in the propeller model E779A. Moving reference frame method is used for solution. Rotational velocities of the rotating domain were 7.5 rps, 9 rps, 11 rps, 13 rps and 15 rps respectively. The propeller blade defined as wall and has no relative velocity with respect to neighbour meshes. A uniform velocity of 6.22 m/s was defined at the inlet. The far field wall also has no velocity. Table 5 shows the details of non cavitating flow for the propeller model DTMB 4119.

The methods for investigating the cavitation phenomena can be categorized into two groups which are single-phase 10odelling and multi-phase 10odelling. In this study, two-phase modelling method was preferred because of its high accuracy and consisting of two phases as water and vapour.

Table 5. Details of non-cavitating flow of the model propeller DTMB 4119 (1)

Propeller type	DTMB 4119
Propeller Diameter (m)	0.3048
Domain Size (m)	Length = 2.1 Diameter = 1.22
Mesh Count	371816 elements
Mesh Orthogonal Quality	Min: 0.241
Viscous Model	$k-\varepsilon$ (2 equations)
$k-\varepsilon$ Model	Realizable
Near Wall Treatment	Standard Wall functions
Solver	Pressure Based – Steady
Pressure Velocity Coupling	SIMPLE
Pressure	Standard
Discretisation scheme for convective fluxes and turbulence parameters	Quadratic Upwind (QUICK)

3 Results and Discussion

The performance of a marine propeller is represented in terms of non-dimensional coefficients. These are thrust coefficient KT, torque coefficient KQ and efficiency as well their variation with advance coefficient J. The computational analyses for the flow around the propellers were carried out using Ansys-Fluent 16.0 software. The thrust and torque were estimated from the computational solutions for different rotational speeds of the propellers.

The estimated thrust and torques for the model E779A are shown in Figure 5. The results indicate that KT and KQ coefficients are decreasing with increasing of the advance coefficients (J) as. Maximum efficiency is observed at J = 0.9133.

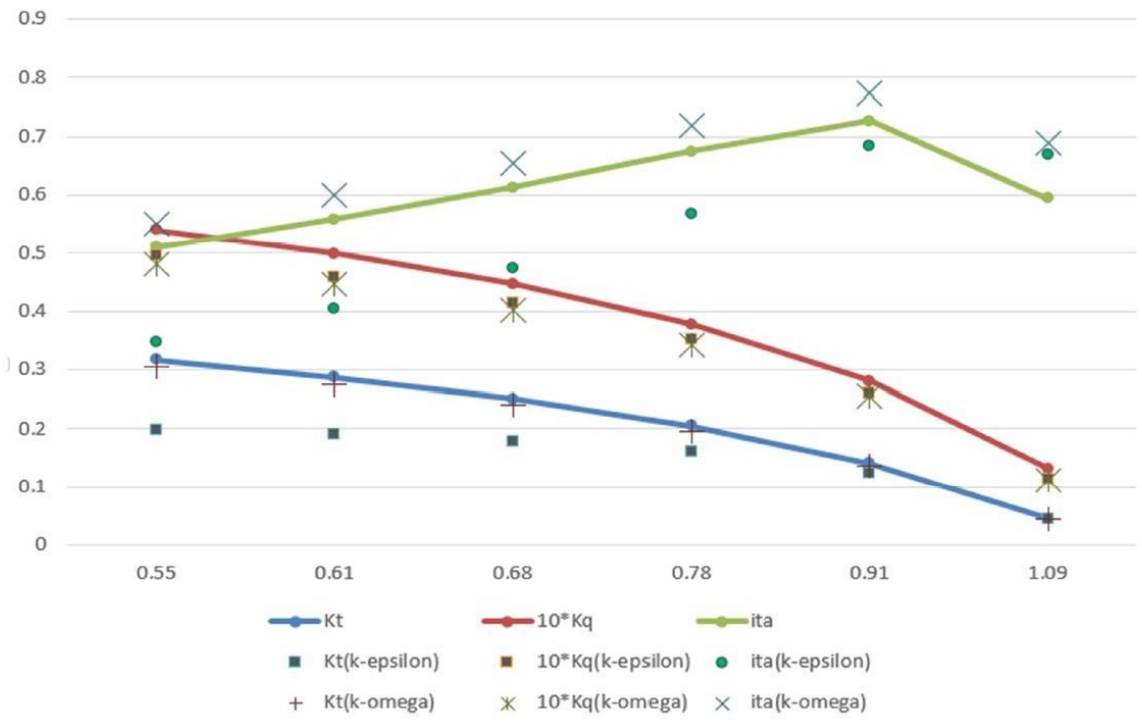

Figure 5. Computed and measured performance of the propeller model E779A in open water

Figure 6 shows the pressure distribution on the surface of the propeller blade at advance velocity 6.22 m/s, rotational speed 50 rps and advance coefficient J=0.548 . The pressure and the suction side are exposed to high pressure and low pressure respectively. However when propeller is operating at low revolution values, the thrust cannot be generated so that a reverse trend in pressure is observed.

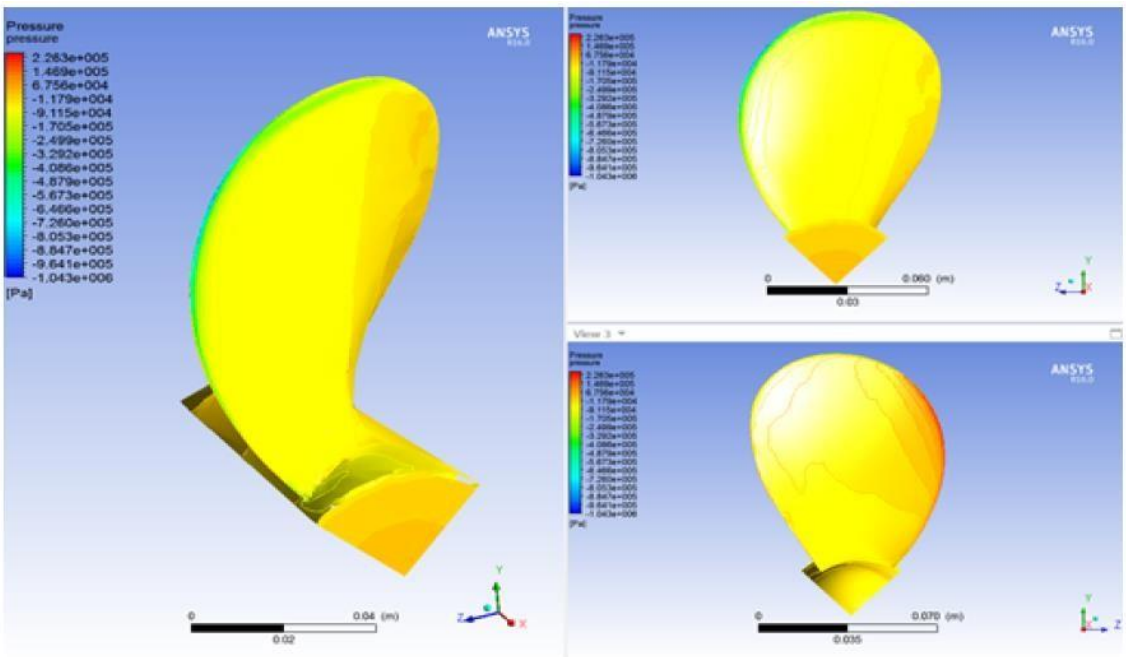

Figure 6. Pressure distribution on the surface of the one blade of E779A at advance velocity 6.22m/s, rotational speed 50 rps

The results show that KT and KQ coefficients are decreasing with increasing of advance coefficients (J). Maximum efficiency is observed at J = 0.9. The comparison of predicted KT and KQ values with experimental data are given in Table 10 and Figure 7. Figure 8 shows the pressure distribution on the surface of DTMB 4119 at advance velocity 2.54m/s, rotational speed 15rps.

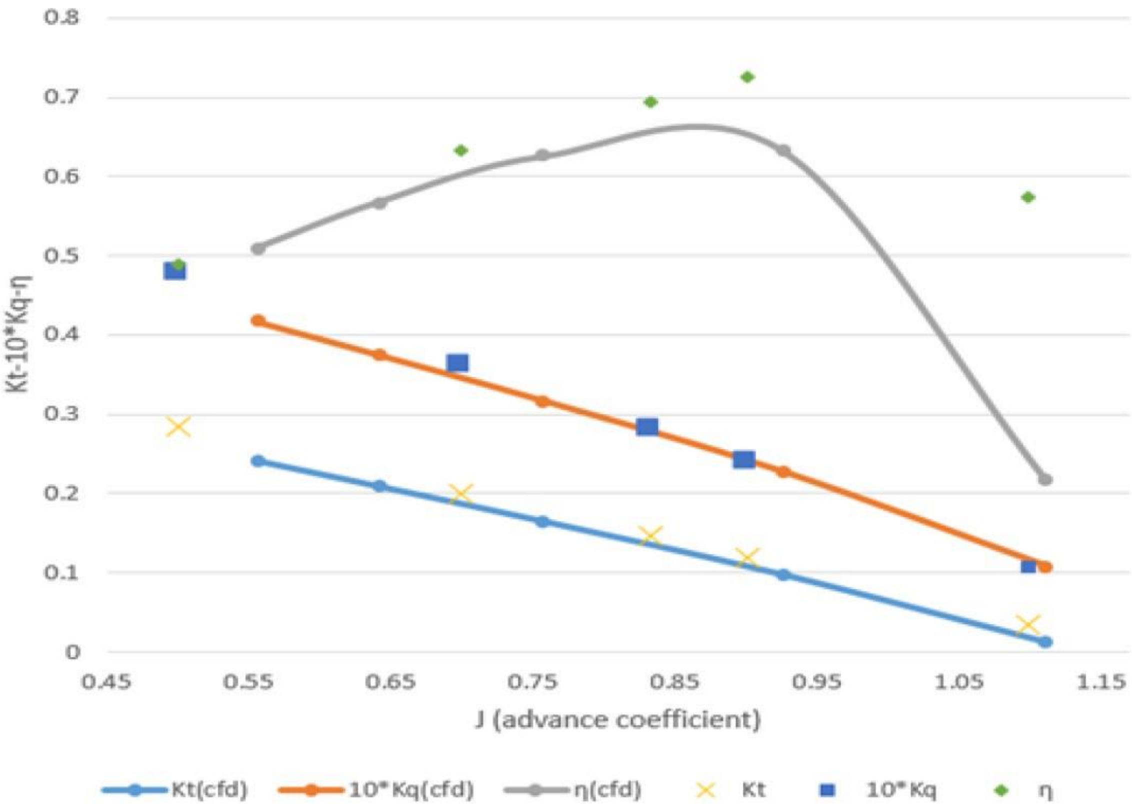

Figure 7. Computed and measured performance of the propeller model DTMB 4119 in open water

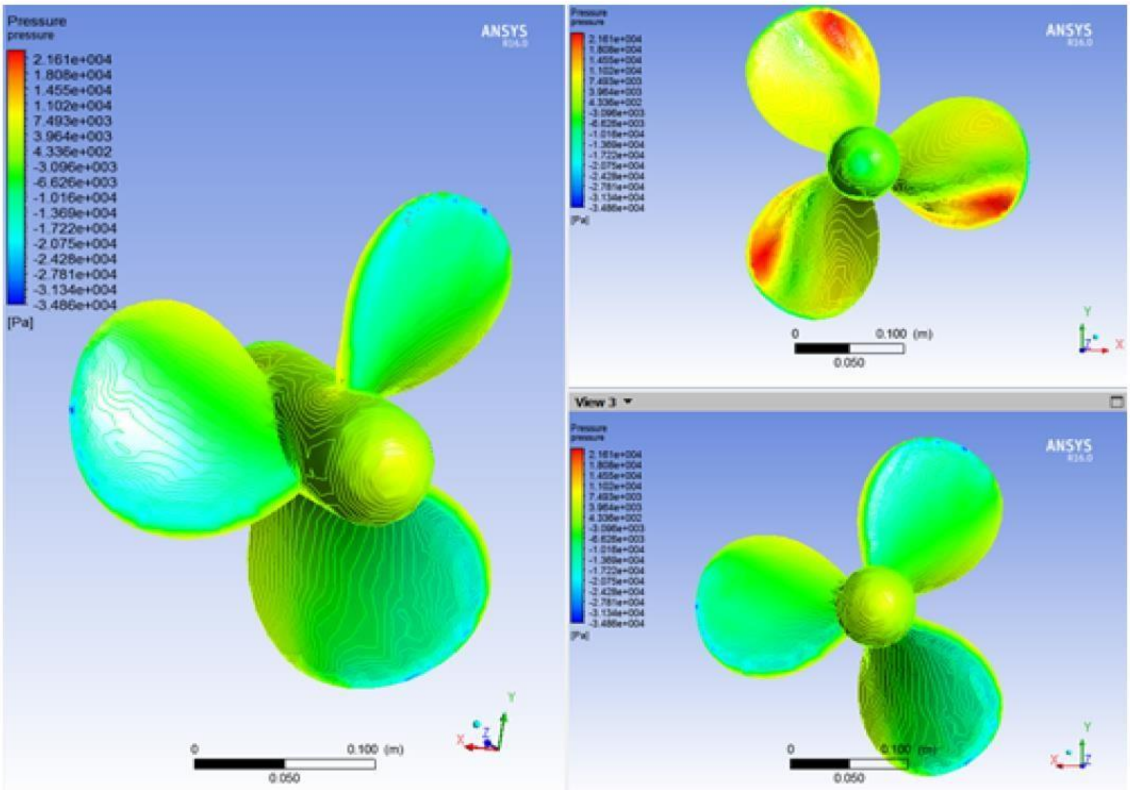

Figure 8. Pressure distribution on the surface of the propeller model DTMB 4119 at advance velocity 2.54 m/s, and rotational speed 15 rps

4 Conclusions

The performances of the two propellers in non-cavitating conditions, and the one propeller, E779A, in cavitating conditions were exhibited. The cavitation number for this cavitating condition is 5.48, and the propeller E779A is marginally cavitating. It has been observed that there was only a small drop in thrust coefficient, while the torque demand was increased slightly.

Numerical predictions indicate a good agreement with experimental observations. The differences are likely to be caused primarily by a lack of mesh density in computational domain. It can be noted that either KT or KQ was a little over predicted, but the most efficient J values did not effected by this difference. KQ values shows more scatter than KT. Differences between all results are about 5% for KT and KQ. Difference in KT are more visible.

Increasing the resolution of the mesh in the area where propeller cavitation occurs has vital importance, because the interface between the cavity and the surrounding water also should be better captured. Cavitation modelling techniques are still a matter of ongoing research activities. Prediction of different cavitation types on propeller blades and propeller hub remains a matter of further research and improvement. The aim is, covering the basic features of cavitating flows without raising the complexity of the simulation and the post processing too much. Practical aspects as working time for mesh generation and CPU time for the computation are not presented in this study. It can be focused in further researches and comparative computations can be presented.

For further studies, the scale effects on open water propellers with different magnitude of skews in the higher pitch ratios should be investigated. This further research can be especially useful for controllable pitch propeller (CPP) design. More research work can focus on the open water performance of the propellers under different rotational speeds. Also off-design conditions like oblique flow is another topic to be work on.

The overall results suggest that the present cavitation model is practicable for simulating cavitating flows around a marine propeller. Compared with previous representative studies, the current results present a better agreement between experiments and numerical simulations.

Authors' Contributions

Mesut Taner: To take responsibility for the conduct of experiments/analyses, to organize and report the data, to take responsibility for the logical explanation and presentation of the findings, to take responsibility for the literature review during the research, to take responsibility for the creation of the entire article or the original section.

Aytunç Erek: Creating ideas or hypotheses for research and/or article, planning materials and methods for conclusions, reworking the article not only in terms of spelling and grammar, but also in terms of intellectual content before submitting the article.

References

Arazgaldi, R., Hajilouy, A., & Farhanieh, B. (2009). Experimental and numerical investigation of marine Propeller cavitation. Mechanical Engineering, 16 (6), 525-533.

Bertetta, D., Brizzolara, S., Gaggero, S., Viviani, M., & Savio, L. (2012). CPP propeller cavitation and noise optimization at different pitches with panel code and validation by cavitation tunnel measurements. Ocean Engineering, 53, 177 195.

Chang, Y., Hu, C., Tu, J., & Chow, Y. (2010). Experimental investigation and numerical prediction of cavitation incurred on propeller surfaces. Journal of Hydrodynamics, 22 (5), 764-769.

Da-Qing, L. (2006). Validation of RANS predictions of open water performance of a highly skewed propeller with experiments. Journal of Hydrodynamics, 18 (3), 520-528.

Dubbioso, G., Muscari, R., & Di Mascio, A. (2013). Analysis of the performances of a marine propeller operating in oblique flow. Computers & Fluids, 75 (20), 86 102.

Kinnas, S. A., Lee, H., & Young, Y.L. (2003). Modeling of unsteady sheet cavitation on marine propeller blades. International Journal of Rotating Machinery, 9, 263-277.

Martio, J., Sipila, T., Sanchez-Caja, A., Saisto, I. & Siikonen, Timo. (2011). Evaluation of the propeller hull vortex using a RANS solver. Second International Symposium on Marine Propulsors, Hamburg University of Technology, Hamburg, Germany, 15-17 June.

Molland, A. F. (2011). The maritime engineering reference book: a guide to ship design, construction and operation. Oxford: Elsevier.

Peng, H., Qiu, W., & Ni, S. (2013). Effect of turbulence models on RANS computation of propeller vortex flow. Ocean Engineering, 72, 304-317.

Salvatore, F., Greco, L., & Calcagni, D. (2011). Computational analysis of marine propeller performance and cavitation by using an inviscid-flow BEM model.

Second International Symposium on Marine Propulsors.

Salvatore, F., Streckwall, H., & Terwisga, T. V. (2009). Propeller cavitation modelling by CFD-results from the VIRTUE 2008 Rome workshop. In Proceedings of the First International Symposium on Marine Propulsors.

Zhang, Q., Zhang, X., Hu, D., & Liu, Y. (2011). Flow simulation in steering rotary valve. Electric Information and Control Engineering (ICEICE) International Conference, 2208-2212.

Zhao, Y. (2015). Investigation in scale effects on propellers with different magnitude of skew by CFD methods. MSc Thesis, Ålesund University College, Ålesund.