COMPARISON BETWEEN CFD AND OPEN WATER TESTS FOR PPTC PROPELLER IN NON-CAVITATING CONDITIONS

Kiril Todorov

Institute of Metal Science, Equipment and Technologies with Hydro- and Aerodynamics Centre "Acad. A. Balevski", Bulgarian Academy of Sciences, 1, William Froude St., 9003 Varna, Bulgaria, e-mail: k.todorov@bshc.bg

Abstract. The present paper considers the capabilities of the open-source software for computational UID dynamics calculations Open Field Operation and Manipulation (OpenFOAM) to calculate the characteristics of a modern propeller design. With the results, engineers can make modifications to the design in order to improve propeller performance.

Keywords: marine, propeller, performance, OpenFOAM, Computational fluid dynamic.

1. INTRODUCTION

In the recent years a lot of attention has been given to marine pollution. One of the types of pollution is the emissions from ships' engines. One of the ways to lower those emissions is by designing the ship's propeller to be as efficient as possible.

With the increase in computing power of modern CPUs, the CFD application in this area has been increasing.

The propeller used in this study is one of the officially accepted test cases for validation purposes – PPTC (Potsdam Propeller Test Case). The propeller was developed by Schiffbau-Versuchsanstalt Potsdam (SVA Potsdam) [1]. They have provided data for different cases such as open water characteristics, cavitation tests and more.

In the paper, the software used for performing the CFD calculations is the open-source OpenFOAM (Open Field Operation and Manipulation).

http://doi.org/10.7546/EngSci.LXI.24.01.02

2. OPENFOAM SOFTWARE

OpenFOAM is an open-source software that has been developed for some years now. Unlike other CFD software, this one has no Graphical user interface, but instead has a programming one. This leads to the user learning how to program the simulations by editing and creating the files needed for the CFD simulations execution. Although it is difficult to learn at first and to get used to, at the same time this offers the chance to prepare the next simulation while the calculations are being performed. The package has its own modules for mesh generation – blockMesh and snappyHexMesh, many solvers and an array of turbulence models. The OpenFOAM software does not have an integrated post-processing application but has been designed to work perfectly with ParaView, which is installed when installing OpenFOAM [2].

3. PROPELLER DESCRIPTION

As mentioned above the propeller used for this article is the PPTC. The propeller was designed back in the year 1998 and later on used by different workshops for validation purposes. The propeller's main particulars are shown in Table 1. The 3D model is shown in Fig. 1.

Table 1. Main particulars of the PPTC propeller model

Diameter	D	[m]	0.250
Pitch ratio $r/R = 0.7$	$P_{0.7}/D$	[–]	1.635
Area ratio	A_E/A_0	[–]	0.77896
Chord length $r/R = 0.7$	$C_{0.7}$	[m]	0.10417
Skew	θ_{EXT}	[°]	18.837
Hub ratio	d_h/D	[–]	0.300
Number of blades	Z	[–]	5
Sense of rotation		[–]	Right
Type		[–]	Controllable pitch propeller

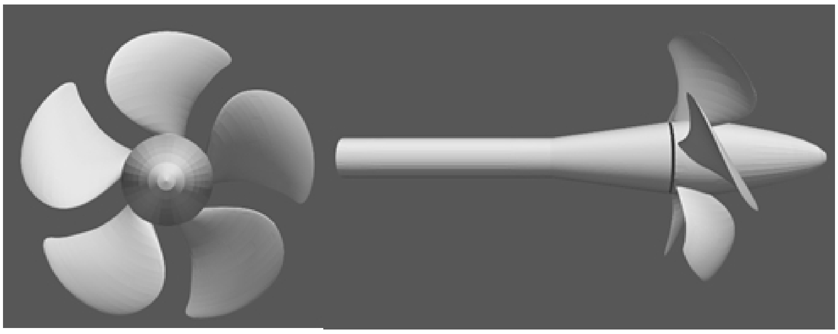

Fig. 1. View of the 3D model from the front and side

4. MESH DESCRIPTION AND COMPUTATIONAL SET-UP

4.1. Mesh description

For the simulations that were carried out the model of the propeller was scaled to diameter of 0.200 m.

The mesh domain is a cylinder with a diameter of 0.6 m and a length of 1 m. The total cell count is about 500000 cells, meaning that the mesh can be considered as relatively coarse. The domain was generated using blockMesh module of the software.

In order to simulate the rotation of the propeller it was needed to use the so-called CyclicAMI (Arbitrary Mesh Interface), which is useful when cyclic movements such as rotation. For that, an inner cylinder is created which is slightly larger than the propeller and extends forward and aft of the blade section of the propeller.

On Fig. 2 and 3 visual representations of the mesh are shown.

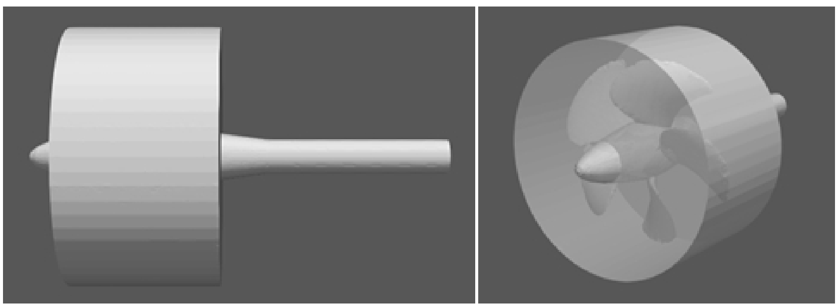

Fig. 2. Side view of the inner cylinder part of the mesh and position of the propeller inside it

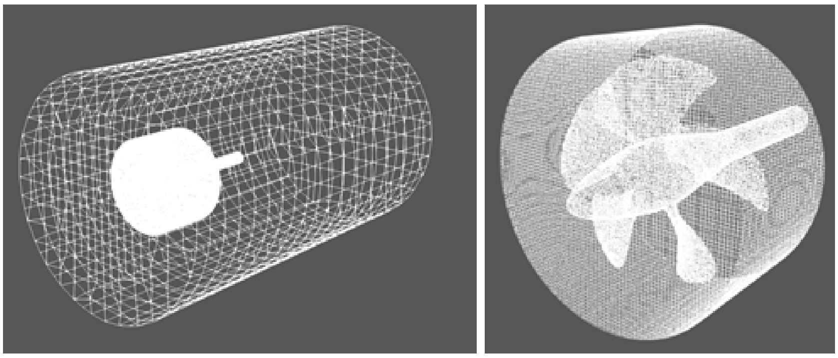

Fig. 3. Cell distribution inside the domain and around the propeller

4.2. Computational set-up

For this paper, the computational set-up is as follows:
- The turbulence model used was kEpsilon "$k - \varepsilon$". The model equations are as follows [3, 4]:
  - ∘ The turbulent kinetic energy equation, k [3] [Eq. 2.2-2, [4]]:

$$\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho D k \nabla k) + P - \rho\varepsilon \tag{1}$$

where
k – turbulent kinetic energy [$\mathrm{m^2 s^{-2}}$];
D_k – effective diffusivity for k [-];
P – turbulent kinetic energy production rate [$\mathrm{m^2 s^{-3}}$];
ε – turbulent kinetic energy dissipation rate [$\mathrm{m^2 s^{-3}}$];
ρ – mass density of the fluid [kg/m^3].
∘ The turbulent kinetic energy dissipation rate equation, ε [3][Eq. 2.2-1, [4]]:

$$\frac{D}{Dt}(\rho\varepsilon) = \nabla \cdot (\rho D\varepsilon\nabla\varepsilon) + \frac{C1\varepsilon}{k}(P + C3\frac{2}{3}k\nabla \cdot u) - C2\rho\frac{\varepsilon^2}{k} \tag{2}$$

where
D_ε – effective diffusivity for ε [–];
$C1$ – model coefficient [–];
$C2$ – model coefficient [–];
$C3$ – model coefficient [–];
ρ – mass density of the fluid [kg/m^3].
- The divergence schemes are chosen as follows:
  - ∘ for U – Gauss linearUpwind grad(U);

- ○ for k – Gauss upwind;
- ○ for ε – Gauss upwind.

– The solvers for the pressure, U, k and ε are as follows:
- ○ for the pressure – GAMG – Geometric agglomerated algebraic multigrid solver with smoother DICGaussSeidel and tolerance of 1*e-5;
- ○ for U, k and ε – Smoothsolver with smoother symGaussSeidel and tolerance of 1*e-6.

– The $y+$ values for the propeller are: minimum is 36.6864 maximum is 588.264, giving us an average of 134.838.

NUMERICAL RESULTS

The open water tests' data are taken from [1]. In order to compare between the open water tests and the CFD calculation the thrust and torque are used to calculate the thrust coefficient K_T and the torque coefficient K_Q. Also, the advance coefficient J and the propeller efficiency η_0 had to be calculated. The formulas used are:

$$J = \frac{V}{n.D} \tag{3}$$

$$K_T = \frac{T}{\rho.n^2.D^4} \tag{4}$$

$$K_Q = \frac{Q}{\rho.n^2.D^5} \tag{5}$$

$$\eta_0 = \frac{J}{2\pi}.\frac{K_T}{K_Q}. \tag{6}$$

Definition of the variables in the equations:

– ρ – mass density of the fluid [kg/m^3];

– n – rate of revolutions [1/s];

– D – diameter of the propeller [m];

– V – velocity of the fluid [m/s].

Since the test in open water were conducted for two different numbers of revolutions ($n = 10$ s^{-1} and 15 s^{-1}) the CFD calculations were done for the same revolutions of the propeller. The results are shown on Fig. 4 and 5.

5.1. Results for $n = 10\ s^{-1}$

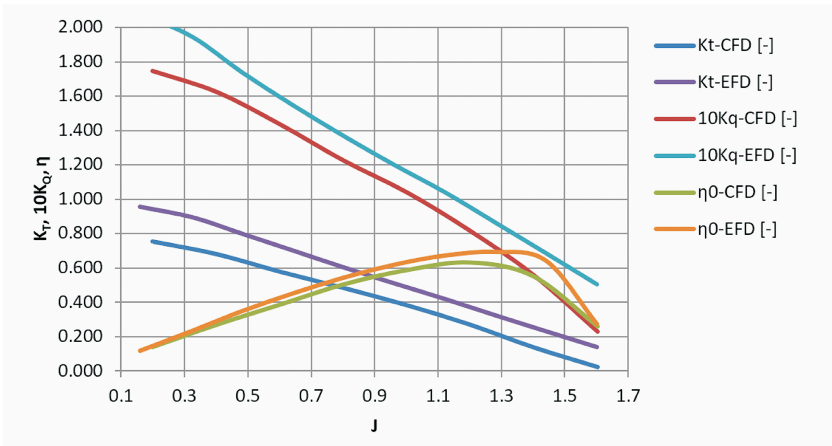

Fig. 4. Chart representation of the calculated coefficients for open water test and CFD calculations

5.2. Results for $n = 15\ s^{-1}$

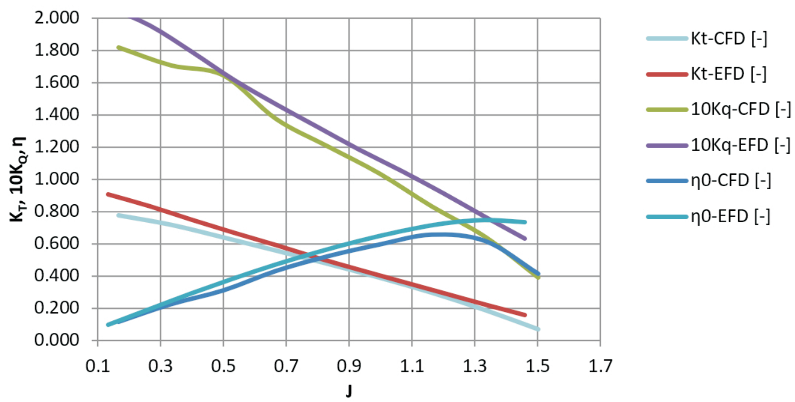

Fig. 5. Chart representation of the calculated coefficients for open water test and CFD calculations

On Fig. 4 and 5 it can be seen that the CFD results are close to the open water test results. It can be noted that even though the curves are mostly parallel, at the higher advance coefficient the propeller efficiency calculated from the CFD results appears lower than the test results. This can be due to the mesh not being the right one for the higher velocities at these advanced coefficients.

In order to be understood why there is a larger difference between the CFD results and the open water test results at the $n = 10\ \mathrm{s}^{-1}$ cases compared to the $n = 15\ \mathrm{s}^{-1}$ cases a more comprehensive study on the mesh must be made. This may be due to the mesh being more compatible with the higher revolutions' cases and for the lower a finer mesh may be needed. This can be done in a future continuation of the study.

6. 3D GRAPHIC RESULTS OF THE SIMULATIONS

The 3D visualizations of the CFD calculations are for $J = 1.3$ at which the propeller in both cases $n = 10\ \mathrm{s}^{-1}$ and $n = 15\ \mathrm{s}^{-1}$ has near maximum efficiency.

6.1. Results for $n = 10\ \mathrm{s}^{-1}$ revolutions

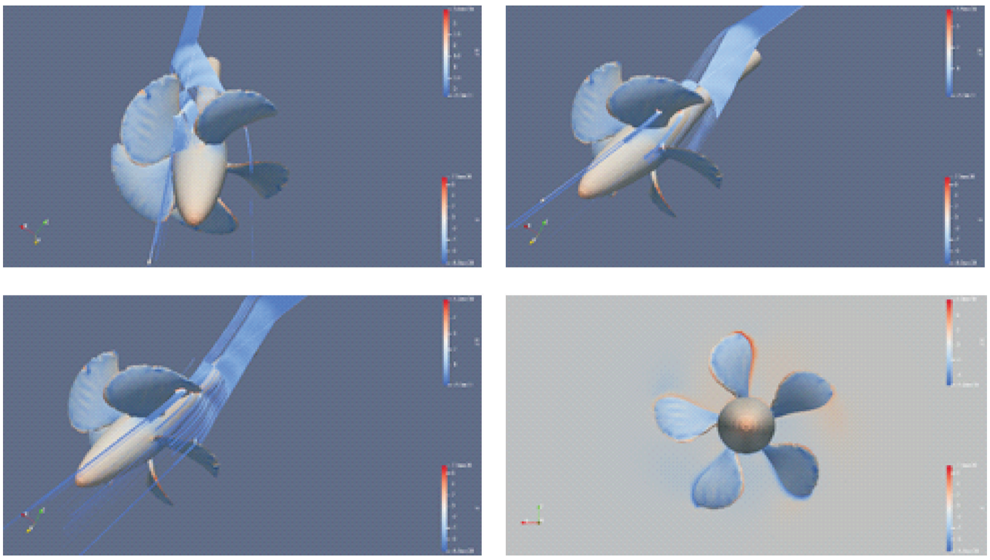

Fig. 6. Different views of the propeller where the pressure distribution on the blades is shown and streamlines have been added

6.2. Results for $n = 15$ s^{-1}revolutions

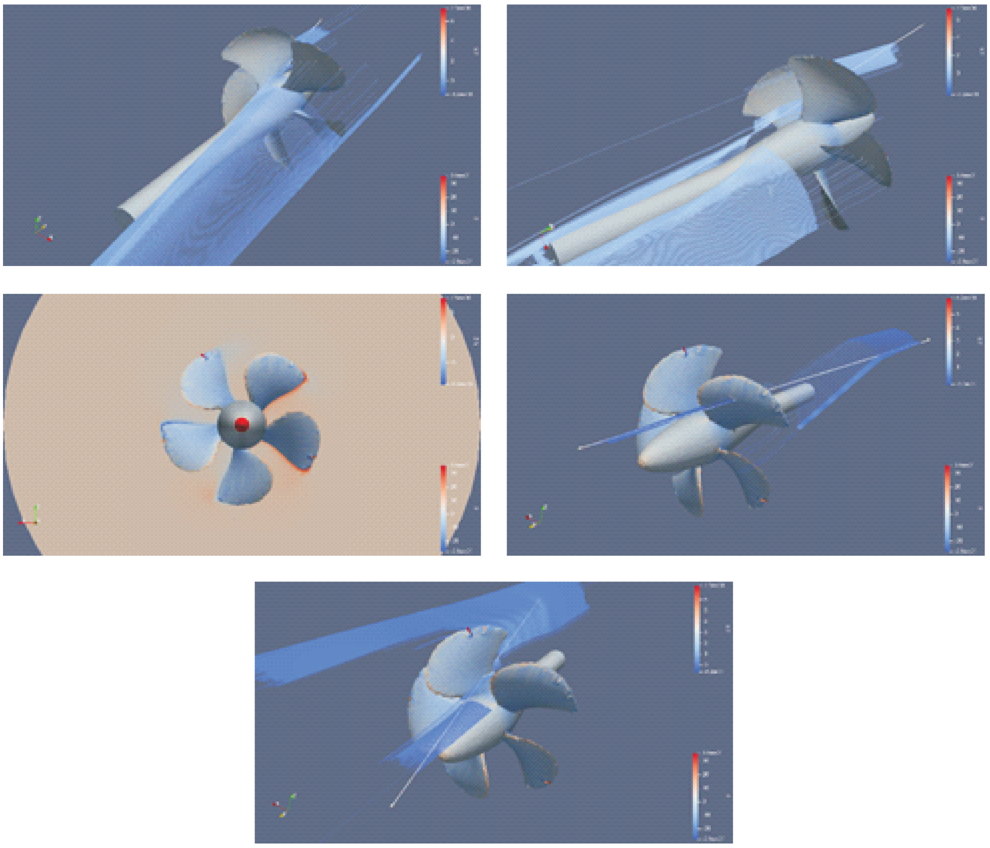

Fig. 7. Different views of the propeller where the pressure distribution on the blades is shown and streamlines have been added

7. CONCLUSION

While open water tests and cavitation tunnel tests are still the most reliable to determine the propeller performance, the CFD methods are catching up. The presented results show that even a coarse mesh can provide good results to predict the propeller performance during the design process and thus to become a cost-effective method for optimization.

Further CFD calculations with more refined meshes will determine how the density of the mesh affects the results. Also, calculations concerning cavitation should be done to determine how well the software handles these cases.

REFERENCES

[1] U. H. Barkman, Potsdam Propeller Test Case (PPTC), Open Water Tests with the Model Propeller VP1304, Report 3752, Potsdam, April 2011.

[2] OpenFOAM and The OpenFOAM Foundation, online: https://openfoam.org/.

[3] k-epsilon, in: OpenFOAM: User Guide v2112, The open source CFD toolbox, online: https://www.openfoam.com/documentation/guides/latest/doc/guide-turbulence-ras-k-epsilon.html.

[4] B. E. Launder and D. B. Spalding, The numerical computation of turbulent flows, in: Computer methods in applied mechanics and engineering, (1974) **3** (2) 269–289.

Received January 23, 2024