Journal of Engineering Science and Technology
8th EURECA 2017 Special Issue August (2018) 28 - 38

STUDYING THE EFFECT OF PITCH RATIO ON SHEET CAVITATION IN MARINE PROPELLERS

MOHAMED ABDOU*, ABDULKAREEM SH. MAHDI AL-OBAIDI

School of Engineering, Taylor's University, Taylor's Lakeside Campus,
No. 1 Jalan Taylor's, 47500, Subang Jaya, Selangor DE, Malaysia
*Corresponding Author: mohamed_ihab96@live.com

Abstract

Cavitation is a phenomenon that occurs when the pressure of a fluid drops below its saturated vapour pressure. Even though cavitation could be used in some applications in a beneficial way, it poses serious adverse effects in the marine propulsion field. The explosion of formed cavitation bubbles in the wake field of the propeller puts large stresses on the propellers blade causing blade erosion in the long term, costing the marine industry millions of dollars annually for maintenance. It also causes vibrations and noise in ships, which for some cases require the propulsion system designers to minimise during the propeller design stages. Non cavitating flow and cavitating flow are studied in the research, using a common CFD tool that uses RANS approach. The effects of altering the pitch ratio on sheet cavitation is studied in cavitating flow using the INSEAN E779A propeller. The study used ANSYS Fluent for the simulation and only one blade is considered in the simulation. Periodic boundary conditions are applied for the 4-bladed propeller. The propeller is simulated in conditions set by the Wageningen worksop and the domain size is based on the cavitation tunnel size used in the measurements of the torque and thrust. The flow considered is uniform and has a speed of 5.808 m/s, propeller rotational speed 36 rps, advance coefficient 0.71, and cavitating flow at cavitation number 1.763. For the cavitating flow, the Zwart-Gerber-Belamri cavitation model along with the k-ε turbulence model are used. Pitch ratios of 0.8, 1.0 and 1.5 are considered and analysed. The results show an increase of the extent of sheet cavitation on the blade surface with an increase in pitch ratio.

Keywords: Cavitation, ANSYS Fluent, Numerical simulation.

1. Introduction

The shipping industry has always had a major role in the world trade, around 90% of the world trade is carried by the international shipping industry. This have had a large impact on the marine industry in terms of technological development and sustainability. This meant producing propulsion systems, which are more economical and have a lesser impact on the environment. This drove propulsion system technologies into greater improvement and the introduction of new technologies, which had a positive impact on performance including lesser use of fossil fuels, less noise and vibration which have had an impact on the marine life and also a better user comfort and safety. The technological surge in computational technologies has helped the marine propulsion industry in this improvement. As numerical simulations have replaced expensive and time-consuming experimental testing with a constant improvement in accuracy and ease of use.

Propulsion efficiency is mainly judged based on the force output considering the work input by the engine. The force output is the thrust generated by the propeller, which is equivalent to the lift force in aerodynamics, and it is generated due to the difference in pressure between the suction side and the pressure side of a propeller blade. In addition to the thrust, torque is also an important factor considered in propulsion systems. Thrust and torque coefficients are calculated using the thrust and torque measurements and they describe the efficiency of a propulsion system.

Therefore, designers seek to increase thrust and torque coefficients while reducing losses in operation caused by friction and cavitation. Cavitation bubbles form during the operation of marine propellers at a high RPM due to a drop in pressure [1]. The drop in pressure puts the liquid under tension, which when reaches a tension high enough it causes nucleation and subsequently bubble formation [2]. These bubbles subsequently collapse and place a large amount of stress on the propeller blades, which has many detrimental effects including noise generation and a drop in performance displayed by a decrease in thrust and torque in addition to vibration. Many factors have a direct effect on cavitation and propeller performance, which makes it a complex area to study. Propeller performance studies were always previously based on tow tank and cavitation tunnels results but due to the high costs of building such facilities and the improvement displayed in computational results the marine industry is moving towards computational simulations which is continuously improving its results accuracy and shortening the time taken in generating the simulation results.

Another factor that is very critical to be considered in propellers is the blade pitch ratio, which is the ratio of the pitch to the diameter of the propeller. As pitch has a direct effect on thrust and torque generation, it is very important for a designer to determine the most suitable pitch ratio in the design stage based on a number of factors.

Controllable pitch propellers are a relatively new propulsion system technology compared to the commonly used fixed pitch propellers that have been in use for hundreds of years. Most ships use fixed pitch propellers but in the recent years, an increase in the use of controllable pitch propellers is seen. It has a number of advantages over fixed pitch propellers including generating a larger torque and thrust. It also enable ships to move both back and forth without a need to change the direction of rotation of the engine, which results in using a smaller engine subsequently resulting in a reduction in cost. It also enables a faster change in ship speed as it stops the need to send a command to the marine engineer on board. Even though the ability to control the pitch of a propeller gives flexibility to the user in terms of thrust and speed control, it is important to understand the effects of change in pitch on the extent of the cavitation on blades, as to make sure the propeller has the longest usage lifespan and maximize rider comfort in ships.

The objective of the study is to find the relationship between pitch ratio and cavitation formation on propellers blades. This will enable controllable pitch propellers designers to find the optimal range of the pitch ratio for the propellers. Possibly increasing usage life and enhancing propellers performance. Three different pitch ratios were considered and analysed; 0.8, 1.1 and 1.5. Prior to studying the cavitating flow of the propellers, non cavitating flow was simulated and compared to the experimental results to ensure accurate simulations.

2. Research Methodology

ANSYS Fluent CFD software was used in the simulations presented in the following paper. The INSEAN E779A propeller model was used the study, and only one blade was considered for the simulation. The case presented is a low Reynold number turbulence, cavitating flow. The *k-ε* turbulence model was used along with the Zwart Gembali cavitation model.

2.1. Geometry

The INSEAN E779A propeller was selected as reference for this study. Due to the wide availability of tests and research performed at INSEAN over the last decade. Description of the INSEAN E779A experimental data is done by Pereira et al. [3] in early 2004 [3] and one later in the same year [4].

The INSEAN E779A propeller is a four blade, right handed, fixed pitch propeller with a diameter of 0.227 m and a fairly small rake and skew angle. It has a constant pitch ratio of 1.1 along the blade diameter. Figure 1 shows the INSEAN E779A propeller. The propeller specifications are listed in Table 1.

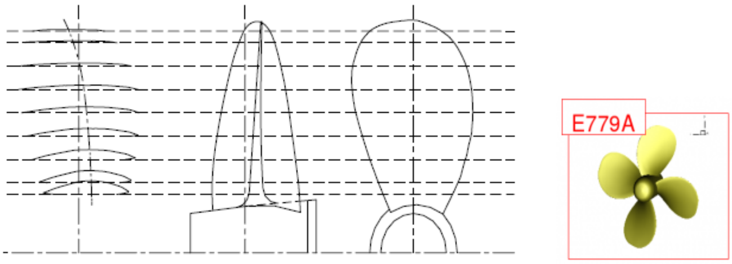


Fig. 1. INSEAN E799A propeller [3].

Table 1. INSEAN E779A propeller specifications [3].

	Specification
Number of Blades	4
Pitch over Diameter Ratio, p/D	1.1
Skew	4.5
A_p/A_0	0.69
Diameter of propeller (mm)	227.27

Prior to the simulations, the geometry of the propeller was modified. These modifications included simplifications, drawing of the fluid domain and changing the pitch of the propeller. In order to reduce the computational duration and resources only one blade was considered in the simulation and periodic boundary conditions were applied. The domain size was based on the domain size proposed in the Wageningen Workshop in 2007 [5]. The ideal tunnel size that is used as the fluid domain in the simulation is shown in Fig. 2, where R_{tun} =1.471 D and D is the diameter of the propeller.

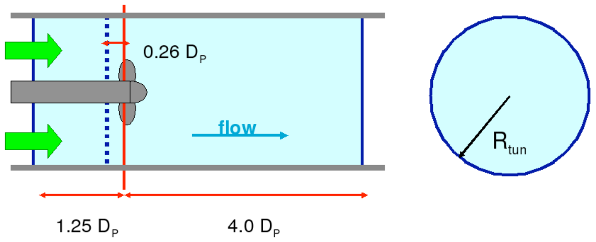


Fig. 2. Idealized tunnel size [6].

2.2. Mesh

Since only one blade was used in the simulation, the domain cylinder was divided by 4 and the simulated domain was only a 90° section. This reduced the element count by 4 times. An unstructured grid was used for the simulation using the default mesh interface for ANSYS Fluent. The domain consisted approximately of 700000 elements compared to more than 2 million elements in a full cylindrical domain. The elements had an average skewness of 0.28 with a maximum skewness value kept below 0.85 in order to ensure accurate results generation. The mesh mainly consists of tetrahedrons with a small percentage of wedges used in the inflation around the blade surface. The inflation was applied around the blade in order to capture the boundary flows more accurately around the blade tip. Ten layers of inflation was used with a growth rate of 1.4. In addition to that proximity and curvature, based meshing was used to automatically set the element size around hard curves and thin edges. The mesh of the blade and domain is shown in Fig. 3.

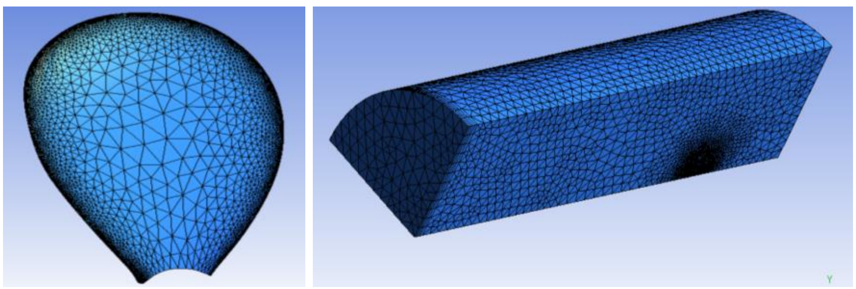


Fig. 3. Mesh for blade and domain.

2.3. Numerical setup

Initially non cavitating flow is considered and compared to the benchmark results database of the INSEAN E799A propeller. After the results had an acceptable accuracy comparing it to the experimental results, cavitating flow was considered and the vapor fraction on the blade surface was shown.

For a given propeller advance coefficient, J, defined in Eq. (1)

$$J = \frac{U}{nD} \tag{1}$$

where U is the advance speed, n is the rotational speed of the propeller, and D is the diameter of the propeller.

A specific cavitating flow was applied according to the flow cavitation number defined in Eq. (2)

$$\sigma_n = \frac{P - P_V}{0.5\rho_L(nD)^2} \tag{2}$$

where P is the outlet pressure, P_V is the vapour pressure and ρ_L is the density of the liquid.

The flows considered is uniform and has a speed of U = 5.808 m/s and propeller rotational speed n = 36 rps, propeller advance coefficient J = 0.71; cavitating flow at cavitation number σ_n = 1.763;

The flow is considered a steady turbulent flow. Realizable k-ε turbulence model was used, with standard wall function which is a common turbulence model shown to give accurate results for cavitating flows along with the k-ω turbulence model. A turbulence intensity of 2% and a turbulence viscosity ratio of 10 is used in the computation which were also specified in the Wageningen Workshop along with the tunnel size. Zwart gembali cavitation model was used in calculating the cavitating flow as it was shown to give reliable results in a study by Heinke [7].

The factors considered were the thrust and torque generated by the propeller. The thrust and torque coefficients were calculated using Eqs. (3) and (4) respectively and compared with the experimental data.

$$K_T = \frac{T}{\rho_L n^2 D^4} \tag{3}$$

$$K_Q = \frac{Q}{\rho_L n^2 D^5} \tag{4}$$

where T is thrust, and Q is the calculated torque.

For the non cavitating flow the solver used was SIMPLE, with the standard pressure solver and the second order upwind for the turbulence factors and momentum. As for the cavitating flow the solver used was COUPLED, with PRESTO pressure solver and second order upwind for turbulence and momentum. Due to the quick nature of cavitation formation, a very small time step had to be used for the simulation or else divergence occurs. After multiple trials with the most suitable time step, a time step of 0.00002 was used, as a time step larger than that caused a divergence.

2.4. Parametric study

Following the verification of the accuracy of the flow using both cavitating and non cavitating flow. The effect of pitch on cavitation on the propeller blade surface was considered. The effect of pitch on thrust and torque generation have been thoroughly studied in the past decade and their relationship is shown in Fig. 4.

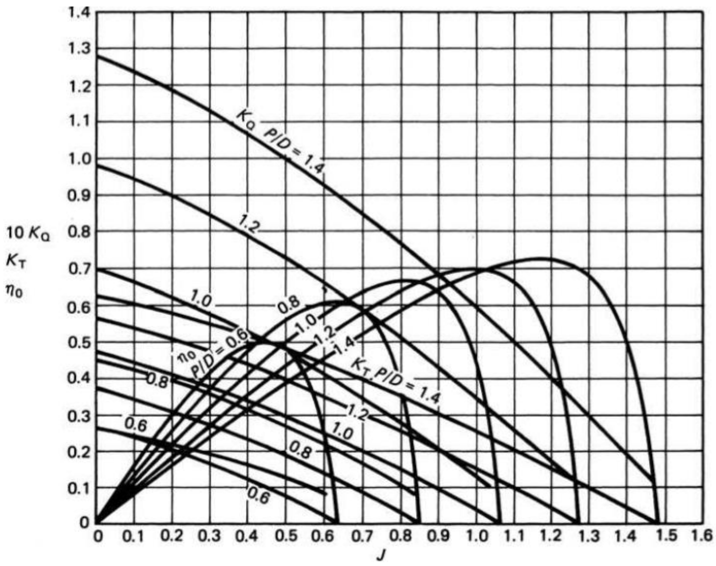
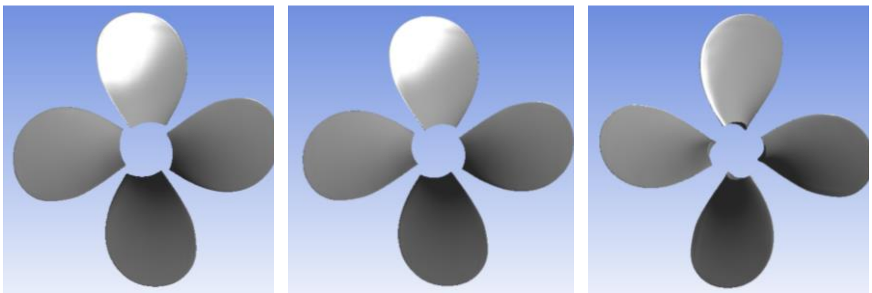


Fig. 4. Open water diagram for Wageningen propeller [8].

It is seen that with an increase in pitch ratio, there is an increase in both thrust and torque coefficients and subsequently an increase in torque and thrust. However, on the other hand, the extent of the formation of cavitation bubbles on the blade surface is not well studied and therefore the detrimental effects of cavitation on the propeller at higher pitch ratios is not known. With the increase in use of controllable pitch propellers, it is necessary to know the optimal operating pitch ratios for both thrust and torque generation along with the minimal cavitation in order to enhance the operation of a propulsion system.

Three different pitch ratios are considered in the current study; 0.8, 1.1 and 1.5, as shown in Fig. 5. The thrust and torque coefficients are calculated for the three cases and the extent of the cavitation on the blade surface is visually observed.



***p*/*D* = 0.8** ***p*/*D* = 1.1** ***p*/*D* = 1.5**

Fig. 5. Pitch ratios used in the study.

Additional factors could be considered in the study in the future including varying the rake and skew angles and studying their effect on thrust and torque generation along with cavitation formation.

3. Results and Discussion

3.1. Uniform non cavitating flow

In order to ensure correct simulation setup and accurate results a uniform non cavitating flow was done prior to completing the parametric study. The flow conditions specified in the Wageningen Workshop was applied and results were compared to the results published by Subhas [9]. Results also published by Watanabe [10] are used for reference.

Only one blade was modeled in a 90^o periodic domain, in order to reduce the computational time. In order to view the four blades, 4 graphical instances were set up to complete the 4 blades of the propeller. The pressure contours of the blade surface suction and pressure sides were set up and the torque and thrust generated were calculated using ANSYS CFD POST. Subsequently the thrust and torque coefficients were calculated and compared to the experimental results. The pressure contours of the 3 pitch ratios of 0.8, 1.0 and 1.5 are shown in Figs. 6.

As shown by the pressure contours results, as the pitch ratio increases, there is an increase in pressure on the blade surfaces. This is seen in the pressure distribution on the blade. At a pitch ratio of 0.8 and 1.1 it is seen that the pressure is highest towards the tip of the blade, which slightly moves inwards towards the blade surface with an increase in the pitch ratio. This is due to the increase in area of the blade surface that is in contact with the fluid. This increase in pressure on the pressure side creates an increase in the generated thrust and torque as calculated in Table 2.

Table 2. INSEAN E779A propeller specifications.

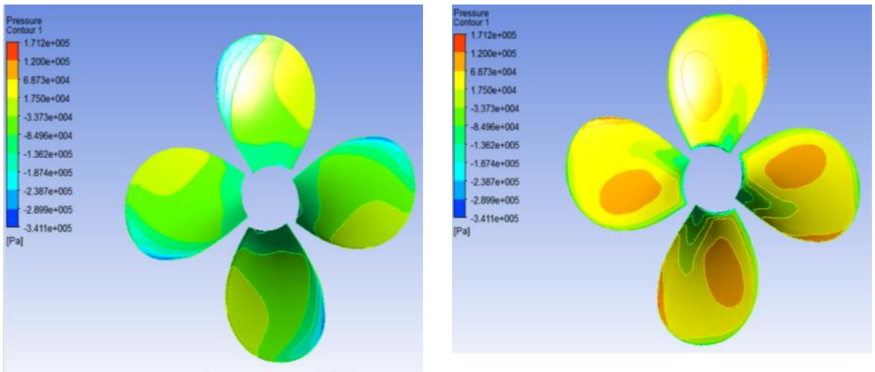
Pitch Ratio	Thrust (N)	Torque (N.m)
0.8	612.31	20.82
1.1	760.48	34.67
1.5	832.66	43.73

The thrust and torque coefficients were then calculated and compared to the experimental results of the INSEAN E779A propeller which has a pitch ratio of 1.1. The results are listed in Table 3.

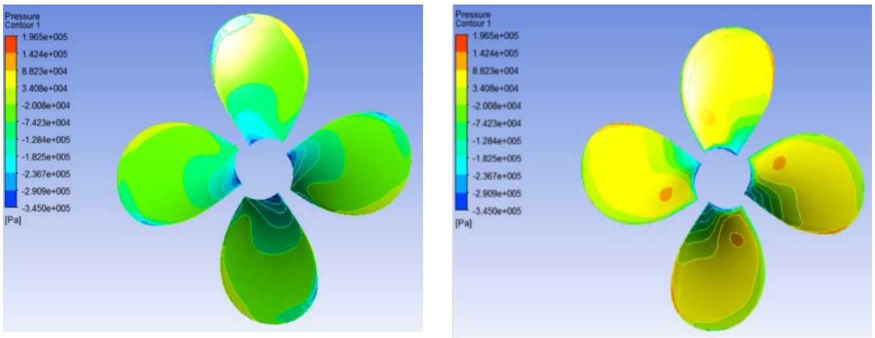
Table 3. Comparison between calculated and experimental K_T and K_Q values.

Coefficients	Measured (tunnel)	Calculated (Simulation)	Percentage difference (%)
K_T	0.256	0.221	13.67
K_Q	0.464	0.445	4.09

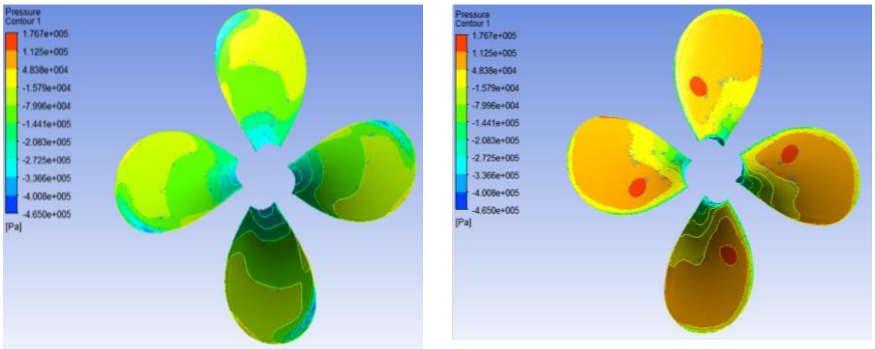
Suction side **Pressure side**



Pitch Ratio = 0.8



Pitch Ratio = 1.0



Pitch Ratio = 1.5

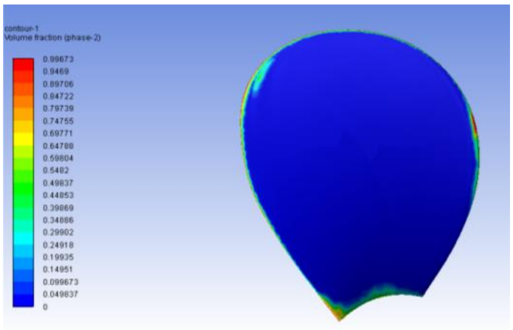
Fig. 6 Pressure contours of different pitch ratios.

3.2. Cavitating flow

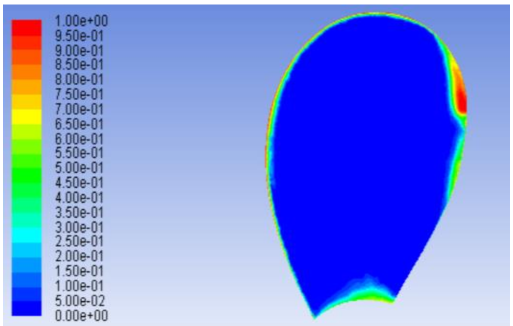
Uniform Cavitating flow using the same conditions as the non cavitating flow was simulated in order to find the relationship between pitch ratio and cavitation area on the blade surface. Three pitch ratios were used including a pitch ratio of 0.8, 1.1 and 1.5. The vapor formation on the blade surface is shown in Fig. 7.

The results in Fig. 7 show that as the pitch ratio increases, sheet cavitation on the blade surface increases. Which could lead to adding large loads on the blade surface leading to blade erosion in the long term usage.

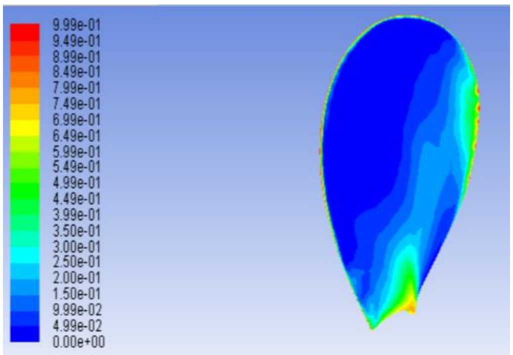
Despite the positive effects, the pitch ratio has on torque and thrust generation, it also increases deteriorative effects such as cavitation and blade contact area with the water which increases friction and causes losses in the work input, therefore causing a drop in propulsion efficiency. Therefore, in order for a propeller designers to choose an optimum propeller pitch, those number of factors have to be taken into consideration.



Pitch Ratio = 0.8



Pitch Ratio = 1.0



Pitch Ratio = 1.5

Fig. 7. Cavitation formation on the blade with different pressure ratios.

4. Conclusions

The effect the pitch ratio has on sheet cavitation in marine propellers was studied. Both non cavitating and cavitating flow were simulated using the same conditions previously set in the Wageningen in 2007. The pitch ratios that were used in the study were 0.8, 1.0 and 1.5. Initially the non cavitating flow was used to ensure the accuracy of the simulations and the torque and thrust for the different pitch ratios were measured using ANSYS Fluent. The thrust and torque coefficients were then calculated and the results showed a difference between measured and calculated torque and thrust coefficients that were within acceptable range.

The cavitating flow used the same pitch ratios used in the study, with the Zwart Gerber Gembali cavitation model used in the simulation. The results of the cavitation on the blade surface showed that as the pitch ratio increased the cavitation formation on the blade increased significantly.

Nomenclatures

D	Propeller diameter, m
J	Propeller advance coefficient
K_Q	Torque coefficient
K_T	Thrust coefficient
n	Propeller rotational speed, rps
P	Outlet pressure, Pa
p	Pitch, m
p/D	Pitch over diameter ratio
P_V	Vapour pressure, Pa
Q	Torque, N.m
T	Thrust, N
U	Advance speed, m/s
V	Flow velocity, m/s

Greek Symbols

σ_n	Cavitation number.
ρ_L	Density of the liquid, kg/m^3

References

1. Brennen, C.E. (1995). *Cavitation and bubble dynamics*. New York: Oxford University Press.
2. Brennen, C.E. (2011). *An introduction to cavitation fundamentals*. WIMRC Forum, University of Warwick, UK.
3. Salvatore, F.; Pereira, F.; Felli, M.; Calcagni, D.; and Di Felice, F. (2006). Description of the INSEAN E779A propeller experimental dataset. *Technical Report INSEAN* 206-085, Rome, Italy.
4. Pereira, F.; Salvatore, F.; Di Felice, F.; and Soave M. (2004). Experimental investigation of a cavitating propeller in non-uniform inflow. *25th ONR Symposium on Naval Hydrodynamics*, St. John's, Newfoundland, Canada.

5. Streckwall, H.; and Salvatore, F. (2008). Results from the Wageningen 2007 workshop on propeller open water calculations including cavitation. *Proceedings of the RINA Symposium on CFD Models*, London, UK.
6. Salvatore F.; Streckwall H.; and van Terwisga, T. (2009). Propeller cavitation modelling by CFD - Results from the VIRTUE 2008 Rome workshop. *First International Symposium on Marine Propulsors*, Trondheim, Norway.
7. Heinke, H.-J. (2011). Potsdam Propeller Test Case (PPTC). Cavitation tests with the model propeller VP1304. Schiffbau-Versuchsanstalt, Potsdam GmbH, *Report* 3753.
8. Carlton, J. (2007). *Marine propellers and propulsion*. (2nd Ed.) Burlington: Butterworth-Heinemann.
9. Subhas S.; Saji, V.F.; Ramakrishna, S.; and Das, H.N. (2012). CFD analysis of a propeller flow and cavitation. *International Journal of Computer Applications,* 55(16), 26-33.
10. Watanabe, T.; Kawamura, T.; Takekoshi, Y.; Maeda, M.; and Rhee, S.H. (2003). Simulation of steady and unsteady cavitation on a marine propeller using a RANS CFD code. *Fifth International Symposium on Cavitation* (*CAV2003*), Osaka, Japan.